Exterior water flow simulation

Erik Nilvé and Daniel Persson

DIVISION OF PRODUCT DEVELOPMENT | DIVISION OF ENERGY SCIENCES |
DEPARTMENT OF DESIGN SCIENCES | DEPARTMENT OF ENERGY SCIENCES |
FACULTY OF ENGINEERING LTH | LUND UNIVERSITY
2018

MASTER THESIS

Exterior water flow simulation

A simulation study of a car to provide design guidelines for future cars

Erik Nilvé and Daniel Persson

Exterior water flow simulation

A simulation study to provide design guidelines for future cars

Published by
Department of Design Sciences
Faculty of Engineering LTH, Lund University
P.O. Box 118, SE-221 00 Lund, Sweden

Subject: Product Development (MMKM05) Energy Sciences (MVKM01)
Division: Product Development and Fluid Mechanics
Supervisor: Axel Nordin and Johan Revstedt
Examiner: Damien Motte and Christoffer Norberg

Abstract

The Water Tightness group at Volvo Cars is responsible for creating and maintaining requirements and thresholds regarding water flows on a car. This is performed by doing extensive physical testing while making sure that databases and previous requirements are kept up to date.

Previously the big focus for the Water Tightness department has been physical testing as there were no other options. During the last years, CFD (computational fluid mechanics) has had a swift development and consequently the group has made the decision to investigate whether there is an opportunity to apply it in their line of work.

Initially the project was only supposed to include the CFD software Star-CCM+. For the first few weeks the setup and settings in this program were developed for each area of the car. Halfway through the project a new software called PreonLab was introduced. This is a somewhat simplified version of certain functions within Star-CCM+. A new objective was then created within the project to investigate the advantages and disadvantages of each software and compare them to each other.

With this in mind, this thesis is a study not only in finding and studying different water flow phenomenon on a car, but also to get an understanding of whether CDF is possible to apply in these specific cases.

Keywords: CFD, Fluid Mechanics, Product Development, Simulation, Water Flow

Sammanfattning

Water Tightness är en avdelningsgrupp på Volvo Cars som är ansvariga för att skapa och underhålla kravspecifikationer för vattenflöde på en bil. Detta genomförs via omfattande fysiska tester som sedan ställs mot uppdaterade krav baserat på tidigare resultat.

Tidigare har fokuset hos Water Tightness legat på att genomföra fysiska tester då det inte fanns några andra alternativ,men på senare år har CFD haft en snabb utveckling och följaktligen har gruppen beslutat att undersöka vilka möjligheter som finns genom att tillämpa denna arbetsmetodik.

Till en början var detta projekt endast tänkt att inkludera CFD programvaran Star-CCM+. De första veckorna gick åt att testa och optimera inställningar och arbetsgång i denna programvara. Halvvägs genom projektet introduceras en ny programvara vid namn PreonLab. Denna programvara kan anses vara en förenklad version av vissa funktioner i Star-CCM+. Ett nytt delmål blev då att undersöka för- och nackdelarna mellan båda dessa programvaror och jämföra dem med varandra.

Denna avhandling kommer inte endast att bestå av att hitta och studera olika vattenflöden på olika delar av bilen. Utan den kommer även bidra till en förståelse om vilka möjligheter CFD kan bidra med för respektive fall.

Nyckelord: CFD, vatten, personbil, produktutveckling, strömningslära

Acknowledgments

This master thesis has mainly been carried out at Volvo Cars, Gothenburg, Torslanda during the spring of 2018.

We would like to thank Ganesh Vattigunta, our supervisor at Volvo and Daniel Carlsson who is the manager at the department, they have supported us through our process.

We are also grateful to all the people at the Water Tightness department at Volvo Cars, who has been helpful with insight of the working area.

We would also like to thank Mani Johannesson who is a support engineer, working at SIEMENS. His expertise has been helpful during discussions related to the software STAR CCM+.

Furthermore, we would like to thank Wolfgang Schwarz who is working for FIFTY 2 and is the one that provided us with the software PreonLab. He’s always been quick to respond to any questions, no matter the complexity.

Finally, we would like to thank Axel Nordin and Johan Revstedt our supervisors from Lund´s University for their support along the project.

Lund, June 2018

Erik Nilvé and Daniel Persson

Table of contents

List of acronyms and abbreviations

CFD	Computational Fluid Mechanics
CAD	Computer Aided Design
SCCM	STAR-CCM+
CAE	Computational Aided Engineering

1 Introduction

1.1 Using the template

1.1.1 Background

The car industry has long been highly dependent on time consuming and expensive physical testing to establish important parameters. These parameters could relate to any imaginable function in the car. Not only is the physical testing very demanding on its own, it can also create problems further down the production chain. Imagine finding a crucial error in the car design at a very late stage in the production chain. You are now forced to either risk making a change or to start over from a previous point.

This is where simulation comes in. During the last years there has been an exploding interest in all possible kinds of simulations. Not only is it very useful in the obvious cases with wind and water flows, it is also used for simulating more abstract things like production chains and manufacturing. There are many advantages to this approach. The cost of testing facilities and equipment could be severely reduced since most of the testing will now take place in a virtual environment. The testing that might take a group of people in the laboratory can now be performed by a single person. It is obvious that there is a lot to gain from developing a functioning simulation system.

However, there are not only advantages to simulation. At this point in time the simulations are oftentimes not very accurate. A simulation engineer might be quite content with a simulation that correlates with reality at a percentage of under 80% which might seem weak. This means that currently there is no way you could rely solely on simulations to draw conclusions regarding the development process.

1.1.2 Purpose

The main purpose of this report is to create simulations and templates intended for four different parts of a Volvo car. These parts include the front door, the rear door, the trunk and the drainage system of the sunroof. For this master thesis the car that has been provided by Volvo Cars is a four-door vehicle with one openable tailgate and sunroof, which will be studied and simulated. For the front door and rear door

Volvo has found that if water enters the area between the door and the car body and its cold outside, the doors get stuck once the water freezes. The simulations will provide a way to judge whether water enters this area or not. For the trunk, Volvo has seen that water sometimes enters the luggage area of the trunk when the trunk is opened with water resting on it. The simulations will create the opportunity to find whether the design results in water entering the luggage area or not. For the sunroof there are requirements of what water flow rate the drainage pipes should be able to handle. This can also be judged by performing simulations.

As simulation work is often hindered and limited by computational power, the car will not be simulated in its entirety. Instead, the car will be divided into the cases of the different parts. This way the full project can be handled in a much more efficient way than if the car CAD (Computer Aided Design) model was directly imported into the simulation software.

Currently, the department where the work was made uses an engineer who is working in synergy with several departments to create these simulations. This has the advantage of having people specialized in their different areas, such as CAD, meshing, Star-CCM+ (SCCM) and PreonLab providing excellent solutions to the problems. SCCM is an advanced CFD software where the user can simulate a real environment and study different fluid flows that are applied. PreonLab is a simplified CFD software that are based on physics that make it faster but not as customizable as SCCM. As of right now the process from CAD file to final simulation is a time-consuming process. With this report it should be possible to speed up the process by providing user templates for the future employees of the Water Tightness Department. The temples are intended for employees with a good insight into CAD, PreonLab and SCCM. The purpose of the template is to guide the user with values and important information of the case. These user templates will be text files with added pictures to create detailed instructions and make the simulations performed in this project easy to replicate.

1.1.3 Limitations

This project comes with some limitations regarding the different stages of the simulation process. Since this project is more focused on the development process that includes the full process from full CAD file to finished simulation file. This results in some limitations for the study of the SCCM software. This is an incredibly powerful program that has an infinite number of different alternatives. In a previous project, some of the more basic ones have been thoroughly studied and the conclusions of that project will lay the base for the initial parameters in these Volvo simulations. For the meshing, there are a lot of alternatives when deciding on which software to use. Some programs might be more advantageous than the other but, in this project, the built-in meshing function of SCCM will be used for simplicity. The CAD work will be done in Catia as this is what Volvo uses and the CAD part of

SCCM does not seem to be developed to do any advanced work but only to prepare some models that are very close to finished.

The templates are written to employees with a good knowledge of both SCCM and CATIA. It is assumed that the employee can handle the program independently, and only needs the value to set up the simulation.

1.2 Theoretical framework

Siemens has written an excellent documentation for SCCM which makes it possible to find information about every function in the program. Sadly, the FIFTY2 user manual has very little information about the underlying physics and is mostly focused on user interface. Because of this the physics will not be explained in greater detail in this chapter, but some general information regarding the particle physics will be presented later in the report.

1.2.1 **General CFD Concepts**

1.2.1.1 Continuity equation

The first governing equation of is called the continuity equation (Tu, Yeoh, & Liu, 2012). This equation is derived from the fact that mass can neither be created or destroyed.

The requirement for a flow to be considered incompressible is that the highest velocity within the flow does not surpass Ma = 0.3. The surrounding pressure at sea level means that a Mach number of 0.3 results in a velocity of about 100 m/s. Seeing as this is a level of velocity that will likely not be reached, it should be safe to consider the flow incompressible. The continuity equation can be written, in differential form, as Equation (1):

$$\frac{\partial(u)}{\partial x} + \frac{\partial(v)}{\partial y} + \frac{\partial(w)}{\partial z} = 0 \tag{1}$$

1.2.1.2 Momentum equation

Through Newton’s second law one can establish that the sum of all the forces on a fluid element equals the mass times the acceleration (Tu, Yeoh, & Liu, 2012). To show this, the x-direction will be considered. Newton’s second law states shows in Equation (2).

$$\sum F_x = ma_x \tag{2}$$

The acceleration a_x is simply given by deriving the velocity in the x-direction, Equation (3).

$$a_x = \frac{Du}{Dt} \tag{3}$$

The mass for a given fluid element is $\rho\Delta x\Delta y\Delta z$ and this gives the resulting increase in x-momentum, expressed in Equation (4).

$$\rho\frac{Du}{Dt}\Delta x\Delta y\Delta z \tag{4}$$

The forces described in the Equation (2) consist of body forces and surface forces. The body forces can be divided into gravity, centrifugal, Coriolis and electromagnetic forces. The surface forces depend on the normal and tangential stresses of the different surfaces. By summing all the body forces together into a single component, one ends up with the following expression for the x-momentum, study Equation (5):

$$\rho\frac{Du}{Dt} = \frac{\partial\sigma_{xx}}{\partial x} + \frac{\partial\tau_{yx}}{\partial y} + \frac{\partial\tau_{zx}}{\partial z} + \sum F_x^{body\ forces} \tag{5}$$

Where σ is the tensile stress and τ is the shear stress. The momentum equation in the y- and z-direction are derived in the same manner.

1.2.1.3 K-Epsilon turbulence modeling

Since the model chosen for these simulations take turbulence into account it is necessary to decide what turbulence model to use. In this case, the K-Epsilon model has been chosen (Launder & Sharma, 1974). This has become the most widely used turbulence model in industrial applications.

For the K-Epsilon turbulence model, the signature variables have to be explained. The k is the turbulent kinetic energy and can be described in Cartesian tensor notation as Equation (6).

$$k = \frac{1}{2}u_i'u_i' \tag{6}$$

And the ε is the dissipation rate of the turbulent energy and can be described as Equation (7).

$$\varepsilon = v_T\overline{(\frac{\partial u_i'}{\partial x_j})(\frac{\partial u_i'}{\partial x_j})} \tag{7}$$

Where i,j = 1,2,3.

A local turbulent viscosity, μ_T, can be described by the local k and ε values as Equation (8).

$$\mu_T = \frac{C_\mu \rho k^2}{\varepsilon} \tag{8}$$

And the eddy viscosity v_T can then be described as Equation (9).

$$v_T = \frac{\mu_T}{\rho} \tag{9}$$

And it is with the addition of these variables to the governing equations that the random behavior of turbulence is taken into account in the simulations.

1.2.1.4 Reynold-averaged Navier-stokes

As the engineering world is oftentimes content with an approximate solution if it means that a lot of time can be saved, the Reynold-averaged Navier-stokes (RANS) (Tu, Yeoh, & Liu, 2012) process of obtaining solutions to the time-averaged governing equations. This means that by utilizing the fact that certain parameters can be averaged over time rather than obtained at instantaneous time, the solutions can be simplified and the time to solve is consequently severely reduced. This means that the continuity equation with the time-averaged values which shows in Equation (10).

$$\frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} = 0 \tag{10}$$

Results in the following Equation (11).

$$\frac{\partial \bar{T}}{\partial x} + \frac{\partial (\bar{u}\bar{T})}{\partial x} + \frac{\partial (\bar{v}\bar{T})}{\partial y} = \frac{\partial}{\partial x}\left(\frac{k}{\rho C_p}\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(\frac{k}{\rho C_p}\frac{\partial T}{\partial y}\right) - [\frac{\partial \overline{u'T'}}{\partial x} + \frac{\partial \overline{v'T'}}{\partial y}] \tag{11}$$

Where $\bar{u}$, $\bar{v}$ and $\bar{T}$ are mean values and u', v' and T' are turbulent fluctuations. The term $\frac{k}{\rho C_p}$ in Equation (11) is the thermal diffusivity α of the fluid.

1.2.2 **Star-CCM specific implementations**

1.2.2.1 Convective CFL Time-Step Control

In some cases, it proves difficult to find a time-step that is high enough to allow for reasonable simulation times while keeping the calculations from diverging. For some simulations there might be points in time where there is heavy flow in areas with a mesh structure that results in easily diverging calculations while the rest of the simulation does not run into any problems. For these simulations there is an optional physics model that calculates a reasonable time-step for each iteration (Siemens, 2018). This model is based on Equation 12:

$$\Delta\tau = \min(\frac{CFL\ V(x)}{\lambda_{max}(x)}, \frac{VNN \Delta x^2(x)}{\nu(x)}) \tag{12}$$

Where CFL is the dimensionless Courant Number specified by the user, V is the cell volume, VNN is the Von Neumann number, Δx is a characteristic cell length scale, $\nu(x)$ is the kinematic viscosity and λ_{max} is a value calculated from the velocity and the speed of sound.

1.2.2.2 Eulerian multiphase

Flows in fluid dynamics can consist of either a single phase or several. Examples of single phase flows are water flowing in a pipe and air flowing over a wing. Many industrial applications, however, cannot be described by single phases. Some examples of multiphase flows could be air bubbles in a glass of water, sand particles being blown around in the wind or rain drops forming in air. In multiphase flows additional complexity is added to the simulation, particularly due to the interface between different phases where the difference in phase properties must be taken into consideration in mass, momentum and heat equations. When doing simulations of multiphase flows, it is of importance to classify the spatial scales of the interface between the different phases (Siemens, 2018). Figure 1.1 below shows a scale from dispersed into separated flows.

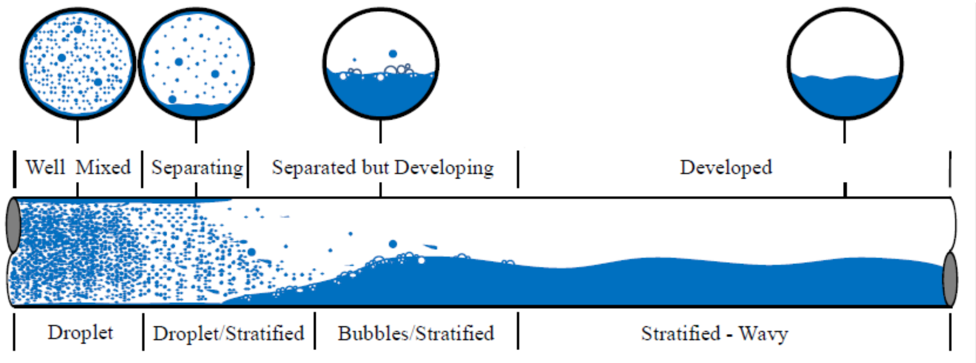


Figure 1.1 Different kind of flows

1.2.2.3 Exact wall distance

This function computes the wall distance form centrum of a cell to the nearest wall with a non-slip condition (Siemens, 2018). Non-slip condition means that the geometry surface is assumed to have some friction. To calculate the exact distance, the calculation is based on a triangulation of the surface mesh. This calculation takes place in the very first steps of the simulation, to decide where the cells are located in relation to the geometry. With the knowledge of the distance, other models can use this value to calculate turbulence and wall treatment. There are two ways that SCCM can calculate the distance. These are: exact wall distance and PDE wall distance. The model used in this project is the exact wall distance and is calculated by making a projection in real space. The program can then use Kd search tress or Implicit tree algorithms to speed up the process.

1.2.2.4 Gradients

Variable gradients are calculated for each cell center and is of use to several functions within SCCM (Siemens, 2018). These functions are the following: constructing variable values at cell centers, calculating secondary gradients for diffusion terms, calculating pressure gradients for pressure-gradient solving and strain-rate and rotation-rate for turbulence models. The gradient model specifies the gradient method and limiter method. The gradient method in SCCM is chosen between two alternatives; Hybrid LSQ-Gauss or Green-Gauss. These methods calculate a vector that is perpendicular to the surface, in order to determine which way the fluid will go towards a new cell. As a help method SCCM provides three gradient limiters, which is Venkatakrishnan, MinMod and Modified Venkatakrishnan. These methods find the min and max bound of the neighboring cell values and make sure the gradient value will be within this area to not diverge.

1.2.2.5 Gravity

This is one of the non-mandatory models. For these simulations, however, the gravity is of big importance and consequently this model is essential.

For fluids, this option means two things. Firstly, the working pressure becomes the piezometric pressure which means that SCCM will take the depth of fluids into account when deciding the pressure (Siemens, 2018). This is decided with the following in Equation (13).

$$p_{piezo} = p_{static} - \rho_{ref} g(x - x_0) \tag{13}$$

Where p_{static} is the static pressure, ρ_{ref} is the density of the fluid, g the gravity constant, x the actual altitude and x_0 some reference altitude.

Secondly, this option will add the body force due to gravity in the momentum equations.

1.2.2.6 Implicit unsteady

It is of importance to the simulations whether the studied case is considered to be steady or unsteady. This means that the user must decide whether the simulation is expected to be constantly changing or be in a constant, non-changing state. Most real-life problems can be considered unsteady and so will the case studied here. For unsteady problems the following equation describes how the mass flow in the domain will change depending on how much mass is entering and exiting through inlets and outlets. It is illustrated in Equation (14) (Siemens, 2018).

$$\frac{dm}{dt} = \sum_{in} \dot{m} - \sum_{out} \dot{m} \tag{14}$$

By definition, an explicit approach means that each difference equation consists of only one variable and can therefore be calculated independent of the other equations.

In an implicit case, all equations need to be solved simultaneously to reach a solution.

1.2.2.7 Realizable K-Epsilon two-layer

This is an extension to the K-epsilon turbulence model (Siemens, 2018). This model contains a new transport equation for the turbulent dissipation rate ε described in length in the Star-CCM+ Documentation. Furthermore, it introduces the important coefficient C_u as a function of mean flow and turbulence properties. This way the program can better handle mathematical constraints of the stresses in line with the physics of turbulence.

The two-layer part describes what wall treatment is used and will be explained in greater detail later on.

1.2.2.8 Segregated flow

The segregated flow model means that the simulation will utilize the segregated flow solver to solve the momentum equations (Siemens, 2018). The segregated flow solver will solve the momentum equations for one dimension at a time and is recommended if the user is not working with cases such as shock capturing, high Mach-number simulations and high Rayleigh-number applications. None of these cases are relevant in this report and therefore the model can be used.

1.2.2.9 Two-layers all y+ wall treatment

This is an extension to the Realizable K-Epsilon two-layer model near the walls (Siemens, 2018). A number is calculated related to the mesh size and the wall distance. Depending on this number, different models can be used. For high y+ values the high y+ model can be used, while the low y+ model can be used for lower models. In this case the all y+ wall treatment Figure 1.2 is used which uses a combination of both of the mentioned models. One can assure herself from the graph that the logarithmic curve (low y+ model, u^{+}=y^{+}) will work well for y+ < 5 while the proportional curve u^{+} = (1/K ln E'y^{+}) fits much better for y^{+} >30.

u^{+}= proportional wall-cell velocity

K = 0.42

E' = roughness dependent number

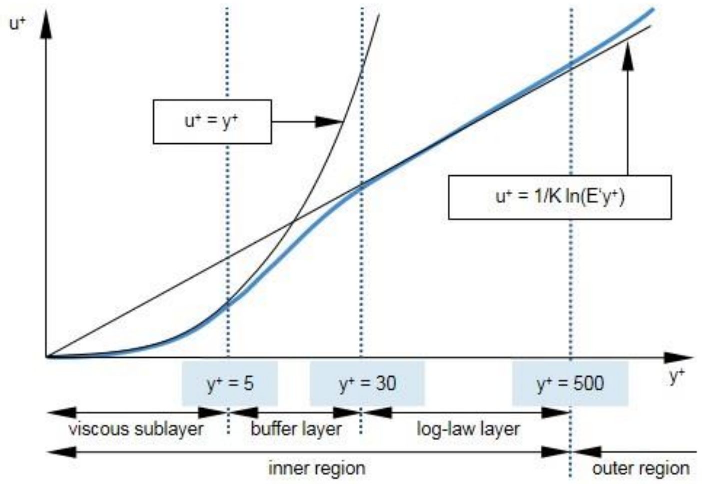


Figure 1.2 Wall treatment illustration

1.2.2.10 Cell quality remediation

Once this model has been selected, the program will automatically go through a couple of checks to make sure the cells are good enough for the future calculations to converge (Siemens, 2018). These checks are made up of a field function that returns a value of 0-3 depending on the state of the cell. 0 means the cell is good and has only good neighbors, 1 means the cell is bad but has good neighbors, 2 means the cell is good but has bad neighbors and 3 means the cell is bad and has no good neighbors. These checks include things such as checking the Skewness Angle which is illustrated as θ in Figure 1.3 below.

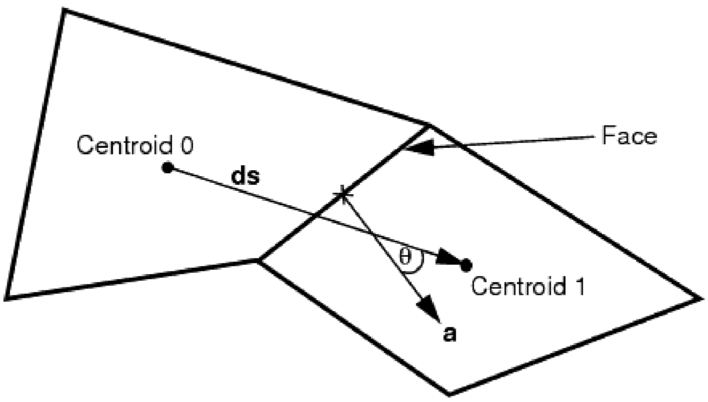


Figure 1.3 Cell quality remediation illustration

1.2.2.11 Volume of fluid (VOF)

The VOF method is one of the methods used for multiphase flow to describe and predict the changing interface between immiscible phases (Siemens, 2018). The volume fraction of phase is described by Equation (15).

$$\alpha_i = \frac{V_i}{V} \tag{15}$$

Where V_i is the volume of the phase in the cell and V is the cell volume. The volume fractions of all the phases must then sum up 1.

In this model the Volume fraction transport equation is described as Equation (16).

$$\frac{\partial}{\partial t}\int_V \alpha_i \, dV + \oint_A \alpha_i \, v \cdot da = \int_V \left(S_{\alpha_i} - \frac{\alpha_i}{\rho_i}\frac{D\rho_i}{Dt}\right) dV - \int_V \frac{1}{\rho_t}\nabla \cdot \left(\alpha_i \rho_i v_{dr,i}\right) dV + \int_A \frac{\mu_t}{\sigma_t}\nabla\alpha_i \cdot da \tag{16}$$

Where a is the surface area vector, v is the mass-averaged velocity, $v_{dr,i}$is the diffusion velocity, S_{α_i} is a user-defined source term of phase I, μ_t is the turbulent dynamic viscosity, σ_t is the turbulent Schmitdt number, and $\frac{D\rho_i}{Dt}$ is the material derivative of the phase densities ρ_i. When a cell consists of two phases, the transport equation is only solved for the first phase. The volume phase of the second phase is then adjusted so that the sum of the volume fractions becomes one.

The continuity equation for this model is described by Equation (17).

$$\frac{\partial}{\partial t}\int_V p dV + \oint_A \rho \, v \cdot da = \int_V S dV \tag{17}$$

Where

$$S = \sum_i S_{\alpha_i} \cdot \rho_i \tag{18}$$

The momentum equation is described by Equation (19).

$$\frac{\partial}{\partial t}\int_V \rho v \, dV + \oint_A \rho \, v \otimes v \cdot da = -\oint_A pI \cdot da + \oint_A T \cdot da + \int_V \rho g \, dV + \int_V f_b \, dV - \sum_i \int_A \alpha_i \, \rho_i v_{dr,i} \otimes v_{dr,i} \cdot da \tag{19}$$

Where p is the pressure, I is the unity tensor, T is the stress tensor and f_b is the vector of body forces.

Finally, the energy equation is described as Equation (20).

$$\frac{\partial}{\partial t}\int_V \rho E \, dV + \oint_A \left[\rho H v + p + \sum_i \alpha_i \rho_i H_i v_{dr,i}\right] \cdot da = -\oint_A \dot{q}'' \cdot da + \oint_A T \cdot v da + \int_V S_E \, dV + \int_V f_b v \, dV \tag{20}$$

Where E is the total energy, H is the total enthalpy, $\dot{q}''$ is the heat flux vector and S_E is a user-defined energy source term.

1.2.2.12 VOF Waves

This model in STAR-CCM is used to simulate surface gravity waves on the surface of a mass of water (Siemens, 2018). The program uses periodic wave train with some different options of sinusoidal functions to describe the movement. The wave train can then be completely specified by water depth, wave length and wave height. The main theories behind this function are the Stokes Theory, which is an assumption that the waves are not very steep and the Cnoidal Theory which is a theory that models water waves in shallow water. There are then a couple of different wave models that the program decides between depending on the previously mentioned parameters.

2 STAR-CCM+

STAR-CCM+ (SCCM) is a high performance CAE (Computational Aided Engineering) software supplied by SIEMENS since 2015. SCCM solves multidisciplinary problems for both solid and fluid continuum. SCCM also provides CAD models repair and meshing capabilities.

2.1 Introduction of STAR-CCM+

SCCM is a powerful program that can handle both CAD and CFD work within the same program. This means that all steps from CAD to post-processing could be progressed through the workflow containing different sequences.

2.2 Process approach

There are two ways of using SCCM, either you use the local desktop capacity or a server. When using the local capacity, the initial option is called “Parallel on local host” when creating a new simulation inside SCCM. This way of working is enough for smaller simulations or when no time limit is set for the simulation.

If the model is bigger or if one needs a very fine mesh, more capacity is needed. It is then recommended to use a server to handle the process, this is performed by using an external software to put the sim file on a server and then connect to it through SCCM. What happens when one uses the server is that SCCM is no longer restricted by the number of CPUs inside the computer but can instead utilize as many CPUs as the server has available. The simulation speed is proportional to the number of CPUs because the software can perform calculations on each CPU simultaneously. This means that the simulations can be performed at a much faster rate by using a remote server than what would be possible on the local computer. Some functions inside the meshers can also be sped up by using more CPUs, but this is not proportional as some functions will cause a bottleneck as they need to be run on a single CPU.

2.3 Terminology

To give an overview of how the simulation process is structured, Siemens has made the picture shown in Figure 2.1. This way, one can see what steps are required to move onto the next one.

Figure 2.1 Terminology structure

To show how the terminology is working, the example in Figure 2.2 Example illustration will be used as a template of explanation for each step. This is a very simplified picture of a general CFD case with a solid bottom, with two areas of fluids separated by a porous medium. For each part of the process these areas will be represented by different categories. As such this will work as an example of the different point of views for each step along the way.

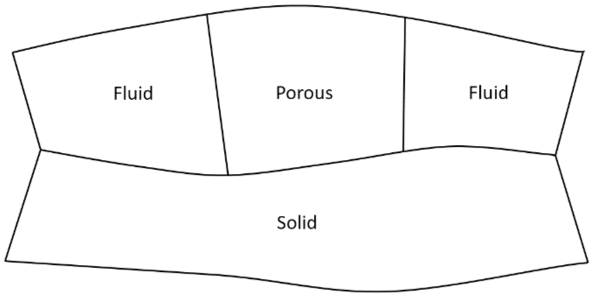


Figure 2.2 Example illustration

2.3.1 **Parts**

A part is a component of a geometry with a geometric shape and an actual size. Figure 2.3 is showing how the template is divided into different parts.

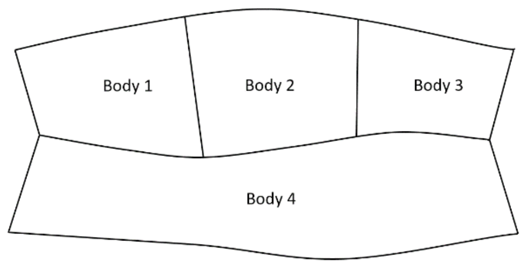


Figure 2.3 Part definition

2.3.2 **Operations**

Operations work as an action to each part, that can easily be modified when inputs or properties are changed.

2.3.3 **Regions**

A region specifies a volume in 3D-, and a surface in a 2D environment. Figure 2.4 shows that the areas that were previously considered parts are now instead considered to be regions. This means that it is no longer simply CAD information but physics are taken into account as well.

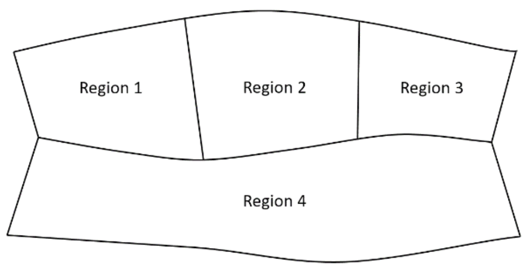


Figure 2.4 Region separation

2.3.4 **Boundaries**

Boundaries are the exterior surfaces of the regions. It could be physical boundaries e.g. a wall, an inlet or an outlet. But it could also be a connection between regions. In the example an inlet is placed on the left side and an outlet is placed on the right.

The rest of the outer boundaries are set to wall. The inner boundaries are all set to interface boundaries, which means that the boundaries are connected to each other. All boundaries are illustrated in Figure 2.5.

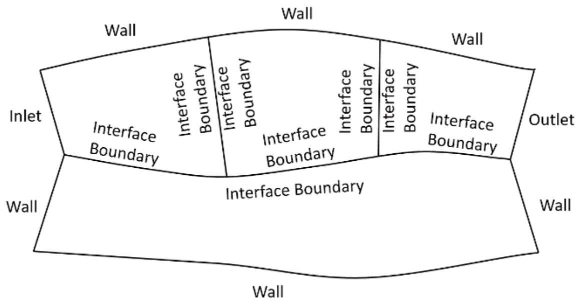


Figure 2.5 Boundary types

2.3.5 **Interfaces**

Interfaces is a function that specifies the transfer of mass and (or) energy between the regions. There are two different settings:

- Internal interfaces, allow transfer of both mass and energy.
- Contact interfaces, allow transfer of only energy.

Figure 2.6 is showing how the interfaces are placed in the example. The boundary next to the solid part has a contact interface due to the fact that no mass transfer is possible into the solid part.

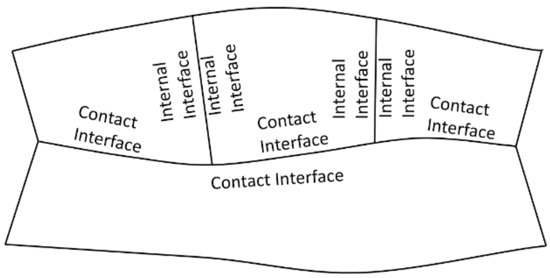


Figure 2.6 Interface specification

2.3.6 **Continua**

The continua is a representation of a model that illustrate either the physics or the mesh of one or more regions. In Figure 2.7the physics settings are illustrated by the different continuum.

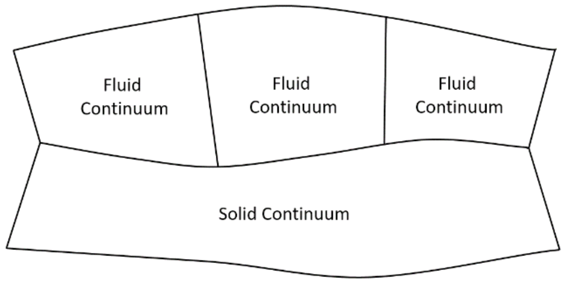


Figure 2.7 Continuum illustration

2.4 Workflow

The workflow in SCCM is built up as Figure 2.8 is showing.

Figure 2.8 Workflow in STAR-CCM+

2.4.1 **Import**

SCCM allows the user to have the flexibility to import both meshed CAD models but also CAD data that is not meshed yet. When importing a meshed file, the file needs to contain surface elements that are defined by three vertices. Regarding the CAD data, there are three alternatives which are; triangulated-, neutral-, and native format.

Neutral formats files are files like IGES, STEP and IDF while triangulated formats files are files like STL and Catia graphics.

The native format requires an additional license due to the CAD data being exchanged from third-party products.

2.4.2 **Surface preparation**

Surface preparation is needed because all geometries are not always suitable to be directly imported into a simulation. There are some steps that needs to be fulfilled before any simulations can be run. CFD simulations require the outer boundaries to be closed and manifold, which means that all outer surfaces need to be connected to each other to close a volume. It is also recommended that in this phase split or combine different surfaces that will be important in the simulation.

Inside SCCM it is possible to modify the geometry due to the built-in function called 3D-CAD. This function allows the user to use functions such as sketching, extrusions, revolving, lofting, sweeping and imprint. It is also possible to add or remove parts, surfaces and lines in 2D and 3D.

Another option is to use the built-in function called repair surface. The difference between 3D-CAD and repair surface is that the changes inside repair surface is made in an environment where the surface of the geometry has a pre-mesh format as triangles. Inside this function it is possible to check the status of the geometry based on six different criteria which are explained subsections 2.4.2.1-2.4.2.6. In general face quality and face proximity do not need to be fulfilled because they usually do not cause any problem.

2.4.2.1 Pierced faces (self-intersection)

Pierced face checks if a surface is intersected by one or more edges into another surface. If pierced face is found it is highlighted with a red color of the edges that are intersected. This is illustrated in Figure 2.9. This error could be fixed manually by deleting the surface with the intersected edges and then building up a new one that is not intersecting. If the geometry has a lot of intersections it is possible to do the repair with the function called surface wrapper, which is further explained in subsection 2.4.4.3.

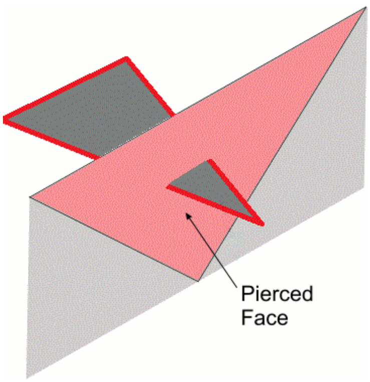


Figure 2.9 Pierced face

2.4.2.2 Face quality (tolerant geometry)

It is optimal to have equilateral triangles over the geometry to create an even surface quality, but this is hard to achieve with complex geometries. Triangles colored by yellow indicate that the triangles have large differences in size. To decide if the face quality is poor or good it is possibly to calculate a ratio. The calculation is done by measuring the largest possible circle that could fit inside the actual triangle and then compare it to the circum-circle that is attached to the vertices of the triangle. A value between 0 and 1 is given by Equation (21). A value below 0.01 consider a poor quality by default and a value close to 1 shows that the actual triangle is similar to the ideal triangle which shows in Figure 2.10.

$$Face\ quality = 2 * \frac{r_{in-circle}}{r_{circum-circle}} \tag{21}$$

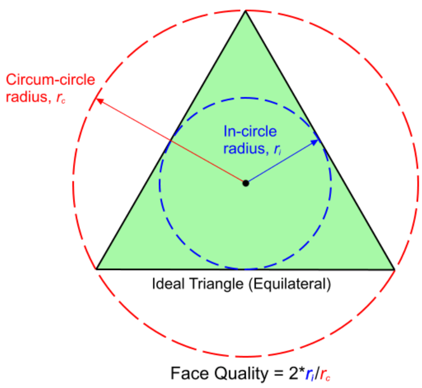


Figure 2.10 Face quality

2.4.2.3 Face proximity (overlapping entities)

Face proximity considers the distance between two faces. If the distance is getting too small, the surfaces will become orange. This is shown in Figure 2.11. This indicates that the surfaces are having a risk of overlapping each other.

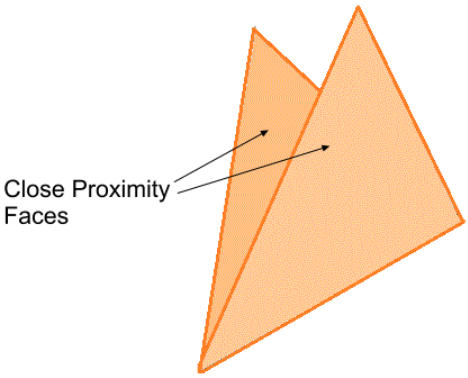


Figure 2.11 Face proximity

2.4.2.4 Free edges (invalid spline definition)

Free edges occur when at least one edge is unconnected to another face. This is illustrated in Figure 2.12, where four faces are having one free edge each. When finding this error, it is marked by a green color on the edges. The case in Figure 2.12, could be solved by inserting a new face connected to all free edges, which creates a closed volume.

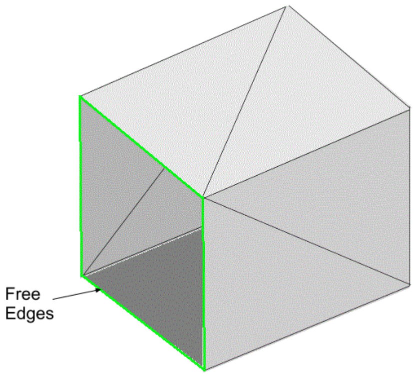


Figure 2.12 Free edges

2.4.2.5 Non-manifold edges (invalid geometry)

A non-manifold edge is when one edge is shared by three or more faces. When finding this error, it is marked by a blue edge. This error is illustrated in Figure 2.13, where the interior surface is joined with the exterior surfaces. This composition creates an edge which is shared by three different faces.

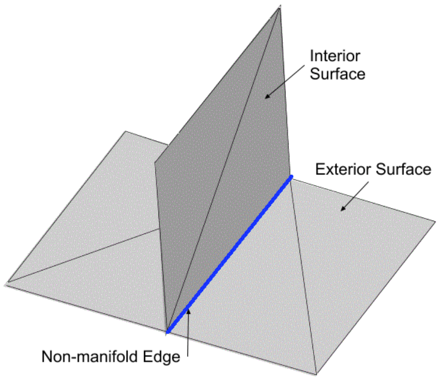


Figure 2.13 Non-manifold edges

2.4.2.6 Non-manifold vertices (invalid geometry)

A non-manifold vertex occurs when the only connection between faces is the vertex. This is illustrated by a blue dot in Figure 2.14.

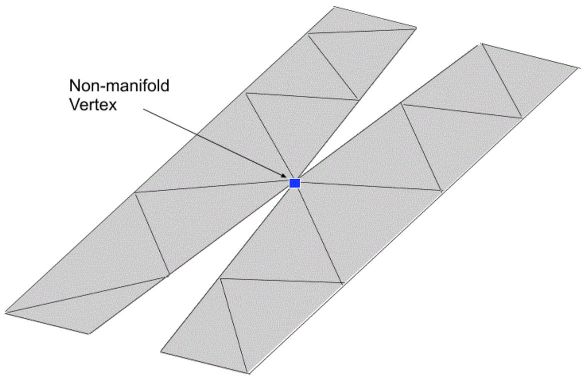


Figure 2.14 Non-manifold vertices

These types of errors are solved by using repair surface, which changes the triangle structure. This function has two ways of proceeding, either by changing manually or by using the auto-repair.

2.4.3 **Boundary types**

To define how to handle the interaction between the parts, boundary types are required to be set. It is important to set the boundaries before the mesh, because the boundaries will be considered when generating the mesh. In general, SCCM have ten different boundary types to choose between. These are listed in subsection 2.4.3.1-2.4.3.8:

2.4.3.1 Axis

An axis boundary is when the geometry has a constant dimension around an axis, it is advantageous in cases of pipes for example. Figure 2.15 is showing the axis as the orange line in a 3D geometry.

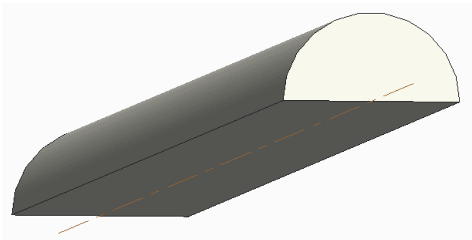


Figure 2.15 Axis boundary

2.4.3.2 Flow-split outlet

A normal outlet boundary is assumed to have the capacity to handle the total mass flow . Flow-split outlet is a ratio defined by a value between 0-1 for each boundary that specifies the split between the boundaries that are set to the flow-split outlet. The sum of the values should be 1, and if just one flow-split outlet is used, all the flow is expected to go through this boundary.

2.4.3.3 Mass flow inlet

Mass flow inlet defines which surfaces will apply the continuum (either a fluid or a gas) into the simulation. This is one of the ways that a fluid can be introduced to a simulation. This option is a good choice if the fluid is known to enter from some point in the simulation area. With this function it is possible to specify the fluid and mass per unit time (kg/s). If a more complex description of the mass flow is needed, field functions could be added, but the mass flow cannot vary across the surface. With a field function it is possibly to create a customized sequence using mathematical formulas.

2.4.3.4 Overset mesh

This boundary function is preferably used when a part/parts are in motion during a simulation. With this function is it possibly to have a fine mesh located all the time close to a specific geometry part that is moving.

2.4.3.5 Pressure outlet

If a boundary is set to a lower pressure than inside the geometry, the surface will start to act as an outlet. Due to the pressure difference, the fluid will start to flow towards lower pressure. This function is often used as an outer boundary in order to eliminate the fluid from the simulation. For the simulation to not be completely filled with the fluid entering through the inlet, there must exist some form of outlet to make sure that the mass balance is maintained.

2.4.3.6 Symmetry plane

Symmetry plane is a function that is of use when the geometry has a symmetric structure. By using the symmetry plane function the simulation will become faster, since only half of the geometry needs to be simulated. Because the symmetry plane is obtained as a mirroring plane and will provide an identical result as simulating the whole geometry. An illustration of a symmetry plane could be studied in Figure 2.16.

Figure 2.16 Illustration of symmetry plane

2.4.3.7 Velocity inlet

This boundary represents a flow that goes into the simulation with a specific velocity value. Velocity inlet allows the user to the insert the flow with different parameters of the velocity vector if needed. If more complex description of the velocity flow is needed, field functions could be added.

2.4.3.8 Wall

Compared to the other boundaries this is a boundary that represents an impermeable surface that prevents fluid and solid matter to flow through the boundary surface. When using wall as a boundary setting, friction and models acting near the wall can be modified. There are two different alternatives regarding the friction, no-slip or slip. No-slip is applied by default which means that fluid adheres to the wall and moves with the same velocity as the flow near the wall. Slip is a setting that makes the fluid flow faster than the velocity flow near the wall.

2.4.4 **Meshing**

SCCM is a mesh dependent software which means it requires a mesh to run the simulation. The mesh could be created by three different geometry shapes, these are polyhedral, tetrahedral and trimmed, for further description see subsection 2.4.4.1.

The purpose of generating a mesh is to divide surfaces and volumes into smaller cells, where each cell could be calculated separately and then become combined into a common result for the whole geometry. In Figure 2.31 there are five different types of cells illustrated. A further explanation of each cell is described in subsection 2.4.4.2.1-2.4.4.2.5.

The meshing process is divided into two steps when using SCCM. The first step in the meshing process is to create a surface mesh on all surfaces, the second step is to create a volume mesh where the surface mesh acts like a foundation to the initial volume mesh structure.

Before generating a mesh, it is important to decide which areas that are of most interest and then refine those areas. Refining an area is done by reducing the size of each cell within the selected volume. A fine mesh generates a more accurate result, but is also more time-consuming. Therefore, it is important to only refine areas that are of interest.

2.4.4.1 Geometry shapes of the mesh

Depending on what kind of simulation, it is preferable to use different types of mesh shapes. The simulation will become better and the calculation will become faster when using a more suitable mesh shape. For example, a polyhedral mesh is preferable to use in cases of heat transfer, swirling flow and complex flow. Tetrahedral is a mesher that works for most cases, but it requires a fine surface of the original geometry to create a good mesh. Trimmed mesh is optimal for electronics cooling and external flow. A separate tool that can be used in order to make sure that the behavior of the flow near a wall is “Prism Layers”. This is a tool that creates a finer mesh along surfaces that is of most interest. The prism layer mesher can be applied together with any of the global meshers (polyhedral, tetrahedral and trimmed).

The global shape structure of the mesh could be generated in three potential volume meshes that are explained in subsection 2.4.4.1.1 - 2.4.4.1.3.

2.4.4.1.1 Tetrahedral

Using tetrahedral provides an efficient and simple solution for complex geometries. Tetrahedral uses the least amount of memory, and that makes this method the fastest.

An illustration of one single tetrahedral could be studied in Figure 2.17 and a group of tetrahedral could be studied in Figure 2.18.

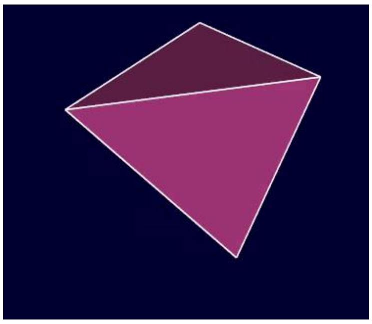


Figure 2.17 Tetrahedral mesh

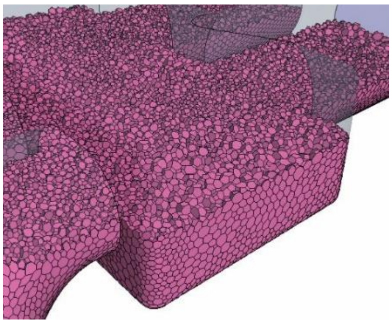


Figure 2.18 Illustration of tetrahedral pattern in 3D

2.4.4.1.2 Polyhedral mesh

This mesh method is adaptable for both simpler and more complex geometries. The preparation when using polyhedral is the same as working with tetrahedral. The advantage with polyhedral is that it contains approximately five times fewer cells than a tetrahedral mesh. This is due to the process when creating the cells. The process starts with a tetrahedral mesh as input which is shown in Figure 2.19. The next step in the process is to calculate a center point of each cell or line (if the line creates the boundary). This is illustrated in Figure 2.20. Those points are later connected to each other by straight lines. This process starts at the boundary and works towards centrum of the geometry. This stage is illustrated in Figure 2.21 – Figure 2.22. The final mesh in a 2D section using polyhedral mesh will look like Figure 2.23.

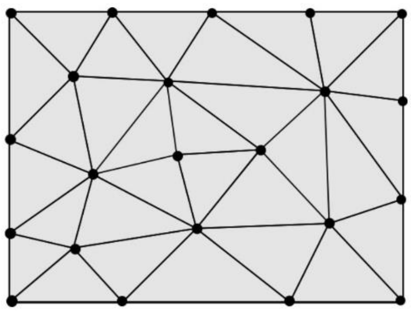


Figure 2.19 Input mesh

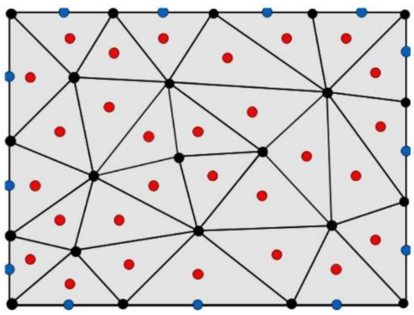


Figure 2.20 Process

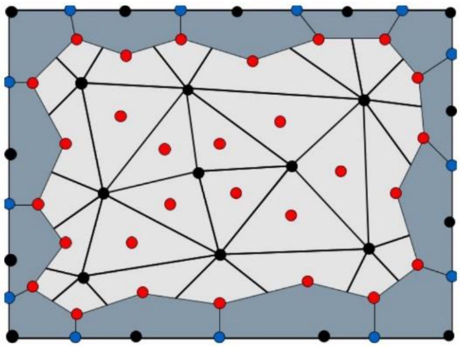


Figure 2.21 Process 2

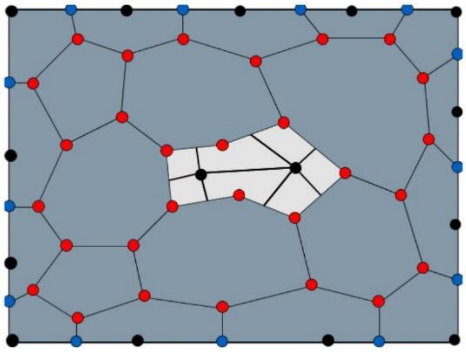


Figure 2.22 Process 3

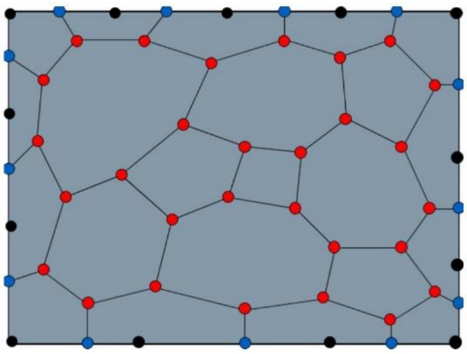


Figure 2.23 Final mesh

To show the final mesh structure even further, Figure 2.24 is showing one single polyhedral in 3D and Figure 2.25 is showing a group of polyhedral mesh in 3D mode.

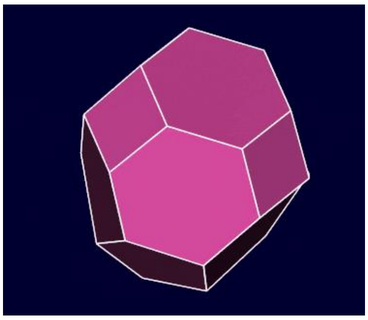


Figure 2.24 Polyhedral mesh

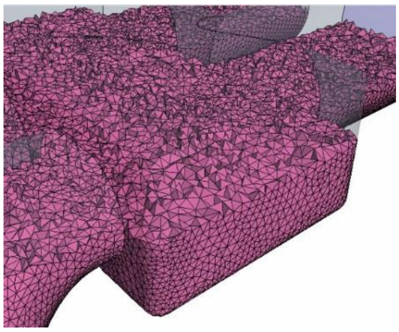


Figure 2.25 Illustration of polyhedral pattern in 3D

2.4.4.1.3 Trimmed

Trimmed mesher is a strong and efficient alternative when a high-quality grid is needed for both simple and complex geometries. A sequence of three operations generates a trimmed mesh. The first step is to define an area/volume of the geometry, which is illustrated in Figure 2.26. Second step is to create a square pattern that cover the whole geometry, this is illustrated in Figure 2.27. Third step is to trim away the square pattern that does not intersect with the geometry, this is illustrated in Figure 2.28.

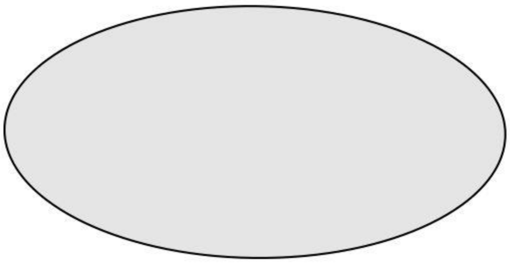


Figure 2.26 Define an area

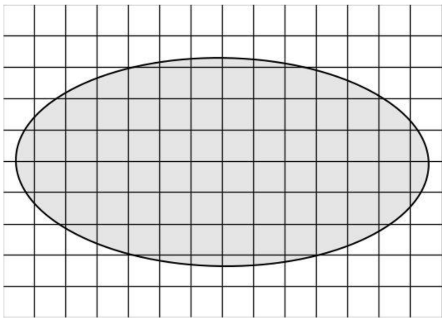


Figure 2.27 Inserting a square pattern

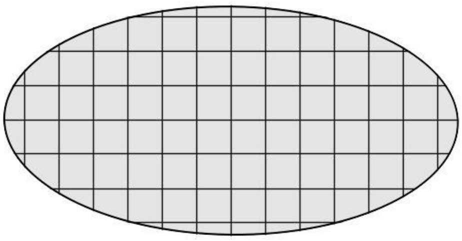


Figure 2.28 Trimming the pattern outside the area

To show further what the trimmed mesh looks like, Figure 2.29 is showing one single cell in 3D and Figure 2.30 is showing a volume of trimmed cells in 3D mode.

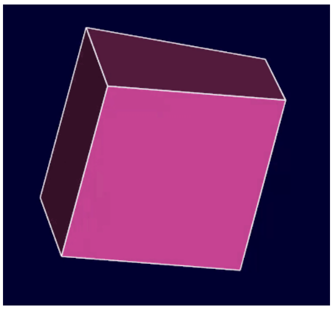


Figure 2.29 3D trimmed cell

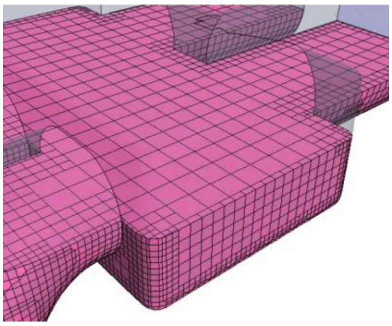


Figure 2.30 Illustration of trimmer pattern in 3D

2.4.4.2 Mesh structure

Independent of the mesh shape, the structure will always be the same. The mesh structure is a mathematical description of the geometry and is made to be able to calculate the simulation. In Figure 2.31 five different structure parts are visualized by red, and subsections 372.4.4.2.1 - 2.4.4.2.5 provide a more detailed description.

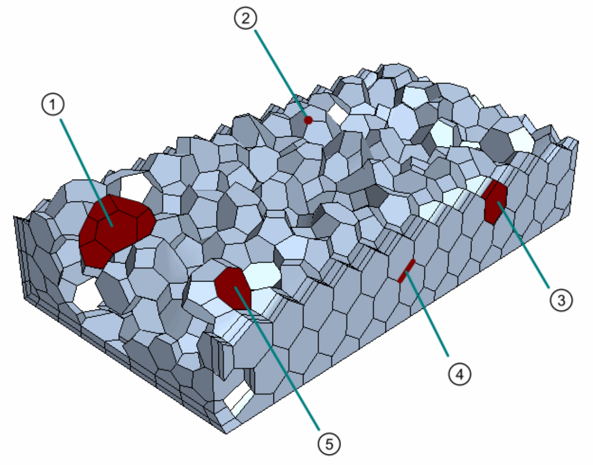


Figure 2.31 Mesh illustration

2.4.4.2.1 Cell

A cell is a volume that could have different shapes but always consist of surfaces, lines and vertex. Figure 2.32 is showing one example of a cell and in Figure 2.31 number one is showing how it looks like in a context.

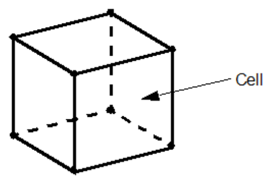


Figure 2.32 Cell illustration

2.4.4.2.2 Vertex

Vertex is a point in the the enviroment that can be defined with a position vector. These verticess are the ones that form a base to create lines and surfaces. In Figure

2.33 a vertex is illustated by a positioned vector related to the coordinate system x-y-z. Another example of a vertex is illustrated in Figure 2.31 marked by number 2.

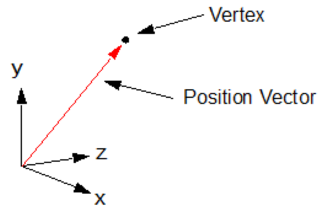


Figure 2.33 Vertex illustration

2.4.4.2.3 Prism layer cell

Prism layer cell is a cell placed in a layer which is located close to the outer surface of the geometry. A prism layer cell is demonstrated in Figure 2.31 marked by number 3.

2.4.4.2.4 Feature curve

When having two or more vertices it is possible to create straight lines in between those vertices. One vertex could be used for different lines. An illustration of a line is showed in Figure 2.34 and in Figure 2.31 marked by number 4.

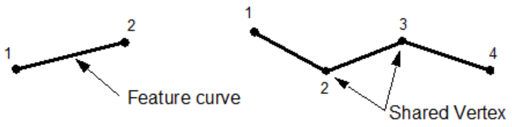


Figure 2.34 Lines/edge illustration

2.4.4.2.5 Surface

With vertex and lines surfaces can be created. A surface needs at least three vertices and three lines to create a complete surface. In Figure 2.35 a surface is described with four vertices and four lines. A surface with even more vertices and lines is available to be studied in Figure 2.31 marked by number 5.

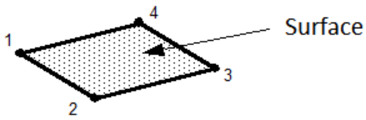


Figure 2.35 Surface illustration

2.4.4.3 Surface wrapper

The surface wrapper is a tool provided by SCCM that simplifies the process of creating a CAD model with a surface mesh that fulfills the requirements for the volume mesher to work. While simple geometries often contain no areas of difficulty, more complex models tend to require some degree of preparation. As the CAD models provided by Volvo are not mainly for simulations, these contain lots of problem areas in a CFD environment. In these big CAD models, the repair tool can often find thousands of surfaces that will prove problematic. Repairing these in a CAD software is a time-consuming process and therefore the surface wrapper provides a simplified way of creating an initial volume to work with.

The surface wrapper has some different options regarding how to initialize the process. These include largest internal, seed point, external and Nth largest. By choosing largest internal, an initial volume is created where the biggest volume is found in the model. By choosing seed point, the wrapper starts creating a volume from the user-generated point which is then expanded until it fills the volume that surrounds it. The external option starts by creating a volume outside the whole CAD model which is then shrunken until it encapsulates the CAD model from the outside. Finally, Nth largest works like the largest internal option but instead of finding the largest volume, it finds the Nth largest where N is a number defined by the user. Once the volume has been defined, a surface mesh is created based on this. No volume mesh is created during the surface wrapping.

There are several points where the Surface Wrapper is of great use. These include:

- A great number of intersecting surfaces.
- The existence of holes and gaps in the geometry.
- Duplicate and overlapping surfaces.
- Complex geometry with great detail that will cause the volume mesher to create low quality cells.

It should be noted that the Surface Wrapper simplifies the geometry and sometimes alters the geometry. This means that the user must be careful that the important parts of the geometry to the flow are maintained. If the shape of the geometry is not maintained the user needs to go back to the CAD to solve these issues.

The Surface Wrapper contains many different options regarding the sizing of the surface mesh created. The user can define global settings for the surface mesh created but it is also possible to make a finer surface mesh in areas where it's presumed to cause issues with the volume mesher.

2.4.4.4 Volume mesh

Once a surface mesh has been created, this surface mesh will be used as an initial layout to the volume mesh that needs to be created for the program to produce simulations. The volume mesh is what decides how accurate the simulation will be and there are many factors to consider. The most important one is the tradeoff between time and accuracy. A mesh with a large number of cells will provide a more accurate solution but will also be a lot more time consuming. It is also important to make an evaluation of which areas are relevant to the flow. There is no reason to have a fine mesh in an area where no fluid will be flowing. Regarding what type of mesh, please refer to subsection 2.4.4.

Just like the Surface Wrapper, the volume meshers also include settings both for a global level but also for a local level. In bigger models it is important to utilize the local functions as being smart about the choices of mesh sizing in different regions is a relatively fast process that can save a lot of time in the end.

2.4.5 Physics

SCCM has a wide range of physics settings that are available to choose between. This gives the user a way to simulate different kinds of scenarios depending on what physics are relevant in the case of interest.

All physics that have been used at least once in one of the simulations are explained in a simplified way in Appendix A to give an overview of the chosen models. If more information is needed, study the chapter theoretical framework, where each physics setting is explained in greater detail. To know what physics that has been active for each simulation, please study Appendix C.

2.4.6 Solver settings

The solver panel in SCCM provides two important values for the simulation. These values are the time-step and the stopping criteria.

The time-step is what time interval the software uses between each calculation. A higher time-step means that fewer calculations needs to be made, which directly reduces the time it takes to complete the simulation. The disadvantage however, is that information might be lost if the time-step is too high. This can result in a simulation that does not represent the reality, as the simulation becomes more and more incorrect. It also increases the risk of the calculations diverging, causing the program to crash. Because of this, it is important that the user chooses the time-step carefully and continually judged whether the value is correct.

The stopping criteria is a way for the user to choose when the simulation should be stopped. As simulations are often a protracted process, sometimes running over night, it is important to make sure the simulations are not run unnecessarily long as the wasted time could be spent on running another simulation. The stopping criteria can be chosen in a couple of different ways. The ones used in this project included the maximum physical time and the maximum inner iterations. The maximum physical criteria let the user decide how many physical seconds simulation should run for. The stopping criteria for maximum inner iterations instead lets the user decide how many times the program should perform the calculations before stopping.

2.4.7 Post processing

As a final step in the simulation process, the user must decide how to visualize the case. This is of importance in this project as this will be the basis for the analysis of the car design. The post processing options are based on different kind of scenes, the ones used in this project are: geometry scenes, mesh scenes, vector scenes and scalar scenes.

2.4.7.1 Geometry scene

The geometry scenes are used to inspect the CAD shapes either imported or created in SCCM. This is a simple scene that is not used during the simulation. Because of this, the settings in this scene are limited.

2.4.7.2 Mesh scene

The mesh scene shows the CAD shapes like the geometry scenes but here the mesh is applied as well. This gives the user a chance to judge whether the mesh is good enough. It can be quite hard to judge the mesh from a 3D point of view and it is recommended to use a 2D plane view instead.

2.4.7.3 Vector scene

The vector scenes will show arrows with the magnitude of the studied property of each cell. This type of scene was used in this project to study the magnitude in different areas to make sure it was at reasonable values. The vector scene provides a set of settings called vector field. In this set the user choose what function should be shown in the scene. There are some premade functions to choose from, but it is also possible to create a customizable function based on a mathematical formula. Vector field also provides a way to decide which values should be shown in the scene. This is done by setting a minimum and a maximum value that limits the information displayed in the scene. If the values of interest are not known, there is also an automatic range function where SCCM puts the maximum value to the maximum value found in the mesh and the minimum value to the minimum value found in the mesh. It is also possible to decide which color should be used to show the values.

2.4.7.4 Scalar scene

Finally, the scalar scenes visualize the simulation where the scalar properties like pressure and volume fraction are colored depending on the value of each cell. This is what was used to create the videos of the complete cases. The visualization settings in scalar scene is reminiscent of the ones in a vector scene. Here it is also possible to decide minimum and maximum values as well as what color to use.

2.4.7.5 Common settings

Each scene comes with its own set of options but some of them are shared between all the scenes. For each scene the user can decide how often the scenes should be updated in the graphical window while running the simulation. This option also decides how often the program captures pictures of the different scenes. It is possible to save all the simulation values with SCCM and then create video scenes once the full simulation is run. The disadvantage of this is that the file size of the stored values become very big and can take a lot of disc space. The other option to capture the simulations is to let SCCM take pictures every time the scenes are updated. This way no values are stored but saved as an image file instead. By using this approach, a lot of disc space can be saved.

3 PreonLab

PreonLab is a CFD program created by FIFTY 2. The approach differs from the traditional way of solving a CFD simulation. This program is using particles to generate the geometry and to simulate the fluid flow. No mesh is needed to solve the simulation.

3.1 Introduction

The intention with PreonLab is to reduce the time to get a result without compromising the simulation quality while providing an easy to use user interface. PreonLab is using a free-surface flow to solve the CFD simulation. In addition, it provides the possibility to set parts in motion during the simulation without making the pre-processing more complicated, due to smart functions within the working environment.

3.2 Process approach

Due to the different approach of solving the simulation it is possible to finish a simulation on a local computer much faster than SCCM. There is, however, a minimum requirement of 64 GB RAM memory to handle computations. If needed there is also a possibility to speed up the simulation even more by running the program on a cluster with high CPU capacity. It should also be noted that the main reason why the simulations are faster is that some physics that are handled in SCCM are not at all taken into account in PreonLab. This is a simplification that does not have a big impact on some cases but will greatly reduce the accuracy of some cases.

3.3 Workflow

The first thing to do when creating a new simulation is to prepare the geometry in an external CAD program which will later be imported into PreonLab. Inside PreonLab there are three steps to go through, the first step is to set up the pre-processor, second is to run the simulation, the last step is to study the result with the help of the post-processor. To find a more detailed description of the CAD preparation and each step inside PreonLab please read subsection 3.3.1 - 3.3.4.

3.3.1 CAD preparation

Depending on what kind of simulation, different CAD preparations are necessary. It is important to do all changes before importing the geometry, because there is no option to modify the geometry inside PreonLab. To make sure that the simulation of fluid flow can run in PreonLab,it is necessary to study the expected fluid path and see that the geometry is similar to the reality in this area. Another thing to check is if parts are able to move properly, which means that parts should not interact with each other during motion. If a part should move in the simulation, it is important to create this part as a separate file. This will make it easier in the pre-process.

3.3.2 Import

When importing the geometry from an external CAD program it is important that the file is saved as an STL-file because an STL-file contains a pre-mesh which PreonLab can transform to a geometry consisting of particles. Normally, the resolution of the STL should not have any impact on the simulations but for simulations with extremely small particles, it has been observed that these particles might flow in between the solid particles that form the CAD structures. Because of this, it is recommended to use the highest resolution that the CAD software where the CAD model is exported from allows.

3.3.3 Pre-processor

The pre-processor in PreonLab provides the user with the ability to completely setup the studied case. To perform a realistic simulation, the pre-processor provides several useful tools which are explained in greater detail below. The information in this chapter is gathered from the PreonLab user manual created by FIFTY2 Technology GmbH (GmbH, 2018).

3.3.3.1 Inlet sources

PreonLab enables different types of particle injection into the simulation and those can be explained by two main groups, continuous and one-time. A continuous inlet means that the injection of the particles is happening during a period, while a one-time source is specified to a specific time with a limited period. Settings as size, position and directions are all possible to adjust for all inlet sources. To understand what the difference is between the inlet sources please study subsection 3.3.3.1.1 - 3.3.3.1.6.

3.3.3.1.1 Square and circle source

Square and circle is a plane source that emits fluid continuously as the shape of the inlet source. A square source injects a rectangular stream while a circle source is injecting stream flow as a circle. These sources have the option to adjust the volume per time, which is available due to a function called keyframe which are explained further in subsection 3.3.4.15.

3.3.3.1.2 Area source

Area inlet is similar to square and circle source, which emits fluid continuously from a 2D plane with the possibility to use the function keyframe. The difference is that it only works when it covers an entire region. This is illustrated in Figure 3.1, where the left side of the tube is covered entirely, and the right side is not completely covered.

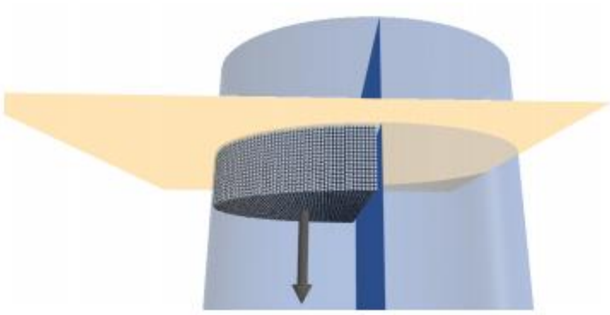


Figure 3.1 Area source

3.3.3.1.3 Flat jet source

Flat jet inlet is a source that emits fluid from a convex plane with a continuous fluid flow. The width and the angle which creates the length of the plane are both set during pre-process. Figure 3.2 is showing an example of a flat jet source.

Figure 3.2 Flat jet source

3.3.3.1.4 Volume source

Instead of injecting fluids from a plane, it is possible to inject a specific volume of fluid into the simulation with the help of a the function called volume source. Volume source is defined by a 3D box which is filled with fluids. This function works as one-time which means that the contents of the box will be released at a specified time, and the box will stay empty by default after the release. If the box should be filled once again after the release, keyframe is needed.

3.3.3.1.5 Filling containers with a volume source

This function is an extension of volume source. It is used when a volume of the geometry should be filled with fluid. In order to fill the volume with fluid two requirements need to be fulfilled. The first demand is that the volume needs to be located inside the box domain which are set by the user with a size and position. The second qualification is that a red dot needs to be set inside the box domain and specify where the fluid should be placed in the geometry. This is illustrated in Figure 3.3, where the upper geometry is empty, but has both requirements fulfilled. The lower geometry in Figure 3.3 is showing when the function has been used and the fluid has been applied.

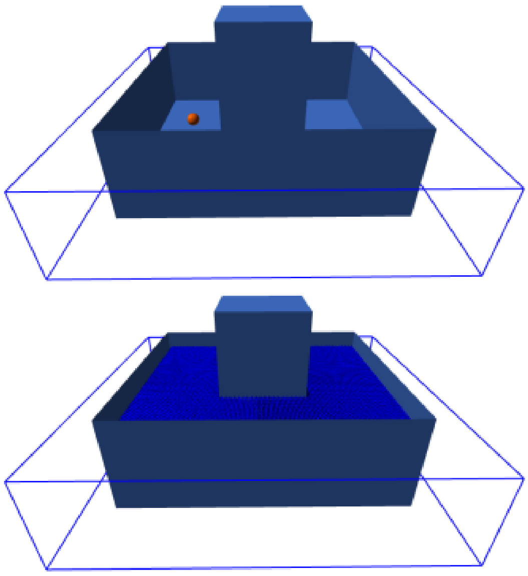


Figure 3.3 Filling containers with volume source

3.3.3.1.6 Rain source

Rain inlet is a modification of square and circle source. Rain source is also referring to a plane which has an inlet with the shape of a rectangular or a circle but instead of injecting the same flow across the plane the droplets are emitted randomly from the plane with varying diameters of the particles within a range of d_{min} and d_{max}. It is still possible to adjust the volume per time unit, but the function is handling the distribution of the particles from the plane. When using rain source, it is possible to calculate the mean size (d_{mean}) of the emitted particles by Equation (22).

$$d_{mean} = \frac{d_{min} + d_{max}}{2} \tag{22}$$

3.3.3.2 Boundary domains and conditions

With default settings the geometry is imported to an infinite surrounding without any outer boundaries. Fluid that is injected into the simulation will be a part of the simulation until the end. This causes each particle to be calculated within each iteration, even if it does not affect the final result. To prevent calculating irrelevant particles, it is possible to restrict the simulation with outer domain boxes. These domain boxes retain particles that are located inside the box and delete particles that are going out of the box domain. This function is advantageous to use in important areas to delete particles that are leaving to non-interesting areas.

3.3.3.3 Physics

The PreonLab software does not provide the many different physics options that can be found in SCCM. As the physics are based on particles rather than mesh cells, properties such as temperature are not calculated. Since there are no air particles interfering with the fluid particles, this means that simulations where air flow would have a big impact are not as accurate. It is, however, possible to simulate an air flow by using certain functions in PreonLab briefly explained below.

3.3.3.3.1 Force field

For the physics in PreonLab, several different force fields can be applied to the chosen volume and this is what creates the motion in this software. Unlike SCCM where motion is created by other factors like pressure differences PreonLab does not at all take air flows into account. Some of these force fields will be explained in further detail below.

3.3.3.3.2 Gravity

This function simply applies a gravity force on each particle downwards. If no other force fields are applied this will be the driving force to initialize the simulation. For the simulations performed in this project, no other force fields were applied during the different scenarios.

3.3.3.3.3 Drag force

If the user wants to simulate some sort of force to each particle, for example a wind blowing from one side to another, a drag force can be applied. By using this tool, the user can also move an object along some trajectory and simulate a moving motion with an air flow working towards the direction of the object.

3.3.3.3.4 Solid object

In PreonLab a lot of different functions are dependent on the user creating an appropriate setup. There are no mass outlets in this software, but instead the particles are removed from the simulation once they leave the domain decided by the user. To assist with the setup, PreonLab provides a set of different premade geometry shapes that can be placed in the simulation space. This way if one wants to modify the geometry it's possible to add shapes like boxes, cones or cuboids to experiment with different setups. If an item is added and is connected to a CAD part that has a motion during some time frame, the added shape will mimic this motion.

3.3.3.3.5 Film wetting

With film wetting, Fifty2 has added a function similar to a fluid film in Star-CCM, which simulates a surface that already has a thin layer of fluid on the chosen surface. In line with the rest of the software, film wetting adds fluid particles to the surface where the user can then utilize some parameters to achieve the wanted effect. A lot of the central parts of PreonLab are dependent on the spacing of the particles. As mentioned previously, this is one of the major parameters of the different fluid sources in the program. The value of the spacing set in the solver also decides the spacing of the values in the film wetting. The two parameters specific to the film wetting is the max film thickness and the film absorption rate. Here, the thickness decides the number of particles stacked on each other that provide the film. The absorption rate decides how many particles are absorbed by the film each second.

3.3.3.3.6 Motion

The major advantage of PreonLab and why the software was of interest is that it is very easy and time efficient to create different kinds of motions. The user can simply decide what motion a certain CAD part should have at what time. The customizable keyframe system where the user decides values for different points in time really proves its value when placing motion with different functions. If, for example, the user wants a motion to be fast at the start and slow down by the end of the motion, a sinusoidal wave can be used to create this effect. The motions consist of six parameters, translation in the X, Y and Z direction and rotation around each axis. With such a system, complex movements can be made by adjusting these values accordingly.

3.3.4 **Post processing**

As a final part of the simulation process it is important to decide in what way the results should be presented. PreonLab include many different options regarding how to visualize the results. A big advantage in this software is that the post processing options can be applied after the simulation is run. This means that the simulation does not need be run again with different post processing settings as is required in SCCM. The information in this chapter is gathered from the PreonLab user manual created by FIFTY2 Technology GmbH (FIFTY2 Technology, 2018).

3.3.4.1 Views

PreonLab does not include any sort of video tool where it can create videos from the simulation, which is possible in Star-CCM. Instead, PreonLab uses a slightly different way of capturing the simulations, which is reminiscent of the way that the simulations in this project was captured.

3.3.4.2 Cameras

The user interface provides an easy way of adding different cameras in the simulation space. These cameras can be placed at certain coordinates and the view can be changed either with the mouse or by setting the direction vector that the camera should be pointing. By default, three cameras are already in place when the simulation is created. These cameras are pointed in each direction of the coordinate system. By switching around the cameras one camera can be chosen at a time but there is also the option to have several cameras showing their respective view simultaneously.

3.3.4.3 Clipping object

These are hidden objects that cannot be seen once the simulation is running. They are instead used to hide and cut other models if the inside of the studied item is what is of interest. Clipping objects are either boxes of a chosen volume or an infinite plane depending on which one fits the scenario best. With a connectivity system in PreonLab, the user can show which items should be affected by the clipping objects.

3.3.4.4 Light

For rendering purposes one or several light sources can be added to the simulation space to improve the visibility of the important areas of the simulation. Much like the cameras, these are easy to place and maneuver. Moreover, the color of the light can also be adjusted to make parts easier to distinguish from each other.

3.3.4.5 Renderer

Finally, once all the cameras and scene parameters are set the renderer can be executed. The renderer transforms the particles that can be seen in the simulation to smooth surfaces that represent a realistic fluid flow. Once the renderer is executed, pictures are taken with a time interval decided by the user. To create a video the user then must utilize some third-party software like Windows Movie Maker to put the pictures together into a video.

3.3.4.6 Materials

For visualization there is also the alternative to apply different materials to the different CAD parts in the simulation. The first material included is the Phong material, which is a model that includes parameters related to light source and level of reflection. The second material is called Texture and works by projecting a pattern onto the surface of the model. This pattern can be imported from a picture file format and then applied. The third and final material is Water. Since the program is often used to simulate water flows, this material provides the option to render water in a more realistic way by adding reflections and visual effects that is depending on the depth of the water.

3.3.4.7 Sensors

PreonLab includes simplified monitors compared to SCCM and while properties such as pressure and temperature are not as easily tracked, there are still some tools that can be used for more detailed information. Implemented in PreonLab is a system where sensors are used to track a few physical values, like force load and volume. Sensors are a way of improving visualization of where particles are located at certain points in time by coloring the affected areas depending on what sensor is used. Because this project is revolved around detecting whether there are particles in certain areas, this proves to be a powerful way of monitoring the information of interest. In the following paragraphs the different kind of sensors are explained.

3.3.4.8 Distance sensor

This sensor is not related to the water particles. Instead this tool is used measure the distance between two objects that are initially placed in the simulation space. This way the user can assure that the objects are placed at a distance from each other that correctly depicts the real case that's being studied. By default, the coordinate of the object point is chosen as the point to measure from, but this can be chosen arbitrarily from any point in the models.

3.3.4.9 Volume sensor

The volume sensor provides a way to measure the amount of water inside a chosen volume. This sensor is useful if the user wants to measure things like a fluid flow rate or how much water is entering a certain area. At the time of writing, the volume sensor can only be chosen as a rectangular cuboid, but this shape can be modified in any direction.

3.3.4.10 Wetting sensor

The wetting sensor is a way for the user to make it easier to distinguish where fluid particles are in contact with a specified CAD part. If only one part of a bigger model is of interest, the CAD models of these parts need to be separated before importing them, as specific areas of a CAD model cannot be used. For example, in this project the inside of the trunk hade to be separated from the rest of the car to have the wetting sensor only detecting particles inside the trunk area.

3.3.4.11 Force sensor

The force sensor is a way to monitor what force is acting on an area. The force that can be tracked include pressure force, adhesion force and friction force. The pressure force is defined by the load of the fluid particles acting on a surface. The adhesion force is the force that makes the water particles stick to the surface. One can imagine the difference in fluid flow over a waxed car compared to an unwaxed car. The final force is the friction force which shows much the fluid is slowed down by the surface material.

3.3.4.12 Particle tracker

The particle tracker is a way for the user to easily follow which way a certain particle is flowing. To choose a certain particle, the user can look up the particle ID in the settings and the path of this particle is then colored and lightened up.

3.3.4.13 Pathline

The pathline sensor works much like the particle tracker but instead of tracking a single particle, pathlines can be created for a set of particles. This way the full fluid flow can be mapped to give a simplified view of the simulation is created Figure 3.4 below shows a simulation view without pathlines while Figure 3.5 shows a simulation with pathlines connected to the source simulating rain.

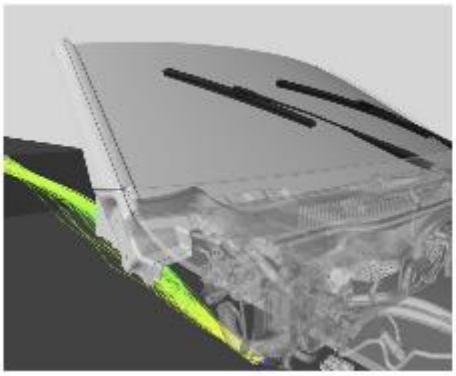


Figure 3.4 Simulation view without pathlines

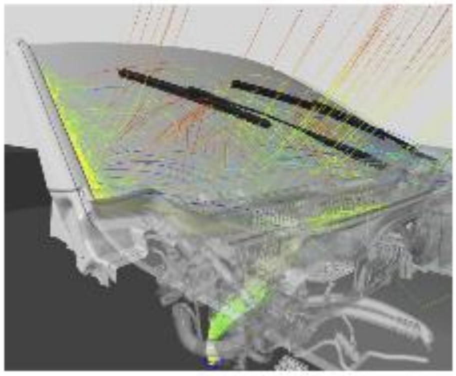


Figure 3.5 Simulation view with pathlines

3.3.4.14 Sensor plane

The final sensor that can be used is the sensor plane. This lets the user place a plane with selected dimensions. For the flow passing through this plane, some properties can then be tracked. These properties include flow rate, velocity magnitude, density and pressure. Once these values have been captured for the simulation they can then be transferred to other software as a text file with the respective value for each time frame.

3.3.4.15 Time

PreonLab utilizes a system with keyframes for the user to easy manipulate values in time. This provides an easy way to keep track and modify what happens at what time in the simulation. In Figure 3.6 below, the user interface for the keyframes is displayed.

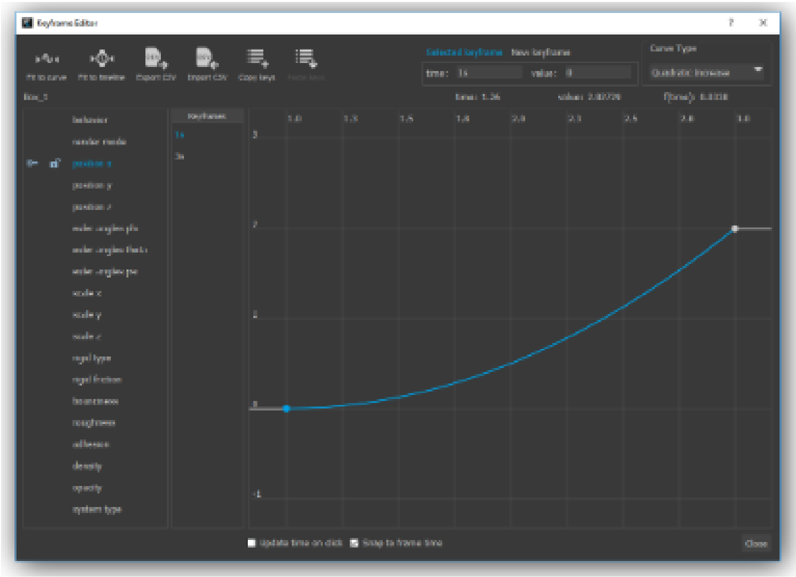


Figure 3.6 Interface of keyframe

The x-axis represents the time while the y-axis can consist of a lot of different values. If a motion is of interest, the coordinates of the moving objects can be chosen for each time frame. If the user wants to have an inlet flow only for the first few seconds, the keyframe editor can be edited to have a flow rate of zero when the inlet should no longer be active. This editor is a very integral part to the simulations performed in PreonLab.

The other part of the PreonLab interface where the physical simulation time is of interest is the simulation recorder. The simulations in PreonLab are performed by recording a solution where the software saves values for each timeframe. Once the values are saved, the simulation can be replayed at any time. The user can decide how many time-frames the simulation should consist of. In Figure 3.7 below the buttons involved in the timeline function are shown. By switching the mode to record and pressing play, the program starts calculating the simulation. Once the selected amount of time has passed, the program stops. The user can then switch the mode to playback to rewatch any part of the simulation.

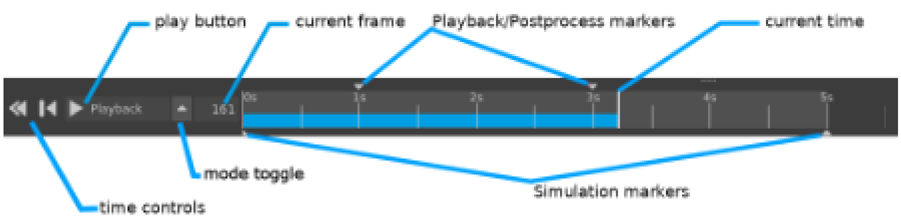


Figure 3.7 Timeline functions

4 Case explanation

In this chapter all cases are explained, what were the challenges and which area were of most interest for each case. Due to confidential material, no values of the requirements will be published.

4.1 Side door

The car models that are provided from Volvo Car Corporation, have two doors on each side with a regular opening. A side door is a part that is connected to both the interior and the exterior environment. This means that the door needs to be divided into a wet and a dry area, where requirements could be set for each area. These two areas are illustrated in Figure 4.1. All surfaces where interior would be places should be dry, which means that the other side of the door structure in the figure is the wet area.

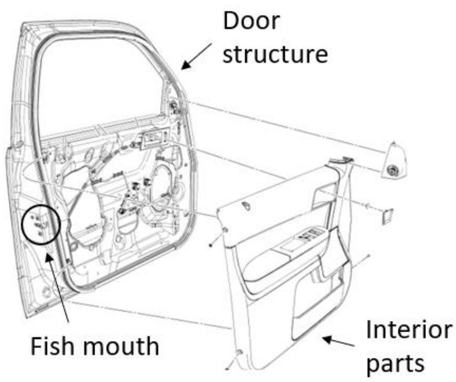


Figure 4.1 Side door structure

The area that is of interest in this report is the wet area, where the latch system and the fish mouth are located. The fish mouth is the opening area into the side door which makes it possible for latch system to connect to the body structure when the door is closed and locked.

The water is entering the wet area through a gap between a horizontal rubber lip and a vertical bar, shown in Figure 4.2.

Figure 4.2 Illustration of the gap

The size of the gap could differ between car models, but together with the Water Tightness Department at Volvo, a mean value has been determined and has been used for all door simulations.

The reasons for why the wet area is of interest is because Volvo Car Corporation have found that too much water is flowing onto the latch system and through the fish mouth. In cases where the temperature is low, the lock system or even the whole door has sometimes frozen in a closed position. Sometimes ice has fallen onto the owners when opening the door. If the temperature is higher, the owners could instead get a volume of water on them while opening the side door. All these situations occur due to the water flowing through the fish mouth and stagnating between the door and the body structure.

The focus area has been the same for both the front and rear door simulation, even though the geometry has changed slightly between the geometries.

4.2 Trunk

The trunk of the car is generally either a sedan or a station wagon. This will have a big impact on the geometry and function of the trunk. On a sedan version the trunk tends to be much smaller and the opening motion of the trunk differs a lot from a station wagon model. Whether this is problematic is a point of consideration and one cannot expect to draw any conclusions regarding a station wagon from the results in this report. However, the difference between the different sedan models are small enough that the case described in the report could likely be applied to any Volvo Cars sedan model to create a realistic simulation.

The major problems with the trunk consists mainly of water entering the luggage area. This is highly unwanted and water entering the luggage area means that the owner of the studied car model could have his or her items destroyed if they are sensitive to water. This means that it is of great importance to review the drainage capacity of the trunk in an early stage, to get an idea of what the water flow might look like.

It should be noted that the water of interest is the water that stagnates on the trunk once it has stopped raining or after washing the car. There is no point in trying to study whether water enters the trunk while it is raining as it would be impossible to prevent this.

There are two ways for the water to enter the luggage area. While opening the trunk, the water will be flowing down the trunk, hit the window and keep flowing towards the luggage area. If the amount of water is high enough or if it reaches a high enough velocity it might then enter the luggage area in this manner. This is shown in Figure 4.3. The other possibility is that the water runs down the sides of the trunk and that this flow might end up inside the trunk.

Figure 4.3 Trunk luggage area

4.3 Sunroof

The sunroof also has a drainage system that takes care of the water that would otherwise be stagnating on the roof of the car. This drainage system consists of four pipes, one on each side in the front and back, shown in Figure 4.4. The left and right version of each pipe should be completely symmetrical, but all the pipes will be studied since there are separate CAD files for them. This could also identify mistakes in the CAD files, should the results differ too much.

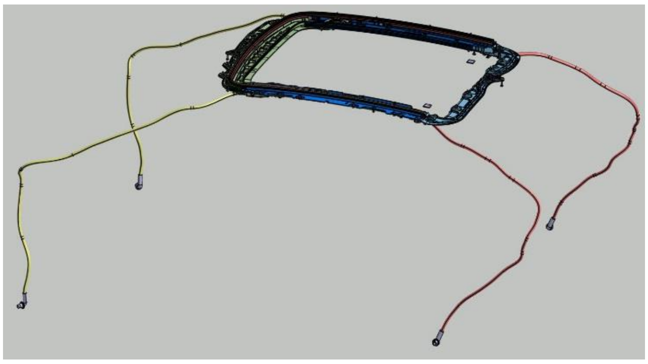


Figure 4.4 Drainage pipe system

There are two major points of interest regarding the drainage system. Firstly, whether the drainage system can drain a specified amount of water within a limited time. This is a threshold set by Volvo Car Corporation and provides the baseline of what the drainage system needs to be capable of. In some cases, objects like leaves might enter one of the pipes and it is then important that the other pipes can handle the reduction in overall drainage capacity. The other major point is whether there is water stagnating somewhere within the pipes. This is unwanted since this tends to create noise and during low temperatures the stagnated water might freeze and block the drainage system.

In reality, all the pipes work with a shared channel, but the specifications apply to each pipe and it is very difficult to study each pipe separately if the pipes are still connected to this channel. Because of this, the pipes have been separated into four different simulations in this report. The inlet to the pipes from the channel are designed in a way that creates an initial velocity as it is slightly tilted downwards.

5 Geometry and setup

This chapter is showing what sequences each case has been put through. It will explain the progression and what has been taken into account.

5.1 STAR-CCM+

The working process is the same for each case using SCCM which looks like Figure 5.1. These three activities are where all the practical work is required. Once the meshing is done the simulation can be performed without any user interaction.

Figure 5.1 Working process using STAR-CCM+

5.1.1 **Geometry**

The initial process regarding the geometry are the same for all cases. All geometries are retrieved from the software called Team Center. This is a platform where information and CAD geometries are stored. The geometries are later imported into CATIA where the CAD cleanup is performed. The purpose of doing CAD preparations for all cases is to create a closed volume as close as possibly to the geometry of interest. The geometries of the door cases are also modified in the CAD inside SCCM. In subsection 5.1.1.1 - 5.1.1.3 is a more detailed CAD preparation explanation of each case.

In this chapter different tools are used in order to create new surfaces or modify already existing surfaces. Tools inside CATIA that are used in purpose of splitting and deleting surfaces and parts are listed in Appendix B.

5.1.1.1 Side door

The focus is similar for both the front and the rear door. The aim is to remove and simplify parts that do not affect the simulation result. This will lead to a less complex model which will be easier to handle in the upcoming steps. The way of cleaning up the front door from any unnecessary CAD details is different from the rear door.

The purpose regarding the front door is to clean up as much of the redundant geometry details possible and only study areas close to the latch. The difference for the rear door is that the CAD preparation should be performed to have the possibility to simulate a complete water flow inside the rear door. This requirement limited the ability to cut the geometry as much as the front door.

5.1.1.1.1 Front door

All CAD parts that involve the front door are gathered from Volvo Car Corporation, which creates an initial geometry that is imported to CATIA. Figure 5.2 -Figure 5.4 is showing different view of those parts that is selected.

Figure 5.2 Front door, outer view

Figure 5.3 Front door, side view

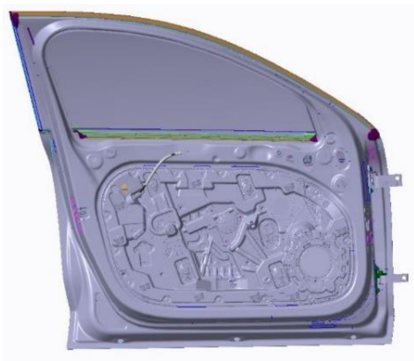


Figure 5.4 Front door, back view

When the geometry is imported into CATIA the first thing to do is to remove parts that are not of interest to the simulation. This is done by using tools inside CATIA, tools that can handle those operations are listed in Table 0.1 in Appendix B.

This tool is used for the upper section of the front door, because it is not of interest to study the water flow upon the upper window section. The same reasoning applies to the front of the front door, which makes it possible to cut the door into half vertically.

A result of what parts that should remain for the simulation is illustrated in Figure 5.5 - Figure 5.6.

Figure 5.5 Cut front door, back view

Figure 5.6 Cut front door, outer view

To get a better view of what parts that should remain inside the door the outer metal sheet on the backside has been hidden in Figure 5.7.

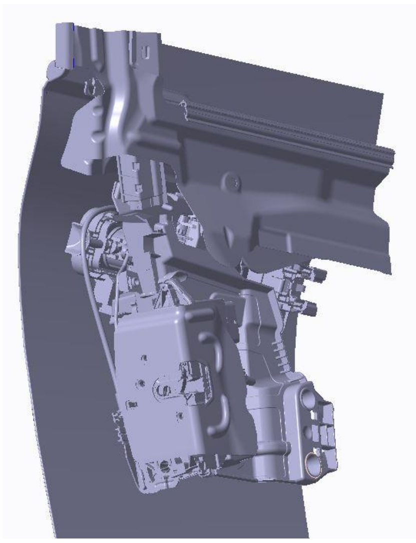


Figure 5.7 Outer metal sheet is hided.

When all unnecessary parts are removed, it is time to make sure that the geometry is similar to the real-life case. One area of the front door that has to be modified is the area by the window and the rubber strips. The reason for modifying this area is because the rubber strips have a solid structure in the simulation environment while they will be flexible in reality. Due to this, all rubber strips that intersect with the window will be removed. This process is shown in Figure 5.8. The parts of the geometry that are highlighted green is deleted.

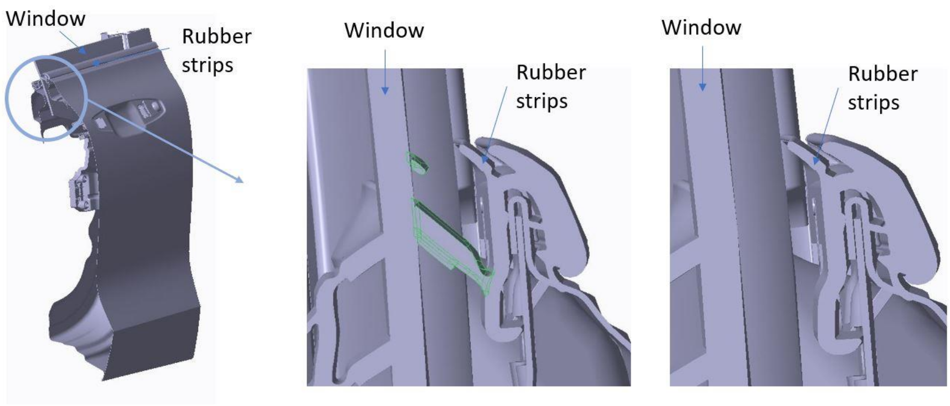


Figure 5.8 Deleting intersecting rubber strips.

The next step is to close the geometry and create pre-defined surfaces acting as inlets and outlets, this is done by tools inside CATIA. Tools that are used are listed in Table 0.2.

The first surface to create is the one that is going to act as a water inlet located on the outer side of the door. This surface is extruded from the top of the window. In order to have the possibility to adjust the area of the inlet one more surface is extruded with same dimension on top of the first one. This creates the opportunity to adjust the width of the water inlet by using either only the first extrusion or both extrusions. Upon the second surface a third surface is extruded with the purpose of acting as an air inlet. This surface is created in the same direction as the two previous ones. Next up is to create the outlet located at the very bottom of the geometry on the outer side. This is created by the tool "extrude surface" with a reference to the bottom edge at the outer side of the door. The width of the outlet is the same as the surface at the top, in the same direction as the inlets at the top of the front door. The result of creating inlets and one outlet located on the outer side is illustrated in Figure 5.9. The blue area is the surface that is created first and the gray in between the blue and the red is the second. The surface highlighted with red is the third, and the orange is the fourth acting as an outlet. The blue inlet will then inject the water once the simulation is started while the outlet will make sure that the water can exit the simulation if it reaches the bottom.

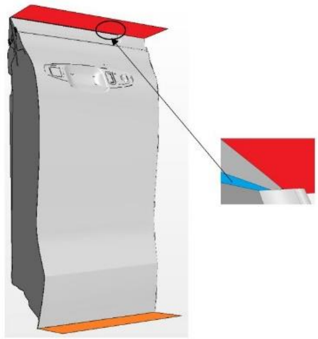


Figure 5.9 Inlets and outlet on the outer side of the door

To prevent from collecting water inside the door is an outlet created on the backside, shown by orange in Figure 5.10.

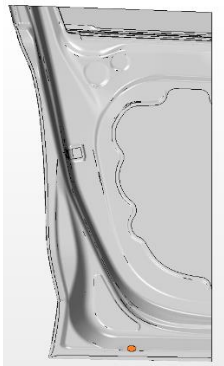
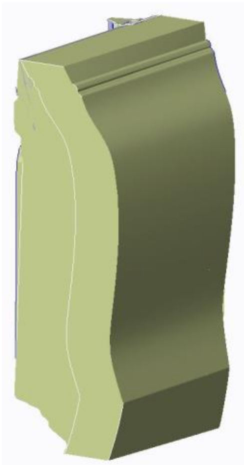
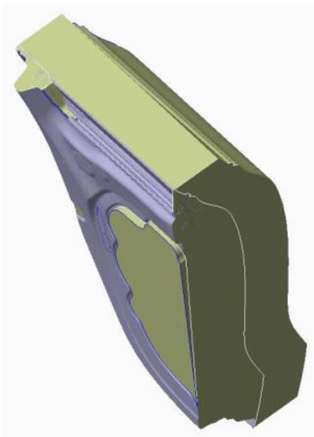


Figure 5.10 Outlet on the back side of the door

After creating surfaces with a special purpose, it is time to close the geometry with additional surfaces. Tools that is helpful to use is the same as within Table 0.1, Table 0.2 and some additional listed in Table 0.3.

An advantageous method of closing the geometry is to create lines that are connected to each other as a loop and placed in the same plane. By joining these lines together as one single loop it is possible to create a surface inside the contour of the loop, due to the tool “fill hole”. The result of a closed geometry is illustrated in Figure 5.11 -Figure 5.13. All surfaces that are highlighted in yellow are surfaces that are created to close the geometry.





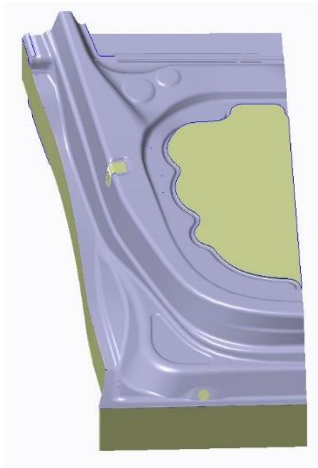


Figure 5.11 Closed geometry of the front door, outer view

Figure 5.12 Closed geometry of the front door, side view

Figure 5.13 Closed geometry of the front door, back view

The last adjustment of the geomety is to recreate a realistic sealing strip in between the window and the outer part of the front door, one additional surface isrequired. This sealing stripe is created inside SCCM by using the tool “Offset/Surfaces”. With this function it is possible to extrude a surface with a direction towards the window and slightly upward, with reference to the outer part of the front door. The surface will be completly tight against the window at the cut section. While a gap starts to widen in between the window and the surface towards the main gap where the surface stops before it hits the vertical bar. The process of the regeneration of the sealing stripe is illustaded in Figure 5.14

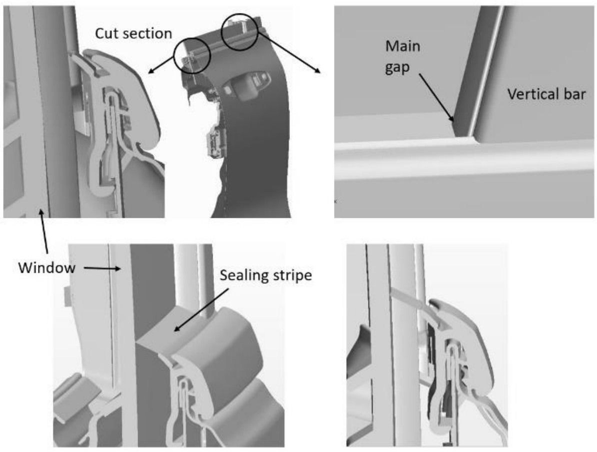


Figure 5.14 Illustration of regeneration of sealing stripe

5.1.1.1.2 Rear door

The structure of the procedure is mainly similar to the front door. The biggest difference is how the cutting process is performed. Those parts that are relevant for the rear door are shown in Figure 5.15 - Figure 5.17 by different views. All parts are imported from Team Center into CATIA to be modified further.

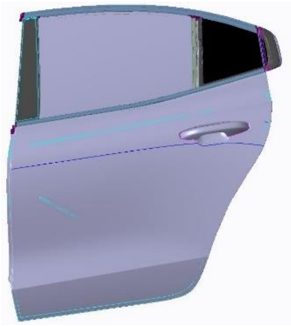


Figure 5.15 Rear door, outer view.

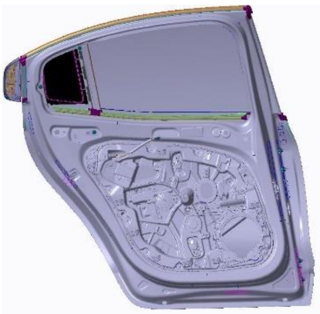


Figure 5.16 Rear door, back view.

Figure 5.17 Rear door, side view.

The CAD preparation of the rear door needs to be performed a bit different in comparison to the front door. To be able to study the complete water flow inside the rear door. The only par that was acceptable to cut off was the upper part of the window, which does not give any vital information to the simulation. By doing this cut section the simulation can be slightly reduced.

Unnecessary parts that do not bring any additional information to the result, can be deleted. Both the cutting and deleting procedure is done in the same way as the front door, by using tools listed in Table 0.1. The result of deleting and cutting of the geometry is illustrated in Figure 5.18 and Figure 5.19.

Figure 5.18 Cut rear door, outer view

Figure 5.19 Cut rear door, back view

To get a better picture of what parts that were of interest to the simulation, study Figure 5.20 Cut rear door, hided metal sheet Figure 5.20. This figure is hiding the sheet metal at the back side, which allows to see the remaining parts in more detail.

Figure 5.20 Cut rear door, hided metal sheet

The rear door has sealings that have intersections with the window. Those sealings needs to be deleted in the same way as for the front door, which is illustrated in Figure 5.8.

When the sealings in the intersection are deleted it is required to generate a new one that illustrates the reality in a better way. This was briefly mentioned earlier, but since the sealings are made from rubber, their position in the CAD model does not represent the real-life case. It is therefore necessary to adjust the position of this sealing so that it better represents the physical testing case. This surface is created inside CATIA, with the tools in Table 0.2.

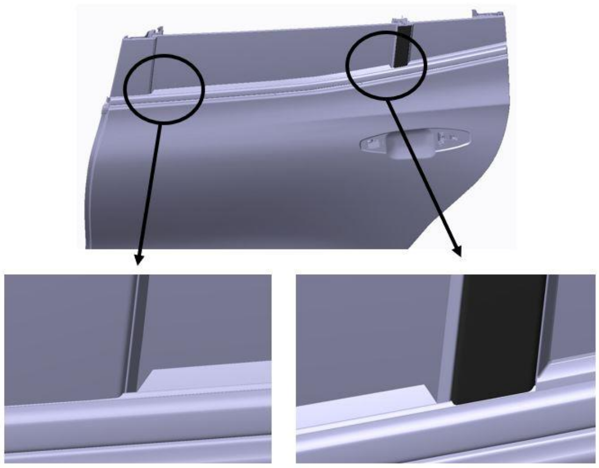


Figure 5.21 Closer view of the main gaps

When only the parts that are necessary for the simulation are remaining, it is time to close the geometry. The same method is applied to the rear door as for the front door, which contains tools listed in Table 0.2 and Table 0.3. This time, however, the rear door is not completely closed when imported to SCCM as the outer side is missing one surface to close the outer volume of the geometry. This surface is created inside SCCM. The result is shown in Figure 5.22 - Figure 5.24.

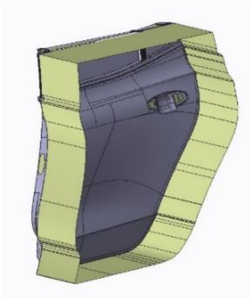


Figure 5.22 Closed geometry of the rear door, outer view.

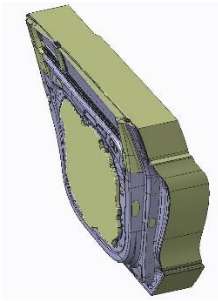


Figure 5.23 Closed geometry of the rear door, side view.

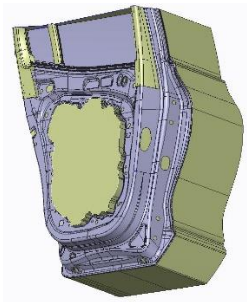


Figure 5.24 Closed geometry of the rear door, back view.

Before creating the surface to close the outer volume in SCCM one additional adjustment is required in order to close the complete geometry. This is done in CATIA before importing to SCCM. On the back side there are gaps/holes that need to be covered. Areas that need to be checked for gaps/holes are highlighted with blue in Figure 5.25. It is important to close all gaps before importing the geometry. This is done by using the same tools as before.

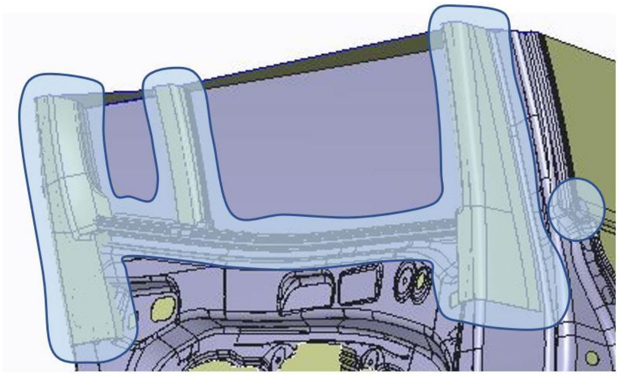


Figure 5.25 Area for checking gaps and holes

The geometry is completely closed by creating the surface highlighted as purple in Figure 5.26 by using the tool "Repair surface" inside SCCM.

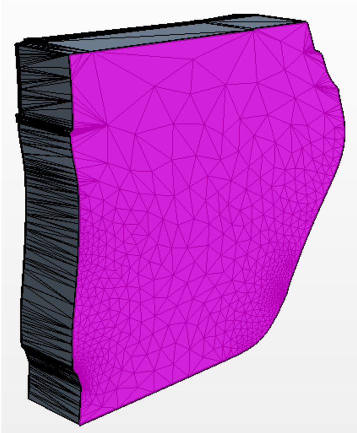


Figure 5.26 Complete closed volume

5.1.1.2 Trunk

The focus when exporting the geometry is that no parts are missing at the upper region of the rear of the car because the water is expected to be flowing in this area. The geometry that is exported from Team Center to CATIA is shown in Figure 5.27.

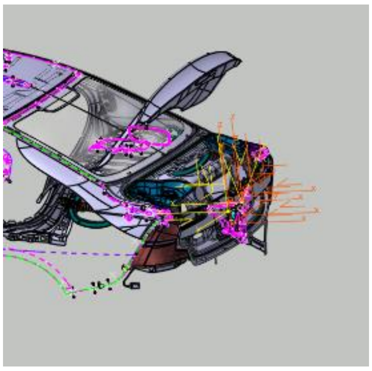


Figure 5.27 CAD exporting from Team Center

Areas of interest regarding the trunk case, are those where the water can flow and later on possibly enter the luggage area. In order to only simulate areas of interest, the geometry is reduced by a process that divides the exported geometry into two areas. One that is of interest and one that can be deleted. The area of interest can be studied in Figure 5.28, which also shows the plane that was divided in the geometry.

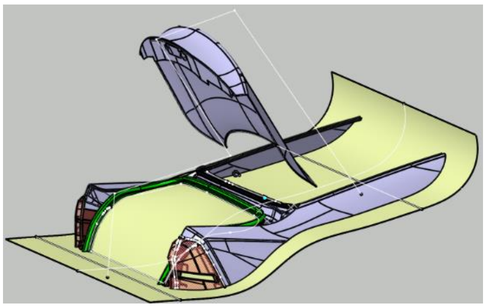


Figure 5.28 Trimmed geometry

Since the remaining geometry have symmetry, the geometry can be reduced once more, which will make the simulation faster. One other adjustment is to delete all inner parts since the main water is flowing on the outer surfaces, which make the inner parts unnecessary to keep.

It is also necessary to close some smaller gaps between the water drainage located at the sides of the luggage area and the lamp. The result of cutting the geometry and deleting parts can be studied in Figure 5.29 and Figure 5.30.

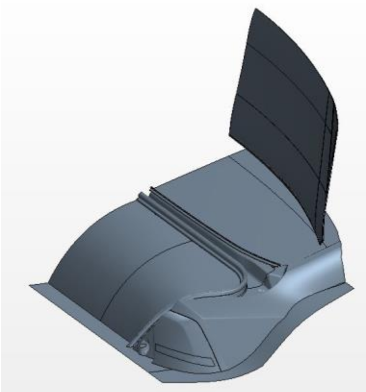


Figure 5.29 Cut and trimmed geometry, view 1

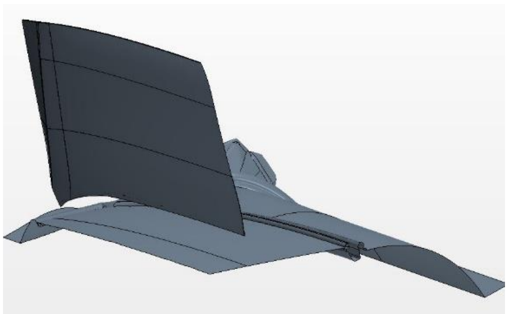


Figure 5.30 Cut and trimmed geometry, view 2

When the geometry size is optimized, the creation of the closed volume can be started. First surface to create is a thin surface that extends across the entire top of trunk. The surface is extruded in a direction which is normal to the surface of the trunk. This surface is created to act as a water inlet. On top of the newly created surface one more surface is created with the same dimensions and direction, this makes it possible to change the area of the water inlet easier in the setup. The surface acting as a water inlet can be illustrated in Figure 5.31.

Figure 5.31 Creating surfaces acting as water inlet

With water inlets finished, it is possible to create a closed volume. The surfaces that create the volume do not have any specific dimensions predetermined. But it is important to create boundary conditions that do not impact the water flow while trying to keep it as small as possibly to minimize the volume due to simulation time. A closed geometry volume is shown in Figure 5.32 and Figure 5.33.

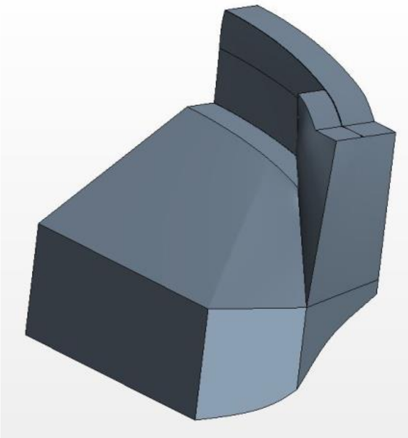


Figure 5.32 Closed geometry volume, view 1

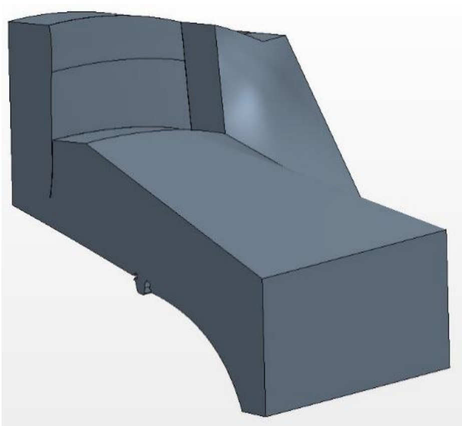


Figure 5.33 Closed geometry volume, view 2

5.1.1.3 Sunroof

All the pipe structures are very similar to each other, which means that the simulation process is very similar for all the different pipes. In this subsection the method is explained.

When studying the water capacity of the drainage pipes, it is important to select all parts that are in direct connection to the pipes. This is done in order to get an understanding of how the boundary conditions should be setup at the ends of the pipe. As there is a requirement for each pipe, the full structure of the sunroof cannot be used. This would lead to the pipes sharing the water channel and the flow might become divided in uneven ways between the pipes. Instead the pipes are separated from the CAD file and split into different simulation scenarios. The imported geometry is illustrated in Figure 5.34.

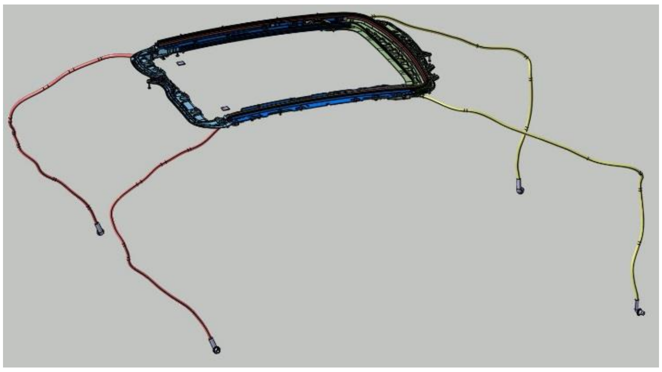


Figure 5.34 Exported geometry from Team Center

Inside CATIA the model is trimmed to be able to study one pipe at a time, in this example the left front is selected. What to do next is to create a plane that separates the left front pipe from the rest of the parts. Only the inlet pipe from the water channel should remain. The tools that are used are listed in Table 0.2 Adding lines and surfaces to the geometry. An illustration of the result can be seen in Figure 5.35.

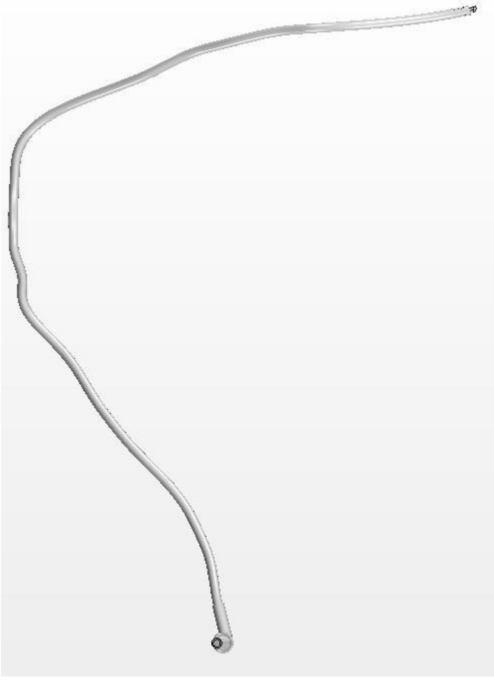


Figure 5.35 Cut left front pipe

At the end of the pipe, a surface is created to enclose the volume of the pipe but also to act as an outlet. The inlet pipe is cut by a plane at the same angle as the water channel was located. The reason of cutting the inlet pipe is that a water tank will be mounted at the inlet pipe. To study the resulting section of the inlet and the outlet, study Figure 5.37 and Figure 5.36

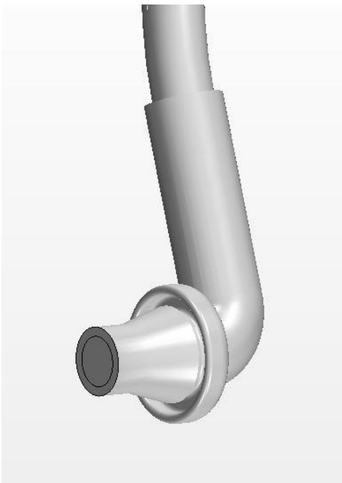


Figure 5.36 Closer view of the outlet of the pipe

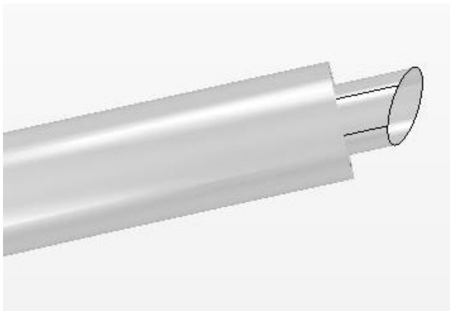


Figure 5.37 Closer view of the inlet of the pipe

The water tank is created and mounted on the inlet pipe. To make sure that all the water that is contained in the tank will stream towards the inlet, the bottom inclination is towards the pipe. The tank is created to have the volume of what the pipe should manage during a specified time. The result of the water tank can be studied in Figure 5.38 and Figure 5.39.

.

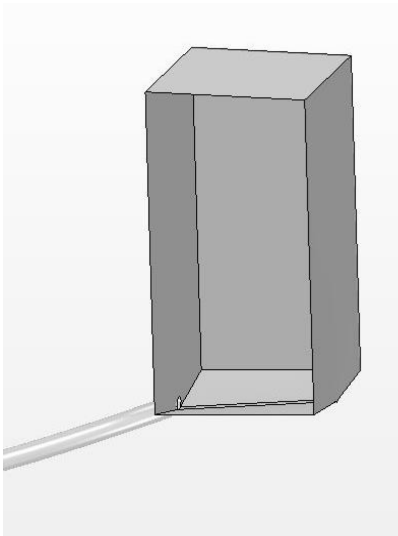


Figure 5.38 Water tank, overview

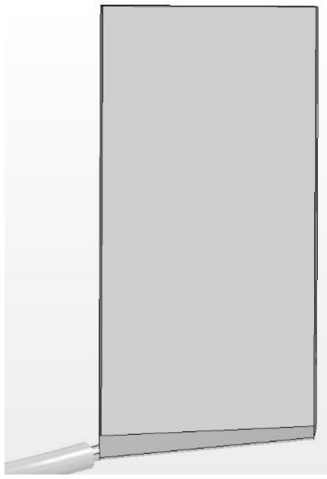


Figure 5.39 Water tank, sideview

5.1.2 **Surface wrapper**

This chapter will work as an extension to the values presented for each car part in the template. For the exact values of the Surface Wrapper (see subsection 2.4.4.3) settings, please study the corresponding Template attached in Appendix C.

5.1.2.1 Side door

While both side doors fulfill the same function and suffer from the same problems with water running out the fish mouth, the shapes of the doors are slightly different. This means that some considerations must be made regarding these differences when it comes to the Surface Wrapper. The following subsections provide an overview of what the differences in geometry mean to the overall work process of the side doors.

5.1.2.1.1 Front door

Both side doors require quite a fine surface wrapper. Some of the parts, specifically the latch and the inside of the window bars, contain such fine details that a coarser wrap will result in a lot of parts blending together. In parts that are of little importance to the simulation, for example the parts on the lower side of the door, this does not contribute to any additional problems. For the upper part, however, the appearance of blending parts might have a big impact of the simulated flow.

The most important thing in the front door is to make sure that the contact prevention and surface controls are chosen in a smart way. The contact preventions generally result in much higher execution times of the surface wrapper. This is especially true for contact preventions with very low thresholds. Therefore, these should be kept to a minimum. It should still be noted that in some cases it is absolutely necessary to use them. For the gaps in the rubber lip a contact prevention must be used to ensure that the gap is retained after the surface wrapper is executed. Since this is integral to the simulation it is recommended to put in some extra time to make sure these values are chosen low enough to retain the gaps but high enough to not create very long execution times.

As for the surface controls, these are of high importance on parts where the surface itself is important to keep intact. The most important part is the mass inlet surface. If this surface is obstructed in any way this may create initial flows with unwanted trajectories and reversed flows. The user should also make sure that the parts where the main water flow will be going should be of high quality. This, however, is hard to judge at an initial stage and might require some trial and error if the model is not very similar to a previous version. In Figure 5.40 below, the rail that leads the water along the designed way is displayed. This is an example of an area where it is very important that the surface is not obstructed as this could have a big impact on the flow.

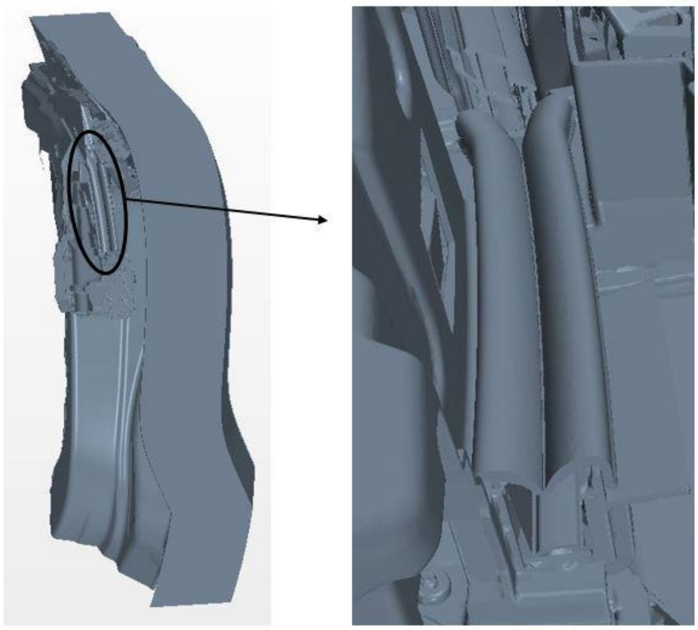


Figure 5.40 Closer view of the rail

5.1.2.1.2 Rear door

Most of the concepts explained in the front door section of this chapter are applicable to the rear door as well. The inner layouts are quite similar which means that the rear door also consists of a lot of smaller details in the lock and latch parts where many intersections will occur if the user is not careful.

The biggest difference in the CAD models is that the window is here divided into two windows, one smaller and one bigger. The smaller window uses a molding on the bottom side which means that no water will enter this way. The careful approach that is taken when wrapping the gaps of the front door are therefore not as important for the small window. Because of this, even though the rear door has a different layout of the windows, the workflow is very similar.

5.1.2.2 Trunk

The trunk has some characteristics that make the wrapping very different from the side doors. This CAD models include basically no complex parts. It also includes parts, like the trunk and the window that are made up of one big plain part. These parts are quickly handled by the wrapper which provides a perfectly clean surface. In this case no contact preventions are needed to create an adequate surface for the simulation.

There are some parts that need some fine-tuning. Just like the doors, the inlet has a tendency to start blending in with the trunk that it is in contact with. Because of this it is required to have quite a fine surface control. Star-CCM also provides an option within the surface control that applies the surface control to the contact between the chosen parts rather than the surfaces. In the case of the mass inlet, this option has sometimes provided good results and it is recommended to use it to avoid using contact preventions.

Another part that provided some difficulties was the rubber lip that outlines the luggage area. This rubber tended to blend in with the surrounding metal. As this area is one of the more important areas for the water flow, the user must make sure that the relevant parts are not blending with each other. As shown in Figure 5.41 some contact issues remained even after the surface controls were adjusted, but these were deemed small enough to not have any major impact on the flow.

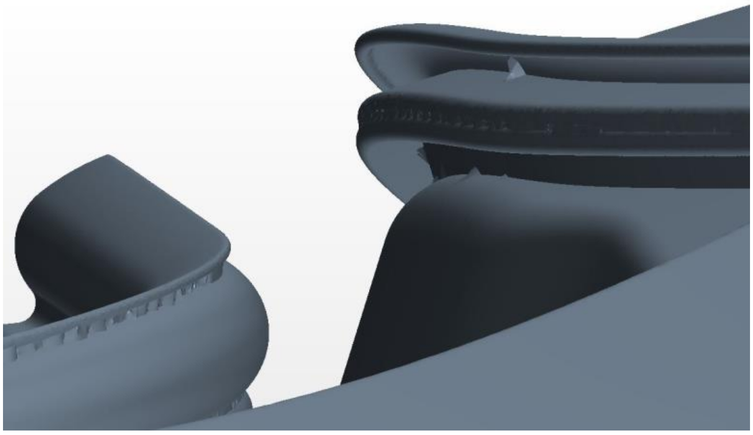


Figure 5.41 Closer view of surface control adjustment

5.1.2.3 Sunroof

For the drainage pipes the surface wrapper is simple. As studies of pipe flow is an integral part in any CFD software, the program handles the studied pipe very easily. The box that is attached to the pipe is also simplistic enough not to introduce any major issues for the wrapper.

The only problem when wrapping the pipe is the area where the rubber pipe is mounted onto the sunroof channel. As there is some tolerance allowed in the CAD, the pipe is not completely closed around the box nozzle. There is some judgment to be made around the most optimal way to work around this problem. The way it was handled in this project was by closing this gap in the CAD. The disadvantage of this is that the model will not be completely realistic as the pipe will not be closed around the mount. Another way to handle it could be by working with the settings in the surface wrapper. A coarser surface wrapper in this area, for example, could result in a more realistic model. Some of these options were tried during this project, but with no success the CAD fix was decided to be more time-efficient. This area is illustrated in Figure 5.42 where the rubber pipe and sunroof channel are connected.

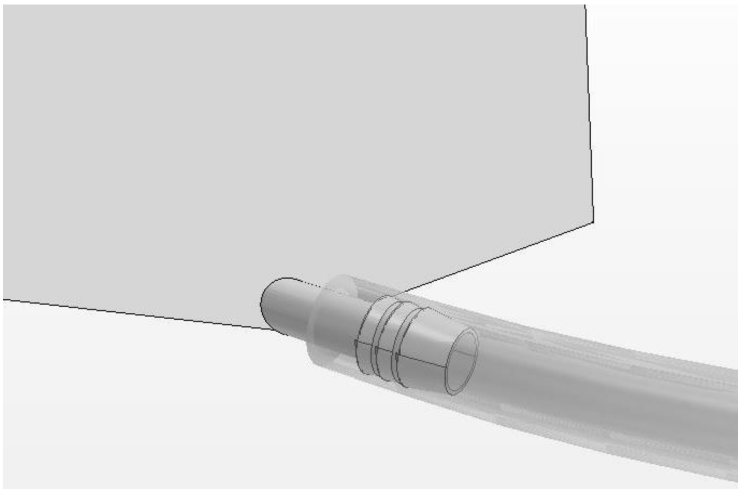


Figure 5.42 Area of the connection between the rubber pipe and sunroof channel

5.1.3 **Volume mesh**

5.1.3.1 Side door

The same scenario where the difference in geometry of the side doors provides some considerations regarding the Surface Wrapper, this applied to the volume mesher as well. The following subsections provide an overview of what the differences in geometry mean to the volume mesher and the related settings.

5.1.3.1.1 Front door

The meshing of the side doors in general provided the biggest obstacle in this project. The small detailed parts of the interior parts of the door meant that many intersections were created from the surface wrapper which then had to be repaired or altered. Smaller parts, like the ones mentioned, also means that a smaller mesh size, especially in these areas, is required. If the user is not careful, information might be lost in the mesh where it is very detailed. This also creates an important trade-off situation where the number of cells is already very high with default settings of the mesh. With such a high number of cells the simulation takes a lot of time. This is why the trimmed mesher was chosen. According to Siemens, the trimmed mesher results in a mesh that is faster to generate inside SCCM. Not only is it faster to generate but the simulations run with a trimmed mesh are also faster than both polyhedral and tetrahedral meshes. This is because the trimmed mesher has a very efficient structure that allows the cells to be distributed in a manner where relatively few cells are required while the accuracy is still maintained.

It was established early on that the important part of this car part was to study how the flow was behaving around the latch. For this reason, a mesh along the path that the water was expected to go was refined and made with greater detail. By using blocks created in the 3D-CAD in Star-CCM, this path could be roughly boxed in. With the help of volume controls in the meshing tool, these boxes could then be refined to different levels depending on the complexity of the flow in the boxed region. The Figure 5.43 and Figure 5.44 below shows that the flow path was made up of five boxes where the mesh was increasingly refined.

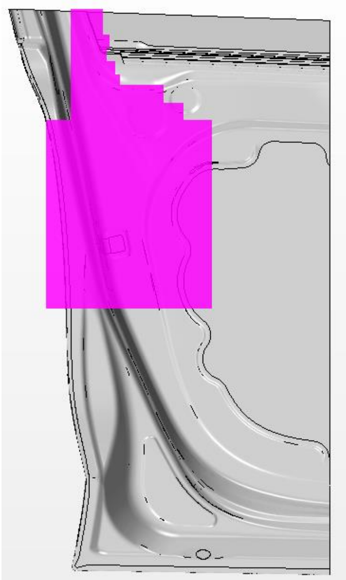


Figure 5.43 Volumetric control, of the front door, back view

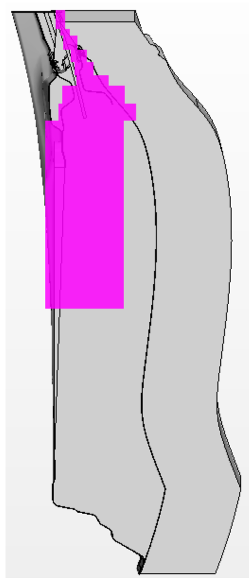


Figure 5.44 Volumetric control of the front door, side view

With the CAD model provided it seems inevitable that certain cells in the mesh start acting up, causing the solution to diverge and a floating-point exception. For this reason, it is a good idea to make sure to monitor the Convective Courant Number. This is a function that calculates the velocity of a flow in a cell compared to the distance between cells in the local area. If this number is too big it means that the flow will be skipping cells and consequently this will simulate an unrealistic flow. If the number is too small, it means that the cells in the mesh might be unnecessarily small and the size could be increased to reduce simulation time.

It is also recommended to monitor the y+ value. For more information on this value, study the Theory chapter. What this means is that if the user finds that a majority of the prism layer cells has a y+ value between 5 and 30, the prism layer needs to be reconstructed. As shown in the theory, values that are within this limit will not be modelled correctly and since a big part of the flow in this project is close to walls, this will have big impact on the general flow.

Because of the very fine details in the latch there have been several scenarios where a cell was not detected as a bad cell but still a monitor of the velocity in each cell showed some extreme velocities. In this case the user needs to manually remove these cells. This is done by creating a cell set based on velocity and set a threshold of an unreasonably high number. In this project, a pair of cells were found to have velocities of about 80000 km/h which caused divergence. As soon as these cells were singled out and deleted the simulation could be run without issues.

5.1.3.1.2 Rear door

As with the surface wrapper, most of the concepts in chapter 5.1.3.1.1 about the front door are also applicable to the rear door. Still, there are some differences.

As the rear door was decided not to be cut down and kept in its entirety, this means that the CAD model of the rear door will contain a bigger volume than the front door. The front door, even though it was cut down, was still a simulation that took a lot of time to run. This means that as the rear door is bigger, this puts bigger requirements on optimizing the mesh size. To keep the number of cells down, a lot of work was put into making sure that the refining blocks covering the flow path enclosed the flow as closely as possible. The smaller blocks are shown in Figure 5.45 and Figure 5.46 below. For comparison to the front door, study Figure 5.43 and Figure 5.44.

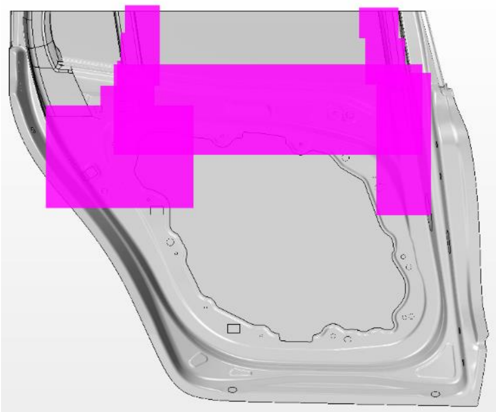


Figure 5.45 Volumetric control, of the rear door, back view

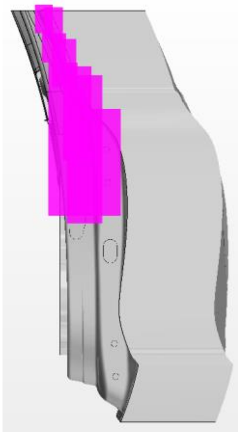


Figure 5.46 Volumetric control, of the rear door, side view

5.1.3.2 Trunk

Creating an initial mesh for the trunk proved to be of little difficulty. With the relatively plain structure that was formed by the CAD, there were very few complex geometry shapes to cause issues in the mesh.

Once the simulation started running it became obvious that some changes had to be made as the water flow was behaving unrealistically. As the water was leaving the edge of the trunk the water started diffusing into the bigger mesh cells and in the simulation it appears as if the water is disappearing. These issues were caused by the fact that the mesh cells of this area were too big. When the water was split into bigger and bigger cells, the volume fraction of water in each cell became so small that it was no longer considered a water-filled cell. When a finer mesh was applied, this problem was solved.

A lot of work had to be put in to make sure the wall treatment on the trunk was working properly. It could be seen with the y+ value monitor that the values were fluctuating a lot. With the settings provided in the Trunk Template of Appendix C good wall treatment could be applied with good results.

Another area of a lot of discussion was the area around the rubber lip. To make sure that the flow was behaving in a realistic manner, the mesh had to be very refined. It would seem like this rubber edge had to be followed very closely by cells not to have water diffusing into cells and "disappear". Thanks to this requirement, the number of cells became quite high which severely slowed down the simulation. Figure 5.47 shows an overview mesh of a plane section.

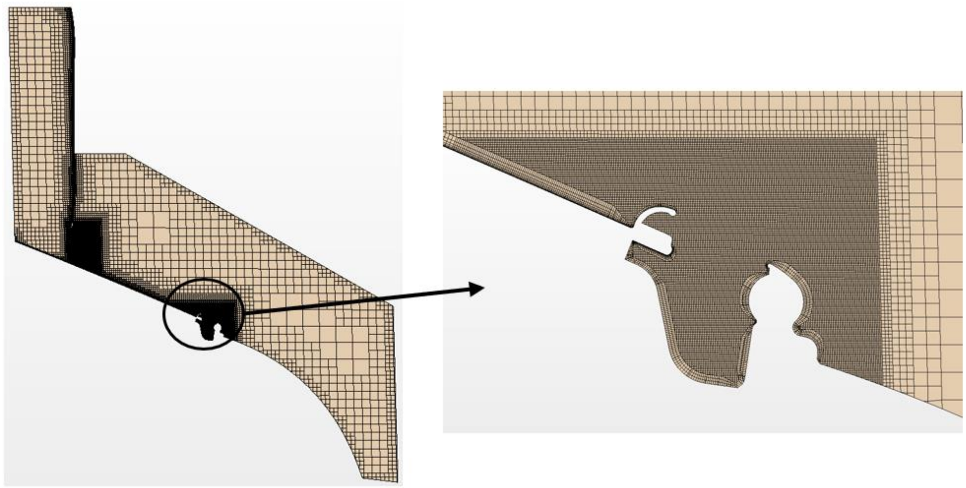


Figure 5.47 Overview mesh of the trunk

5.1.3.3 Sunroof

The meshing of the pipes is quite different from the other car parts. Since this simulation will have a big emphasis on pipe flow, the recommended settings from Siemens for a similar pipe flow was used as an initial template. In both the door and the trunk simulations the flow was expected to be more of a top to bottom flow, in most cases going downwards. This, together with the fact that the models included a lot of cells made the trimmed mesher a solid choice.

For the pipes the walls and their respective wall treatment functions will have a big impact on the flow as the diameter of the pipe is relatively small. It is therefore important to have a prism layer mesher included to make sure that the cells have a structure that is aligned with the wall. From Figure 5.48 one can see that the circular area that is left inside the pipe would be hard to fill with trimmed cells without creating a big difference in the sizes of the cells near the edge. For this reason, a polyhedral mesh was instead chosen. Another reason for using the polyhedral is that the advantage of the trimmed mesher being faster to execute becomes insignificant as the mesh in the relatively small CAD model of the pipes is very fast regardless.

As for the general philosophy of the meshing size, the mesh size was chosen to be on a coarse level. Since there is no interest in studying the flow very closely, but rather get a more general picture of the measured flow rate it was deemed unnecessary to go with a finer mesh. The simulation had to be run for a relatively long time as well, which means that a fewer number of cells speeds up the simulation to some extent.

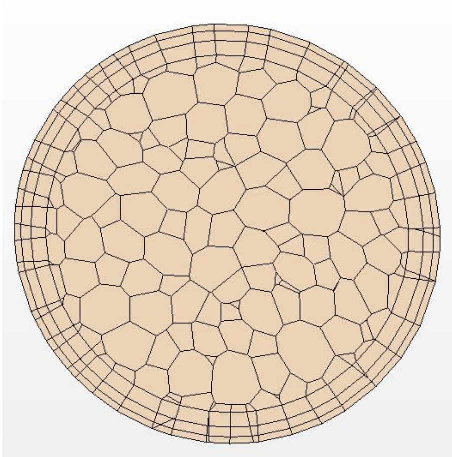


Figure 5.48 Mesh structure inside the pipe

5.2 PreonLab

5.2.1 **Geometry**

5.2.1.1 Side door

With the different structure of the PreonLab software compared to SCCM, the setup for the simulation is simplified. The only modification that needs to be done to the CAD is to create the slots that are formed at each side of the window. Since PreonLab requires the gaps to be twice the size of the particle for the fluid to enter there is no need to keep the gap along the window. If the particles were to be small enough to enter the gap, the number of particles would severely impact the simulation time. The following subsections will explain the CAD preparation in order to simulate inside PreonLab.

5.2.1.1.1 Front door

As mentioned previously, only the gap needs to be added to the front door. The gap is created in the edge between the vertical bar and the horizontal rubber lip, the position of the gap is marked by a black circle in Figure 5.49. Once this has been created, the door can be imported into PreonLab. For post processing it is recommended to have the CAD parts separated into smaller groups as an imported CAD model in PreonLab cannot be divided into smaller parts. If the user wants to add a sensor to some part to easier visualize where the particles interact, this part must be separated from the rest. In this project the latch and other internal parts like the lock were separated from the door structure to improve the visibility. The imported CAD model for the outer front door structure is shown in Figure 5.49.

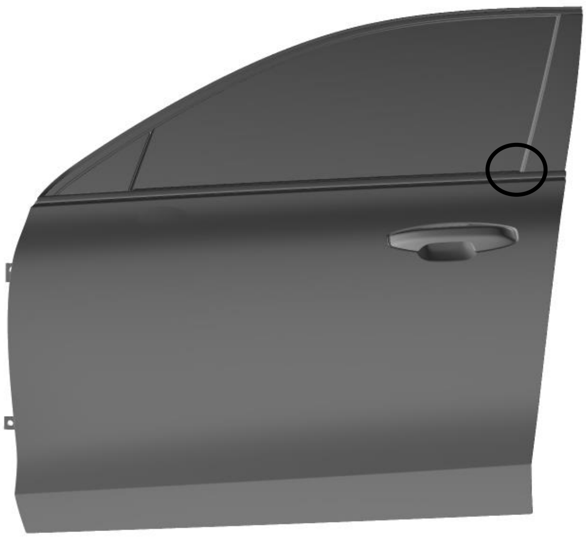


Figure 5.49 An overview of the front door

5.2.1.1.2 Rear door

The setup of the rear door will follow the same process as the one described for the front door in the previous subsection. The biggest difference between the two cases consist of the small differences in geometry. Gaps to the rear door case is created at each side of the main window, the position of the gap is marked by black circles in Figure 5.50. Due to another type of sealings the smaller window at the right prevent water from entering the door in a better way, due to this it is not necessary to study this area. The final geometry of the rear door is shown in Figure 5.50 below.

Figure 5.50 An overview of the rear door

5.2.1.2 Sunroof

In order to reduce the number of particles within the simulation each pipe is divided into a separate subcase. The initial geometry is shown Figure 5.51, which is later on divided.

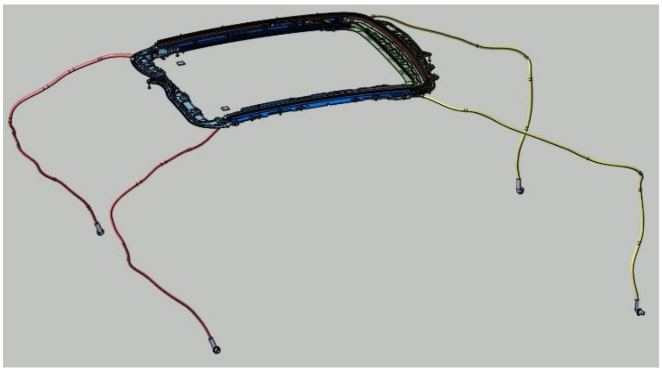


Figure 5.51 Initial geometry for the pipes

Each case needs a box containing the test volume. The box will be mounted on the inlet of the pipe, this process needs to be done in an external CAD program. The process is the same as within SCCM, which is explained in subsection 5.1.1.3. The only difference is that the outlet does not need any surface at the outlet, which instead need an open outlet. The final geometry is shown in Figure 5.52

Figure 5.52 An overview of the drainage system

5.2.1.3 Trunk

Regarding the trunk case it is not necessary to modify the geometry, due to this case involving external water flow. All that is required is to split the geometry in different parts and save it as STL-files. This will make it possible to set different settings to different parts. It is important to split the geometry inside an external CAD program, because there is no option to modify the geometry inside PreonLab. All STL-files together is shown in Figure 5.53.

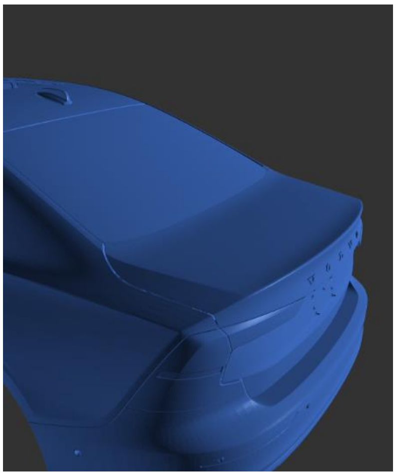


Figure 5.53 An overview of the trunk

6 Results and discussion

This chapter is showing the result from each simulation that has been performed within SCCM and PreonLab. It is also showing the importance of what settings that are needed as well as the value for the setting.

6.1 STAR-CCM+

It is important for the reader to understand that the main objective of this thesis was to create simulations and create templates with initial settings to get a reasonable simulation for each area of the car. As the Water Tightness Department at Volvo Cars has very little previous work within CFD, this project was performed to enable them to start working in SCCM (or PreonLab where it is deemed superior). As such, the resulting product of this project consist of the templates which contain the settings used to create the simulations. The resulting simulation animations have been judged mostly to make sure that the decided settings create a realistic scenario and further work will have to be made regarding the optimization of these settings and models. In this chapter, the settings used will be explained while also making sure that the reader understands why and how these settings were decided.

6.1.1 **Simulation settings**

Four different simulations have been performed in SCCM, which is the front and the rear door, pipe drainage and the trunk. What settings that have been used and a smaller explaining of each step within each case can be studied in Appendix C.

6.1.1.1 Side door

The main areas of focus when setting up the side doors are the gap and the latch area. In order to make these areas as close as possible to the reality, CAD preparations and setups inside SCCM are needed. When the geometry setup is done in a CAD program, it is important to process the settings inside SCCM carefully in order to create a simulation that represents the reality. Subsections 6.1.1.1.1 and 6.1.1.1.2 explain the difficulties and show the result of how the water is streaming.

6.1.1.1.1 Front door

The most time-consuming part of the setup process inside SCCM was finding a mesh that was working. The most important parts to go through to get a mesh working is the surface wrapper, mesh size, prism layer thickness, size of prism layers and the type of mesh. All these settings need to be considered, but it is hard to make an initial guess regarding what values should be used.

When creating the surface wrapper, the goal is to find a good surface mesh for all important surfaces while at the same time avoiding intersections in the geometry. In the end a global cell size of 6 mm with adjustment to the inlet, sealing, window, channels with a size of 16 – 50 % of the global mesh size was chosen.

By getting support from SIEMENS and comparing different types of mesh structures, the trimmed mesh was ultimately decided to be the most advantageous mesher. This is because it generates a fast mesh at the same time as the simulation becomes faster.

To find the optimal size of the cells and the prism layer, a few different options were tried. The result shows that the size could not be homogeneous within the whole geometry, due to water disappearing when it reaches the parts of the mesh that are too coarse. In the areas where the water is expected to be, the size of the cells is set to 3 - 7.5 % of the global cell size, which is set to 9 mm. The reason for having a coarse global size is that the simulation will otherwise become too heavy to simulate. What is important to take into account when having different cell sizes is that the transition is smooth between the coarse and the fine parts, otherwise the water will tend to disappear during the transition. The result of the mesh distribution is shown in Figure 6.1 and Figure 6.2 which are taken from a plane section.

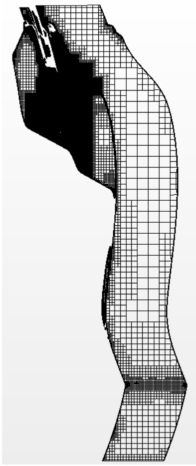


Figure 6.1 Overview of front door mesh

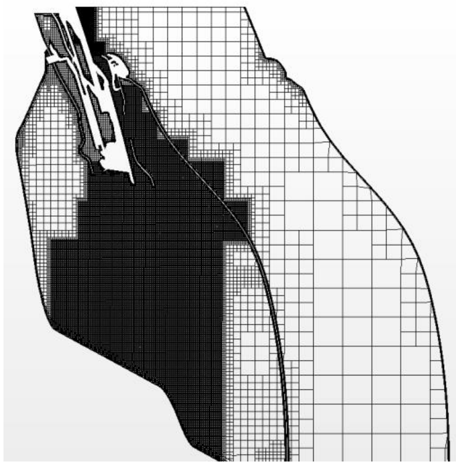


Figure 6.2 Zoomed view of the front door mesh

The prism layer was increased in areas that are expected to contain water. In some areas where the water is either of little interest or the wall treatment is not expected to have a big impact, the prism layer uses fewer layers than the areas where the flow is expected to be highly dependent on the nearby walls and their properties. The thickness of the prism layer is set to 2.5 mm and the number of layers is set to 8 in comparison to the global that has values of 1 mm and 5 layers. Creating these kind of prism layers will prevent the water from disappearing.

6.1.1.1.2 Rear door

The approach and what parts that need to be considered for the rear door is similar to the front door, with small differences because of the somewhat different geometry.

The first test simulation of the rear door consisted of the exact same settings and values as for the front door. After studying the first result, some small adjustments were found necessary. The global surface wrapper was changed from 6 to 10 mm in order to reduce the number of cells, since the rear door has a bigger volume than the front door. By using a simulation that is working as a template, the purpose of a template could be tested. The result shows that a template is very useful to guide the user with necessary information and thanks to the templates a result was found much faster than for the front door.

6.1.1.2 Trunk

Even for the trunk case the most time-consuming part is the meshing. The main task for the trunk can be divided into two parts that need to be solved to get a proper result of the simulation. The first part consists of the phenomenon where water is leaving a surface which occurs in two different areas of the geometry. These areas are where the water is falling from the trunk surface to the window and the gap between the window and the luggage area. In this area it is required to have a fine mesh which is created by boxes, where the cell size is set to 0.8 mm. This will prevent the water from disappearing. These boxes are shown in Figure 6.3.

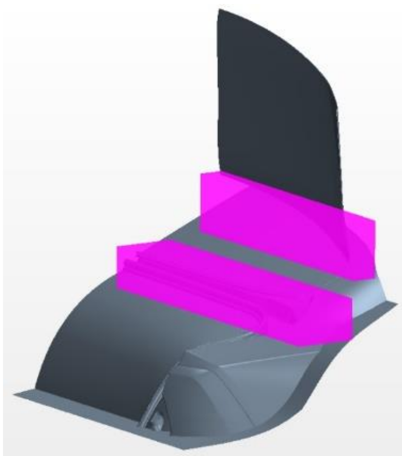


Figure 6.3 Volume boxes defining refined mesh area

The second part that needs to be solved is the area close to the surface of the trunk and the window together with the sealing at the end of the window in the gap area. This task is solved by increasing the thickness and the number of prism layers at the trunk and the window. The thickness is set to 4 mm and with 8 layers of prism layers, compared to the global values which have 3 layers with a thickness of 2.6 mm. The sealing is solved in a different way. It is done by reducing the thickness to 0.8 mm at the top of the sealing and disabling the prism layers underneath. This will make the water drop of the edge of the sealing and not follow the surface of the sealing all time. An overview of the mesh distribution is shown in Figure 6.4.

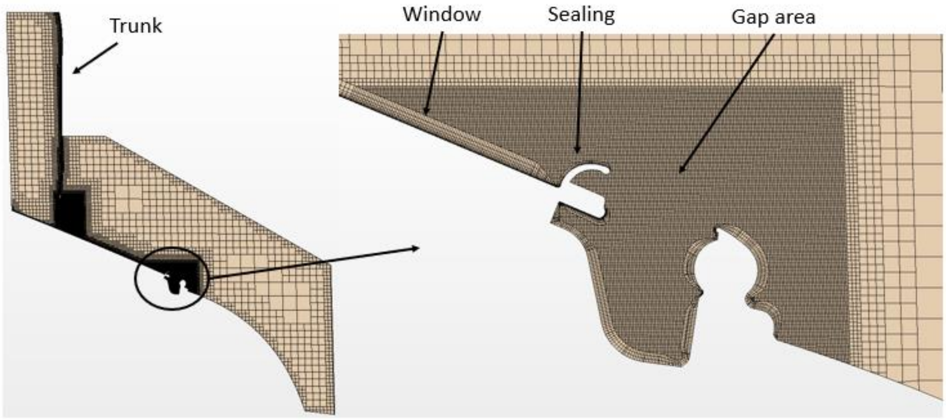


Figure 6.4 Over view of the distribution of the geometry

6.1.1.3 Sunroof

The most time-consuming part of the pipes is the CAD preparation. This is because surfaces are needed to be trimmed and added to the initial geometry.

One aspect that stood out in comparison to the other simulations inside SCCM is that no inlets are used during simulation. The box is acting as an initial volume instead of injecting water during simulation. The outlets in the pipe simulations are also different compared to the other cases. In this case the outlets are set to the pressure function of the flat wave model. With this function the pressure at the outlet will be dependent on the depth of the fluid inside the pipe. In other words, the pressure will be reduced as the simulation is running and the water is exiting the pipe. In Figure 6.5 shows that the outlets are located at the top and the bottom of the geometry.

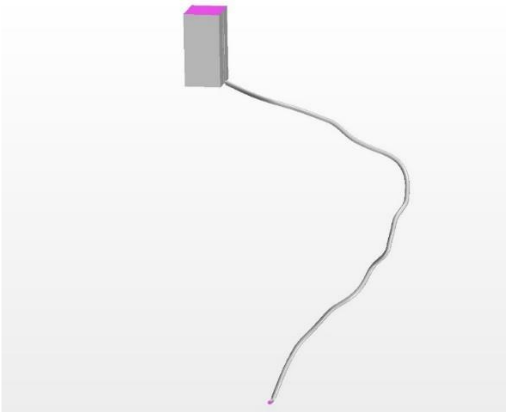


Figure 6.5 Pressure of flat wave

6.1.2 **Post-processing**

Post-processing is showing the result of the simulation within different types of scenes and views. The blue scale in the pictures shows the volume fraction of water range which is set to 0.5 – 1. Due to the scale, water is marked blue in the simulations.

6.1.2.1 Side door

During post-processing at least one of the outer metal sheets are hidden in order to be able to study how the water is streaming inside the door.

6.1.2.1.1 Front door

Figure 6.6 is showing how the water is streaming illustrated from a back view. The result shows that the water is manly streaming along with a vertical channel made to fix the window and to lead water inside the door. This channel makes most of the water streaming past the latch system. Worth noting is that a small volume of water is collected over the latch system, which indicate that it is possible that water could be able to reach the latch system. This is illustrated by an arrow in Figure 6.6

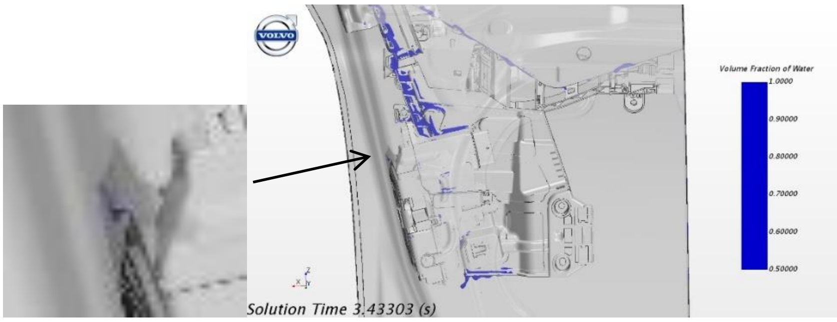


Figure 6.6 Water flow from back view

Figure 6.7 shows that the water is travelling along the rail inside the door that is designed to keep the water from hitting sensitive parts.

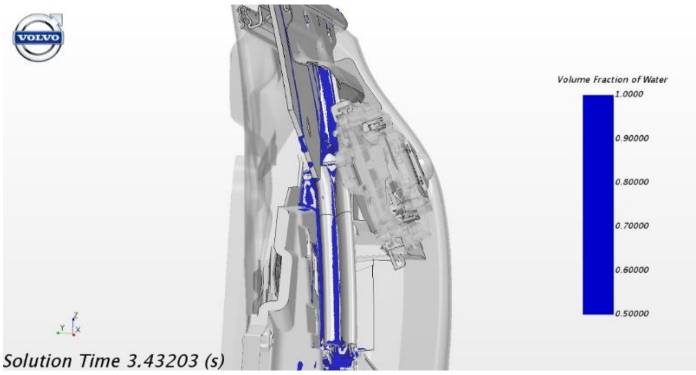


Figure 6.7 Water streaming in channel

Figure 6.8 shows a front view but is also showing that the main stream of the water flows past the latch system. In this view no signs of water in the area of the latch system can be seen.

Figure 6.8 Water flow from a front view

Figure 6.9 is showing an overview of the front door with a view from the front. The result shows that water will be entering the door not only from the main gaps at the sides of the rubber rip, but from the window as well. With this view it is possible to study the water flow on the window where the water will also enter the door. What can be seen is that the main part of water that is entering via the window is collecting together at the bottom of the window inside the door. This results in a larger water flow from the lower part of the glass window.

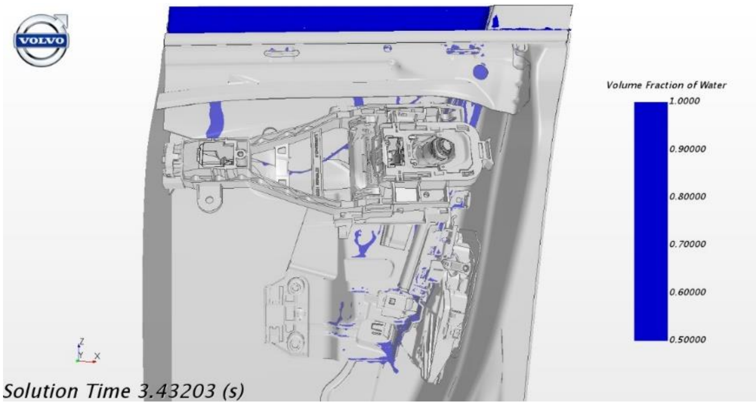


Figure 6.9 Over view of the front door in front perspective

Figure 6.10 shows that the water volume that has entered the inside of the door is mainly streaming in a vertical direction through the center of the door and no water is streaming along the outer metal sheet on the inside. If the channels are not carefully designed, water might be flowing in uncontrolled ways once it enters the inside of the door. As the picture shows, this is not the case.

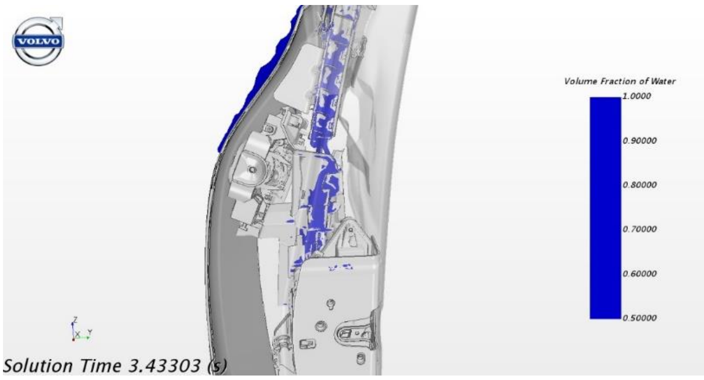


Figure 6.10 Showing the water streaming form side perspective

Figure 6.11 Showing window channel

The physical test has two main flows that could be distinguished around the latch system. The first is flowing upon the latch system that comes from the window channel (shown in Figure 6.11) and the second one is streaming along the window edge towards the bottom of the window where is starts to drop. The result for the virtual simulation, which is shown in Figure 6.9, is close to reality because it has the same behavior as the physical one. From that it can be determined that they are very similar with the settings that are used.

6.1.2.1.2 Rear door

The result of the water flow can be studied in Figure 6.12, which shows how the water is streaming with a view from the back. No water collecting is noted above the latch system and no water is streaming continuously over the latch system.

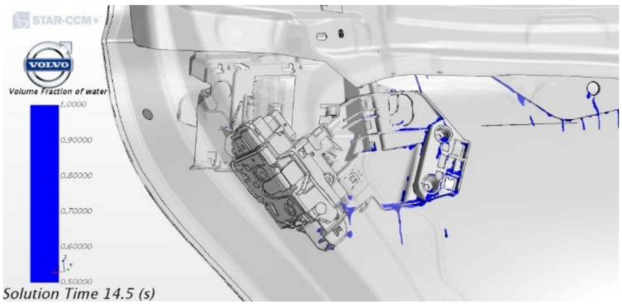


Figure 6.12 Latch system rear door, back view

When studying the same area as Figure 6.13 is showing but with a view from the front and it can once again be seen that there is no water around the latch area. This confirms that no bigger volumes of water will flow above the latch.

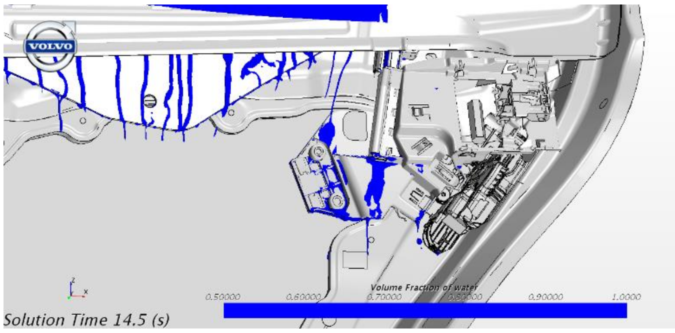


Figure 6.13 Latch system rear door, front view

To study the water flow inside the complete door Figure 6.14 and Figure 6.15 are used. Figure 6.14 shows the water flow in a view from the back side. The result from this view is that no water from the front region is streaming towards the rear region, which states that no water from the front have any impact on the rear region.

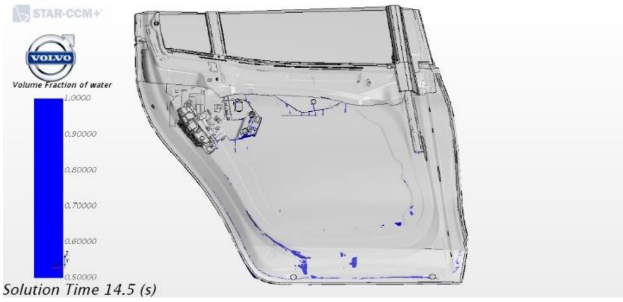


Figure 6.14 Overview rear door, back view

Figure 6.15 is showing an overall view of the water flow in a perspective from the front. From the picture it can been seen that the channels both in the front and the rear impact the flow of the water to a large extent because barely any water can be found outside these channels. This shows the importance of the channel in order to form the water path inside the door.

Figure 6.15 Overview rear door, front view

The virtual simulation result is close to the reality due to the flow streaming similar to the physical test. The water flow for the physical test inside the rear door could be divided into three flows: one upon the latch system, one at the front of the rear door and one on the window. Those three flows are also illustrated for the virtual test shown in Figure 6.15. Due to the similar flows, the settings that are used could be confirmed to be an alternative that generates a virtual simulation which is in line with reality.

6.1.2.2 Trunk

Figure 6.16 is showing the water flowing along with the surface of the trunk. Worth noting is that the water is streaming down quite vertical which is shown on the side where the trunk geometry is a bit bulging and the outermost part is not covered by water.

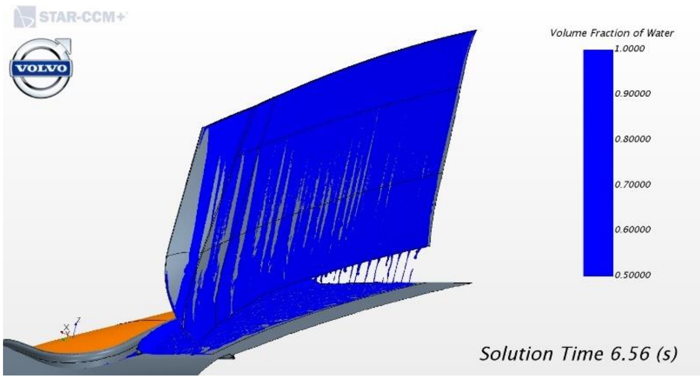


Figure 6.16 Water flow along the surface of the trunk

How the water is acting in the area of the gap with a fully developed water flow can be studied in Figure 6.17. The result shows that when water leaves the rubber sealing it drops directly towards the water channel.

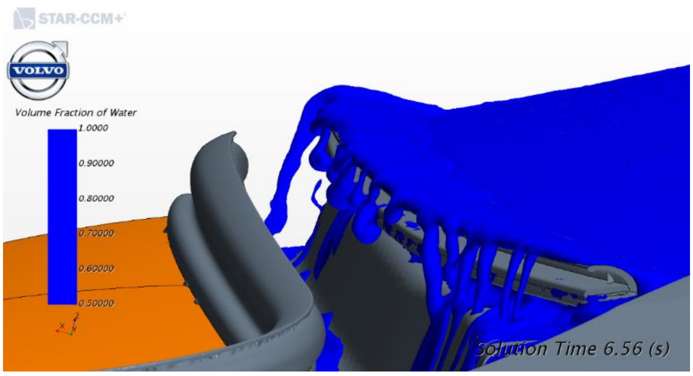


Figure 6.17 Water drop after leaving rubber sealing

In order to analyze the gap even more, Figure 6.18 can be studied. This is a plane section along the length of the car located close the center of the trunk. The mesh cells colored red means that the cell contains a high volume fraction of water while blue means that there is no water. This means that colors separated from blue indicates that there is water in the cell. In the figure one can study a surface tension that occurs when the water is collection before the gap. For the water to flow over, a slightly higher water level than the highest point of the rubber sealing is required.

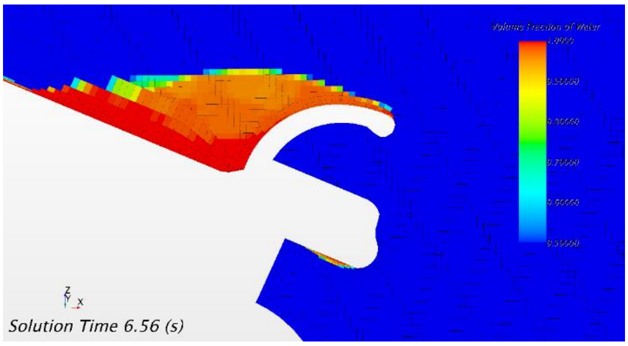


Figure 6.18 Water flow at the gap

Figure 6.19 shows how the water is behaving when water is falling from the trunk surface to the window. It is showing that the water is transformed into droplets once it leaves the edge of the trunk but is then collected into a continuous flow once it hits the window.

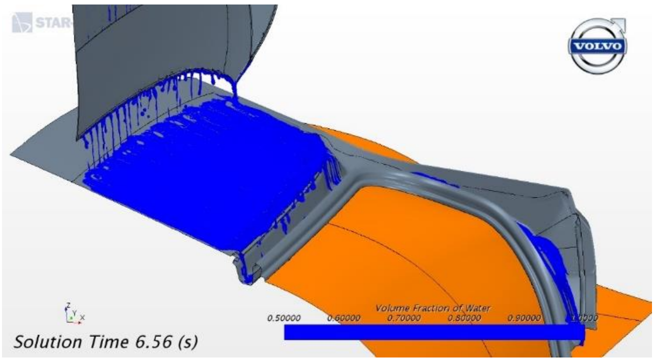


Figure 6.19 Water falling from trunk surface to window

The result of virtual simulation shows in Figure 6.17 that the water is falling into the water rail after leaving the rubber sealing. It also shows that there is no main flow that is falling into the luggage. This kind of water stream behave is in line with the physical test since there is no water that is flowing into the luggage area. Having a virtual result that act as the physical determine that the settings that are used generate a simulation that is close to the reality.

6.1.2.3 Sunroof

Two different pipe geometries are simulated in order to see what impact the geometry has. Both pipe configurations have the same purpose but for each respective model. Red color is indicating a high volume fraction of water while blue is indicating that there is no water.

6.1.2.3.1 Front left model x

The result of the simulation in Figure 6.20 and Figure 6.21 show that the initial water has flown through the pipe after 25 seconds. The current requirement from Volvo Cars for the pipes is measured as a flow rate. As such, one way to simulate this is to create a volume with the same dimensions as the volume required to flow per minute, and then check whether the box is emptied by the one-minute mark.

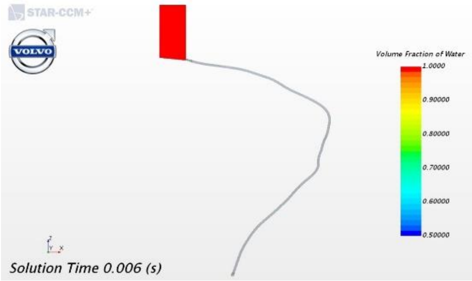


Figure 6.20 Initial phase of the simulation

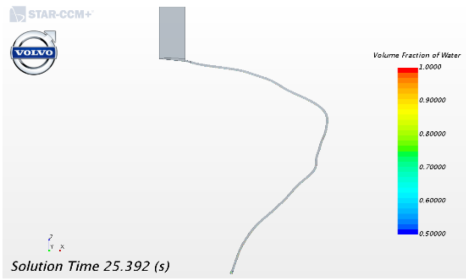


Figure 6.21 End phase of the simulation

6.1.2.3.2 Front left model y

Figure 6.22 and Figure 6.23 are showing that it takes 22 seconds for all water to flow through the tube. The amount of time that was measured for the physical testing was approximately 35 seconds. This shows that the geometry has some impact even though the results are quite close to each other. Having a result close to each other is reasonable as the geometries are quite similar.

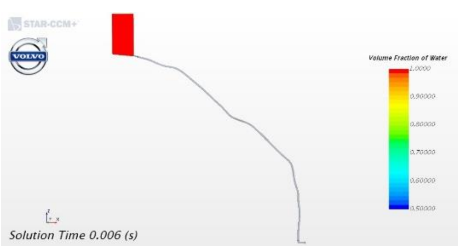


Figure 6.22 Initial phase of the simulation

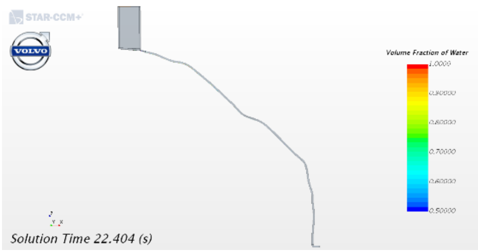


Figure 6.23 End phase of the simulation

According to the virtual simulations, a good understanding of the behavior could be found, as the result for the virtual simulation and the physical test being close to each other.

6.2 PreonLab

The philosophy regarding the results of the PreonLab simulations is somewhat different from the one in the SCCM section. PreonLab was introduced quite late into the project as an alternative to simulate water flows. The templates in SCCM were created to simplify the time-consuming parts of the process, like meshing. For PreonLab the focus will be to perform the simulations and compare them to the simulations performed in SCCM to judge whether PreonLab could prove to be advantageous for certain simulations. Still, to document the progress in PreonLab, templates were made for this software as well. These are not as detailed as the SCCM versions, however, and were made mostly to provide some continuity in the project.

6.2.1 Simulation settings

6.2.1.1 Side door

For a particle to pass through a hole, it is required that the size of the particle is half the size. This requires the particles to be very small to enter the inside of the door, since the main gap is already small.

6.2.1.1.1 Front door

No steps during the setup was significantly more time-consuming than any other, which is one of the advantages of using PreonLab.

The water injection is set to a continuously inject a water flow from a square source, located at the main gap at the back of the window. The reason for having local sources is that the number of particles will be less compared to using a bigger water inlet. The end-results are not affected by this, as there is a limited volume that can flow through the main gap per time unit.

During setup two main settings are necessary to control. The first is to know the size of the smallest gap where water should enter and then set the particles to half of this size. The second is to create an outer boundary that defines where the water is allowed to be. This is done by creating boxes that contain the important particles. The result of the setup is shown in Figure 6.24.

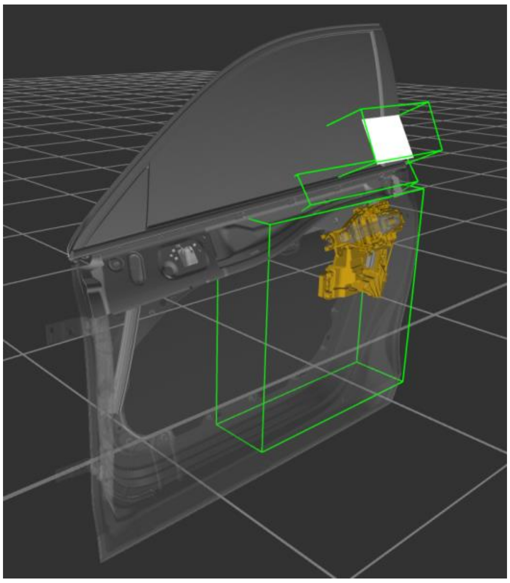


Figure 6.24 Illustration of the settings for the front door

6.2.1.1.2 Rear door

Due to the geometry not cropping, the setup process for the front and rear door will be identical with the geometry as the biggest difference. This difference means that some modifications need to be done to the inlets to make sure that the flow is pointed at the intended area. The angle of the bar separating the windows create a different angle from the one in the front door case. These are very small differences though and do not require high precision. The result of the setup is shown in Figure 6.25.

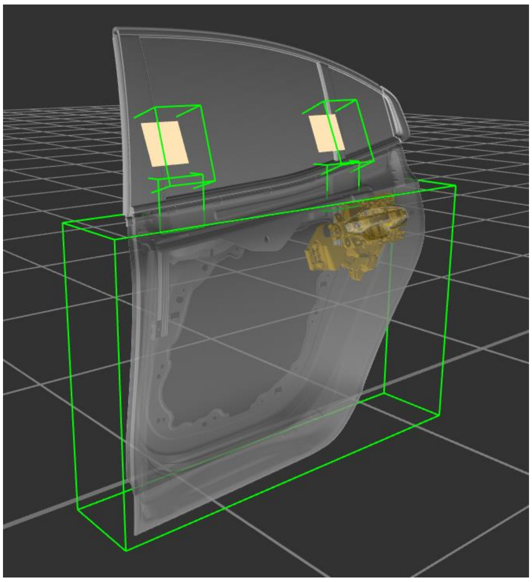


Figure 6.25 Illustration of the settings for the rear door

6.2.1.2 Trunk

When doing an exterior water flow simulation, the size of the particle is not as important compared to cases where water is streaming through a gap. It is still important to create box domains that contain particles located in areas of interest and delete the particles of little interest. Due to this requirement, box domains need to be created slim to the geometry. This is shown in Figure 6.26.

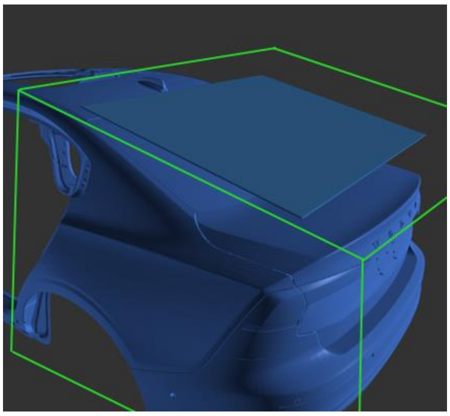


Figure 6.26 Illustration of the settings for the trunk

During simulation, a rotation of the tailgate will cause the trunk to stand in its maximum position. The rotation is slightly modified in order to visualize the reality as much as possible, by adding a keyframe which creates a faster acceleration in the beginning of the rotation. This is done by using a sinus curve. The setup of this type of motion is not much more time-consuming compared to a steady state simulation without motion. The drawback is that the simulation will take a longer time.

6.2.1.3 Sunroof

This case is an interior water flow simulation which requires the size of the particles to be at least half the size of the pipe diameter. The initial volume inside the box is the same as the requirement says that the pipe should handle during a specified time.

Due to an interior water flow simulation the orientation of the outer boundaries is not that important, the only important thing is to remove all particles which leave the outlet at the end of the pipe. This is because they are not giving any information after that stage. A result of the setup is shown in Figure 6.27

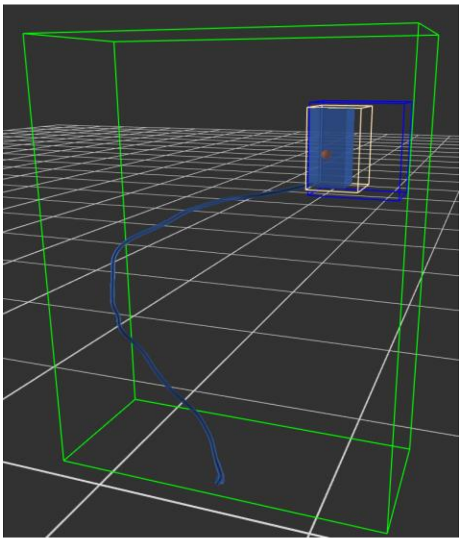


Figure 6.27 Over view of the setup pipe

6.2.2 **Post-processing**

6.2.2.1 Side door

During the simulations of the doors in PreonLab one big obstacle was noted. The gaps on the sides of the window are quite small. This means that consequently the particles will have to be quite small. As the simulation speed is highly dependent on the number of particles, one needs to be very careful not to make the particles too small. The particles should only be small enough to enter the gaps. Even after careful consideration of the particle size, the inlets need to be modified so that a majority of the water applied in the simulation is going into the gaps. Little to no water should be flowing outside of the door and in places of no interest. Another problem with the small particle size is that the particles will become quite hard to distinguish in the simulation. Because of this, wetting sensors will have to be added.

6.2.2.1.1 Front door

The PreonLab provides the same ability in as SCCM in being able to make parts transparent. The outer shell, like in SCCM, is chosen to be transparent to some degree. It should, however, not be completely removed as the shell will have an impact on the flow. Since the latch is the main point of interest in the door, one needs to make sure that it is shown with a color that is easy to distinguish from the rest of the model. In this case, the color was set to a yellow tint as shown in Figure 6.28. It should be noted that most of the flow will be going on the opposite side of this view, but it's still important to capture the backside as there might be flow coming here as well.

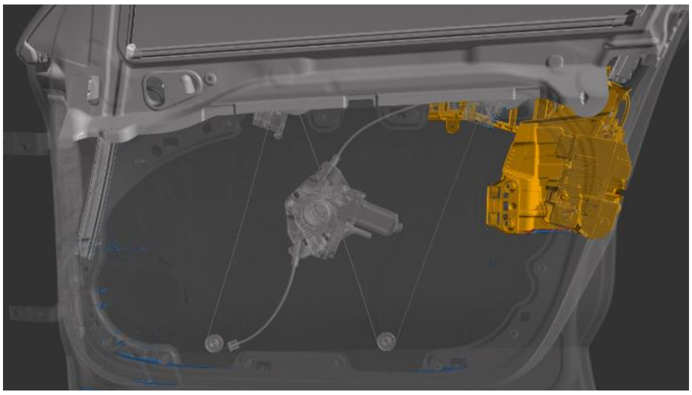


Figure 6.28 Water flow of the front door, back view

As mentioned previously, a wetting sensor is applied to the latch. Since the aim of this project is to evaluate where the particles are in contact with the latch part, this severely improves the visibility. This is shown in Figure 6.29 where the red dots represent the areas in contact with the water particles. A zoomed in view of the latch like this is also highly recommended as the dots can be hard to distinguish from a more distant view.

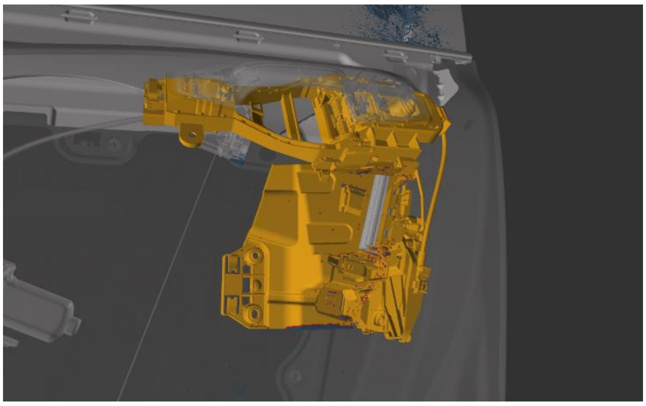


Figure 6.29 Water flow on the latch of the front door, back view

The virtual simulation result in Figure 6.29 is close to the reality as two main streams could be distinguished both for the virtual and the physical test. The first flow is the water streaming upon the latch system and the second one where water is dropping from the bottom part of the window. When the behavior is the same for both the virtual and the physical test, it can be confirmed that the virtual test is close to the reality with the settings that are used.

6.2.2.1.2 Rear door

Figure 6.30 shows the zoomed in view of the latch in the rear door. Compared to the latch in the front door, it should be noted that this latch has a slightly different structure which will have some impact on the water flow.

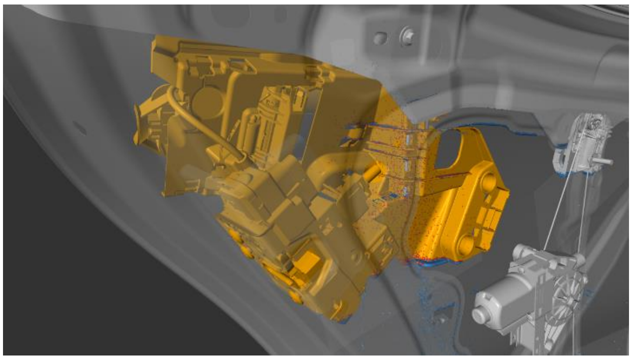


Figure 6.30 Water flow on the latch of the rear door, back view

Figure 6.31, the back view of the rear door, contains some interesting information. Do recall that PreonLab does not utilize a mesh. This might lead to some lost information in the areas with a fine mesh as softwares like SCCM can make very detailed calculations in areas with a fine mesh. It also means, however, that areas where a coarse mesh would be used in a SCCM simulation are more accurate in PreonLab where a mesh structure does not impact the result. The water at the bottom of the door is not of interest in this project and because of this, a coarse mesh was chosen for this area in SCCM. This resulted in no water being shown in the captured simulation inside SCCM. In this case, however, it can clearly be seen that water is stagnating at the bottom. Since this might be of interest to other requirements set by Volvo, it is clearly an advantage that a general sense of the amount of water stagnating water can be seen in the bottom.

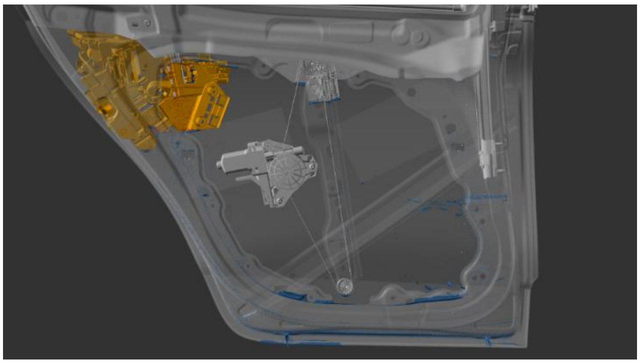


Figure 6.31 Water flow of the rear door, back view

Figure 6.32 provides the same view that was used in the front door. This is a good overview of the front part where the flow will be going and how it is divided between the gaps at the sides of the window and the window itself. As seen on the latch, it is filled with red dots, but it is hard to distinguish the flow from this distance.

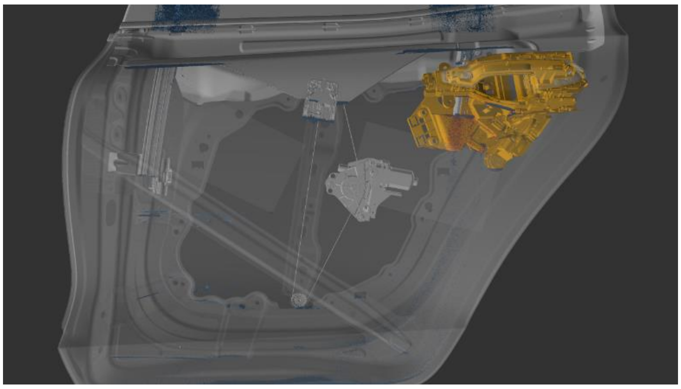


Figure 6.32 Water flow of the rear door, front view

To further improve the visibility, the view in Figure 6.33 is used. At the bottom left of the latch it can be noted that the red dots are now able to give a good idea of the general flow and where most of the particles are going. The rails that are made to direct the flow away from the latch area are clearly working as the flow is going the intended way.

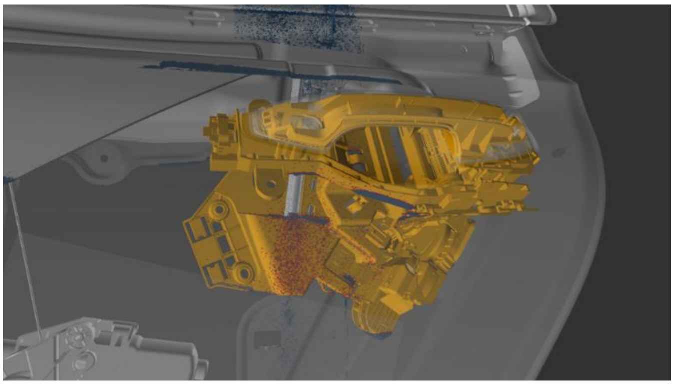


Figure 6.33 Water flow on the latch of the rear door, front view

The virtual simulation is similar to the reality because the virtual flow has two main flows inside the door around the latch system which is reminiscent of reality. One water flow is streaming upon the latch system, which comes along the window channel (gray part in the middle of the latch system, shown in Figure 6.33). The other main flow is the one that streams along the window edge down to the window bottom, where it starts to drop. The same behavior could be distinguished both for the virtual and the physical test and this confirms that the virtual test is close to reality with the settings used.

6.2.2.2 Trunk

The view shown in Figure 6.34 is the angle where one would be standing when the test is performed. It gives a good view, not only of the luggage area but also the bottom of the window where the water will be hitting the trunk. The disadvantage of the view is that some parts of the bottom of the luggage area is hidden.

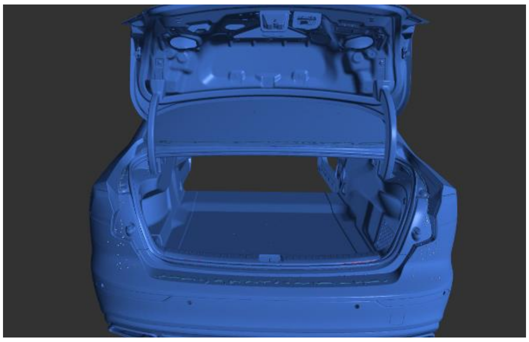


Figure 6.34 Open trunk, back view

The only way to capture the bottom corners of the trunk area is by slightly tilting the view to the left or to the right like the one in Figure 6.35. While this is in other regards not a very useful view, some of the water might be entering through the left or right edges of the trunk. It is therefore important to check whether this happens by adding this view.

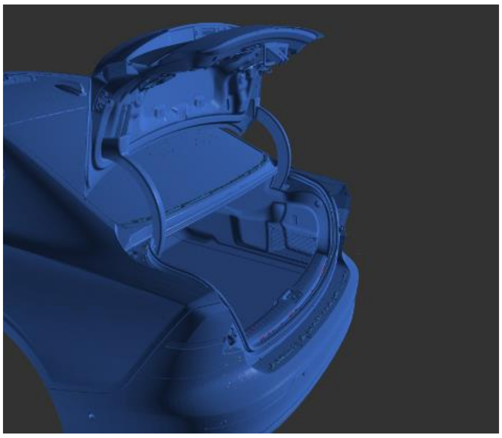


Figure 6.35 Open trunk, side view

Figure 6.36 is a view that was used when filming the virtual testing. This provides very useful information regarding the design of the rubber lip and whether the water ends up in the rail or in the luggage area. As seen in this zoomed in picture, all the water is currently entering the rail according to the design.

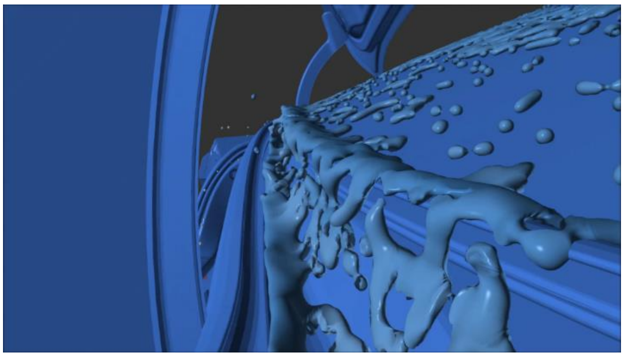


Figure 6.36 Overview of the gap, virtual simulation

The result from the virtual simulation shown in

Figure 6.36 could then be correlated to the physical test. By comparing the physical test and the result from the virtual simulation shown in

Figure 6.36, a similarity can be seen. The similarity is that the water. after leaving the rubber sealing. is entering the rail area and no bigger water stream is entering the luggage area. Due to this verification, the virtual simulation could be confirmed to be close to reality.

Figure 6.37 provides an overhead view of the bottom area of the luggage area. This is another way to judge the performance of the rubber lip and rail. Some particles can be seen entering and touching the bottom of the luggage area. It was noted during the simulation that some of these particles were entering because they were stuck in the area along the edges of the trunk before the trunk was opened. No general water flow could be seen entering the luggage area and it is inevitable that some droplets find their way in.

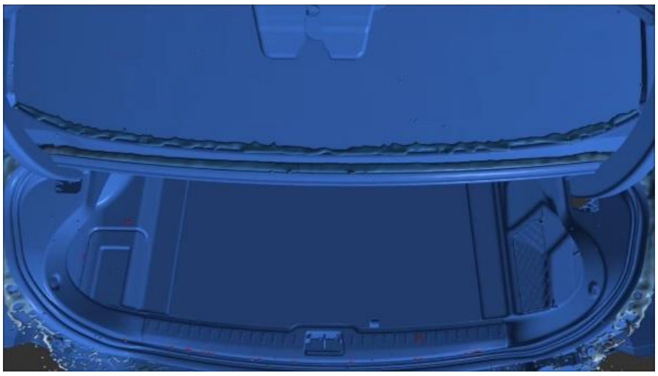


Figure 6.37 Top view of the luggage area

Figure 6.38 once again shows the side view of the trunk, but this time with the simulation running. It can be seen that there are no particles in the corners while some particles are found in the middle just like Figure 6.37.

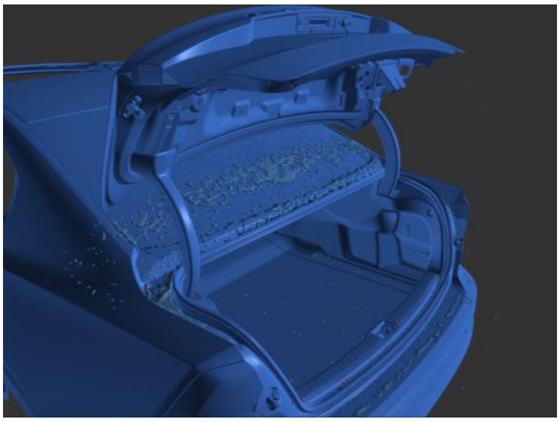


Figure 6.38 Open trunk with water particles, side view

6.2.2.3 Sunroof

As there were some complications in modifying the CAD model around the inlet to the pipe, the view shown in Figure 6.39 was added to make sure that the water could easily enter the pipe. This proved to be no problem in the simulation.

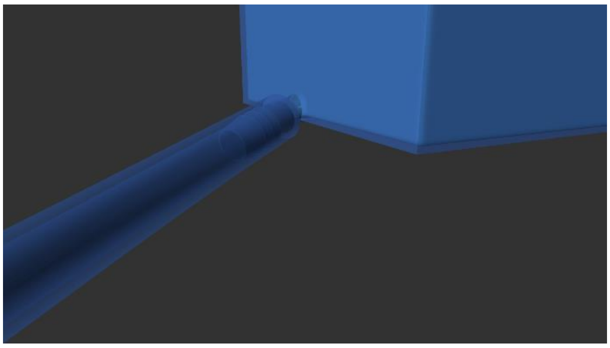


Figure 6.39 Connection between box and pipe

Figure 6.40 is the most important view for the drainage system. As there are limited options in measuring the flow, the capacity of the drainage system will be judged based on the visual representation. The view captured is what the box will look like at the time when the simulation is started.

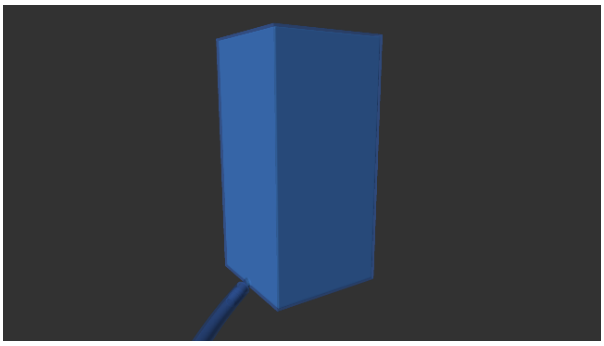


Figure 6.40 Overview of the box

The following pictures will be showing the previous view at different simulation times. Figure 6.41 shows the simulation after 15 seconds. As the requirement for the pipes is to empty this volume by 60 seconds it seems like the water level is exiting at a pace that is enough to fulfill the requirement. Figure 6.42 shows the water level at 30 seconds. Now, almost 60% of the water has left the box.

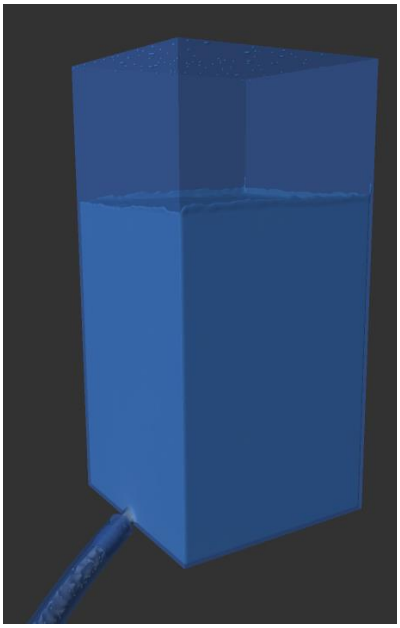


Figure 6.41 15 s

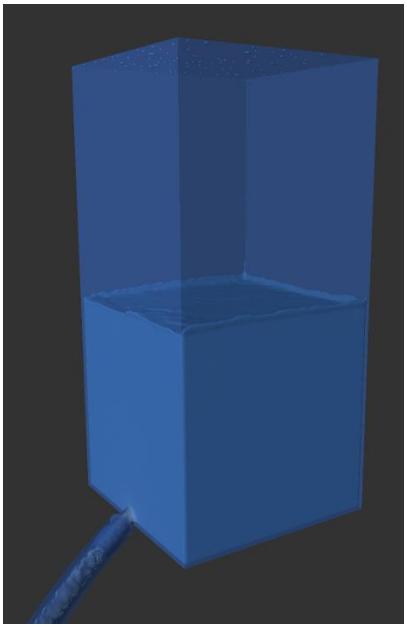


Figure 6.42 30 s

Figure 6.43 shows the box at 45 seconds. It can be noted that the flow rate seems to have somewhat slowed down. Finally, Figure 6.44 shows the water level at 60 seconds. This simulation suggests that the drainage capacity of the pipes is not enough. The SCCM simulation showed a completely different results which was more in line with the physical testing. This means that there is probably some physics that are left out in PreonLab that is providing the inaccuracy. This most likely has to do with the pressure inside the box as well as the pipe. It was noted in the physical testing that the pipe was acting in a sucking manner where the flow seemed to be dragged into the pipe entrance, especially as the water was almost emptied. The simulations suggest that PreonLab cannot be used to evaluate the drainage capacity of the pipes.

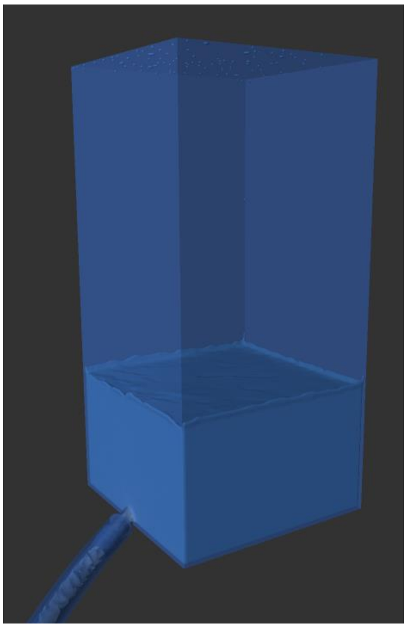


Figure 6.43 45 s

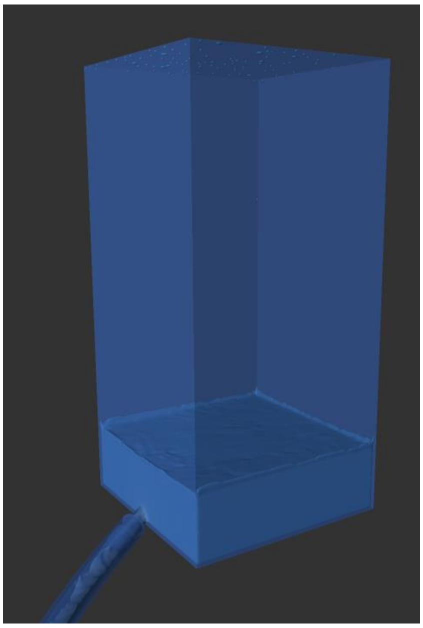


Figure 6.44 60 s

The virtual simulation of the pipe in PreonLab shows that it does not match the reality. This is because in the reality, it took about 35 seconds to handle the specific volume, but the simulation shows that approximately one fourth is still inside the box at 45 seconds and even some volume still remains after 60 second. The conclusion is that there are still some room for improvement before having a simulation close to reality.

6.3 Template

All templates are written to employees with a good insight in SCCM as well as within PreonLab. With the templates, users will be guided with important information and values to setup for each case. With this information will the user faster find a simulation close to the reality, and the simulation will become more credible since those values has already been correlated to reality before in a similar case with another geometry. The templates have not been tested by the employees at the department but they have been evaluated along the process by the thesis students. All templates can be studied in Appendix C regarding SCCM and Appendix D for PreonLab.

6.4 Comparison between SCCM and PreonLab

It is obvious that SCCM and PreonLab have their own advantages and disadvantages. As mentioned previously, SCCM tends to be slower while providing more accurate results. This seems to be especially true for internal flows as shown in the pipe simulations where the SCCM simulation was only seconds away from the real result while PreonLab was more than a minute away. PreonLab has the advantage of being much faster and can finish a simulation within 24 hours, depending on the case, while only the CAD cleanup required for an SCCM simulation might take more than a day. Some cases are also not as affected accuracy-wise as others. In simulations where an air flow or pressure phenomenon are not impactful, PreonLab can create very realistic simulations. If a motion is of relevance to the simulation, PreonLab can also more easily handle these types of scenarios.

For the door simulations, Volvo Cars is recommended to use SCCM. With the small particle size in PreonLab, the particles are very hard to distinguish and there have been times when the small particle size results in some particles clipping through walls. SCCM has settings that creates a simulation that comes very close to the physical testing while being much easier to visualize.

For the trunk simulations, PreonLab provides an excellent option. This is because the opening of the trunk is of major importance to this requirement. As mentioned previously, this can be difficult to setup inside SCCM. To get a good simulation of the trunk inside SCCM, a lot of time had to be spent setting up the mesh and making sure the flow was captured. PreonLab is much faster and a simulation can be started within an hour of work if the user has some previous experience in the software. Compare this to Star-CCM where setting up an appropriate mesh might be a full day of work.

For the pipe simulations, Volvo Cars is recommended to use SCCM. The results in SCCM are quite close to reality and with some modifications they could most likely become very close. SCCM has some physics option that are necessary to correctly predict and simulate the flow inside the pipes. These are absent in PreonLab and the resulting simulations turn out to be very far from the numbers measured in the lab.

As a general conclusion, SCCM performs simulations of internal flows with tight passages much more realistically while simple external flows, especially with motion involved, are much better suited for PreonLab.

7 Conclusion

Four different parts of the car have been divided into separated cases, which have been named front door, rear door, trunk and pipe drainage. Each case has been simulated in both SCCM and PreonLab where the resulting water flow has been studied. The result shows that no water is streaming continually at the latch system neither for the front or the rear door. This was also the case during the physical testing. The simulation of the drainage pipe shows that the pipe is capable of handling the requirements that have been determined by extensive physical testing done by the Water Tightness Department at Volvo Cars. The simulation of the water flow upon the trunk shows that no water is entering the luggage area which is also in line with the outcome of the physical testing.

A pattern that could be noted is that the most time-consuming part during the setup within SCCM is the meshing process but also the cleanup of the CAD. Hopefully the templates will severely reduce the time required to perform a simulation for future car models as some initial values have now been tested and used with some success. PreonLab does not have any process during setup that is significantly bigger than the other, but the majority of the time is instead spent simulating.

The templates can now form a base for the future user to quickly find settings that are optimal for each simulation. With less time needed in order to get a result it is possible to support the designers in a developing process by virtual simulation of the water on the geometry.

If a very fine accuracy is needed, SCCM is recommended for each case. If a good accuracy is sufficient, the choice will be different. In case of the side doors and the drainage pipe which are cases that are having a water flow inside the geometry, the choice is still SCCM. For the trunk, however, there is an exterior water flow, which makes PreonLab more advantageous due to the fact that the size of the particles does not need to be very small.

References

[1] FIFTY2 Technology (2018, May 23). PreonLAB 3.0.0 - User Manual.

[2] Launder, B., & Sharma, B. (1974). *Application of the Energy-Dissipation Model of Turbulence to the Calculation of Flow Near a Spinning Disc.* London: Pergamon Press.

[3] Siemens. (2018, May 23). *Star-CCM+ Documentation.*

[4] Tu, J., Yeoh, G.-H., & Liu, C. (2012). *Computational Fluid Dynamics - A Practical Approach.* Oxford: Butterworth-Heinemann.

Appendix A

7.1.1.1 Three dimensional

Three-dimensional environment makes it possible to simulate a 3D geometry, due to a three-axis coordinate system. For simpler problems it might be relevant to simplify to a 2D model, but in this case the 3D model is required.

7.1.1.2 Eulerian Multiphase

More than one material can be considered with Eulerian multiphase activated. Since this project involves both water and air, it is necessary to use a multiphase model.

7.1.1.3 Convective CFL Time-Step Control

The optional model CFL Time-Step Control provides an alternative way to decide the time-step. Instead of setting it to a set value, the program uses a function to calculate a reasonable time-step at the given time. This is a good option if there are difficulties in finding a good value for the time-step.

7.1.1.4 Exact wall distance

This function computes the wall distance from the center of a cell to the nearest wall with a non-slip condition. This will create a base for different types of models for example relevant turbulence and wall treatments.

7.1.1.5 Gradient

This is an automatically selected model that utilizes gradients, which calculates values for each cell center and center of the cell faces. These values are then used to calculate parameters important to the governing equations.

7.1.1.6 Gravity

This model enables gravity and lets the user decide what gravity acceleration to use within the model. In this project the constant will be the standard gravity acceleration of 9.81 m/s^2.

7.1.1.7 Implicit Unsteady

This model allows the user to decide what solver the program uses to solve the governing equations. The implicit part means that the equations are solved one at a time while the unsteady part is chosen because the simulation is expected to involve fluctuations and therefore the flow will not be constant over time.

7.1.1.8 K-Epsilon turbulence

This is the chosen turbulence model which is used when the program calculates the randomness of turbulence. This model is chosen because it's the most widely used turbulence model within industrial applications.

7.1.1.9 Multiphase equation of state

This is an extension of the Eulerian Multiphase model and means that the simulation will include several phases simultaneously.

7.1.1.10 Multiphase interaction

This is also an extension to the Eulerian Multiphase that handles the interaction between the different phases. This is the biggest reason why the multiphase model becomes more complex.

7.1.1.11 Realizable K-Epsilon two-layer

This is a submodel to the K-Epsilon turbulence model. In their documentation Siemens mentions that the Realizable model generally produces similar or better result than the non-realizable version.

7.1.1.12 Reynolds-averaged Navier-Stokes

This is a model that means that the program uses time-averaged values rather than instantaneous values to solve the governing equations. This severely reduces solving time.

7.1.1.13 Segregated flow

This solver is chosen because according to the Siemens documentation, this model is optimal for flows with a low Mach number. Since the simulations in this report are expected to involve low Mach number flows, this model fits best.

7.1.1.14 Turbulent

With a complex geometry like this a fluctuating flow is expected with some randomness involved. Therefore, a turbulent model is chosen.

7.1.1.15 Two-layer all y+ wall treatment

There are two different wall treatment models depending on the y+ value. Either the user wants the y+ value to be under 5 or over 30. With the two-layer all y+ wall treatment the program combines the two models which makes the simulations less dependent on the mesh.

7.1.1.16 Cell quality remediation

This model makes sure that the mesh does not include any obviously bad cells. It uses a function on the full mesh and the cells that do not pass this function are removed.

7.1.1.17 Volume of fluid (VOF)

This model means that the program will utilize the Volume of Fluid approach which simplifies some equations by assuming, for example, that a phase moves as a unit rather than other phases forming within the primary phase. This includes scenarios like bubbles forming within a fluid. Since this project does not involve boiling or freezing within the water flow, this model can be chosen to reduce solving time.

7.1.1.18 Wave

This model makes the water surface move with the shape of a chosen sinus wave. This gives the water surface a more realistic behavior.

Appendix B

This appendix contains CATIA tools that has been used in the CAD preparation.

Tools that are used in purpose of splitting and deleting surfaces and parts are listed Table 0.1.

Table 0.1 Tools when delete and cutting a geometry

Tool	Purpose
Split by plane	Remove a section of a part by shielding with a plane.
Delete	Delete part parentally

When using split by plane, a plane needs to be created which will generate a line where the part should be cut.

If a part should be deleted permanently, "delete" is the tool that should be used. By highlighting the part and then press delete, the part will be eliminated from the geometry. Highlighting the part could be done in two ways, either select the part in the tree at the left in the environment or select the part directly in the geometry.

Table 0.2 Adding lines and surfaces to the geometry

Tool	Purpose
Join line	Join smaller lines creating one large line
Extrude surface	Create a surface in a direction with reference to a line.
Create line	Create a line between points.
Create surface	Create a surface of three or more points.

Closing the geometry is mainly done using lines and surfaces in different combinations, but the tool listed in Table 0.3 can also be used.

Table 0.3 Additional tools when creating surfaces

Tool	Purpose

Fill hole	Create a surface if all contour edges are connected and create a loop.

Appendix C

Appendix B contains all templates to all cases, related to the software SCCM.

C.1 Template for the front door

Front door simulation using STAR-CCM+

General Information about the process.

<table>
<tr><td>Project</td><td colspan="2">model</td><td>Created</td><td>2018-02-26</td></tr>
<tr><td>Needed part</td><td colspan="2">Parts regarding the structure of the front doors.</td><td>STAR-CCM+ version</td><td>11.06.010</td></tr>
<tr><td>Purpose</td><td colspan="2">Study if water is streaming on the latch system inside the door.</td><td>Assumed performing time</td><td>5 days</td></tr>
<tr><td>Requirement</td><td colspan="2">It is not allowed to have a continuous water flow streaming in the latch system.</td><td>Previous employers performed the process</td><td>Thesis students,
Erik Nilvé and
Daniel Persson</td></tr>
<tr><td>Dept/name</td><td>Side doors</td><td>name</td><td>Required software</td><td>TeamCenter
CATIA
Star-CCM+</td></tr>
</table>

Timeline for the process

CAD (20%)	Surface wrapper (20 %)	Mesh (25 %)	Boundaries (5 %)	Scenes (10 %)	Running (20 %)

1 CAD preparation

Start by exporting the parts shown in Figure 1 and Figure 2 from Team Center. Insert the parts that make up the door structure.

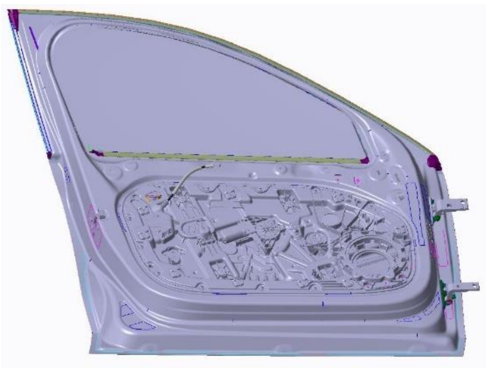


Figure 1 Overview of needed parts, view from inside

Figure 2 Overview of needed parts, view from outside.

Inside CATIA delete parts that are unnecessary to the initial geometry and parts that are not of interest to the simulation. All parts inside the door could be deleted. except for the parts related to the latch system. At the same time as the parts are deleted, cut down irrelevant volumes of the geometry. The result of the trimming can be studied in Figure 3 and Figure 4.

Figure 3 Trimmed geometry, view from inside

Figure 4 trimmed geometry, view from outside

A closer view of the parts that need to remain is shown in Figure 5.

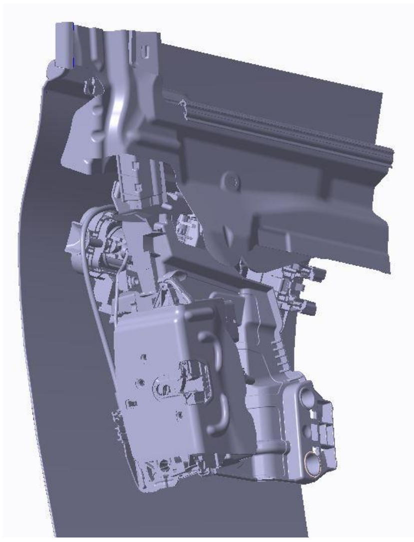


Figure 5 Closer view of parts to remain

Parts that are intersecting with the window are also required to be deleted in order to get a proper water flow at the window. Parts that need to be deleted are highlighted in green in Figure 6, which shows the area between the window and the sealing.

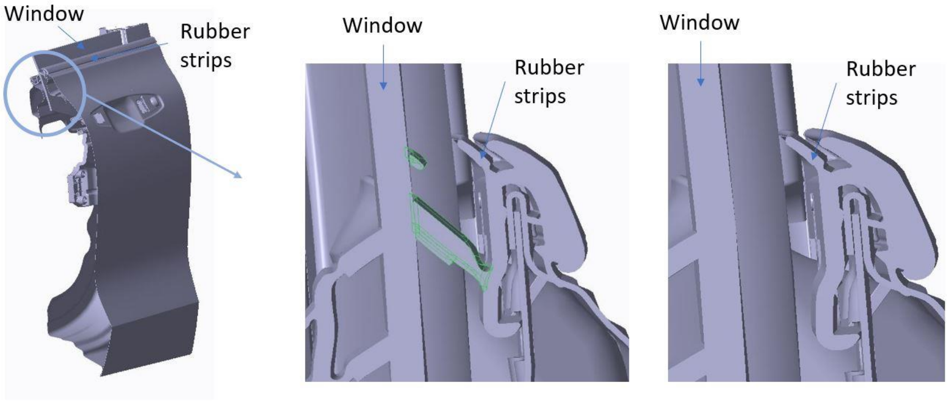


Figure 6 Area in between the window and the sealing

Surfaces acting as water inlets are needed to be created in order to prepare the geometry for simulation. The first surface (blue) is extruded from the edge of the top area of the window. It is extruded horizontally in a direction away from the window and will be acting as a water inlet. The second surface (gray) in between the red and the blue is extruded from the free edge of the first surface, in the same direction as the first. A third surface (red) acting as an air inlet is then extruded from the second surface. At the bottom of the geometry an outlet is extruded on the outside in the same direction as the three others. All these surfaces are shown in Figure 7.

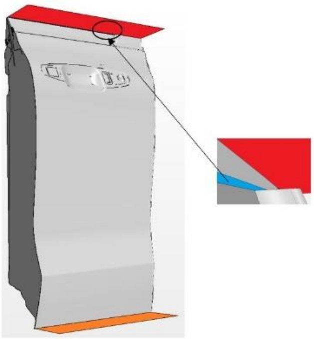
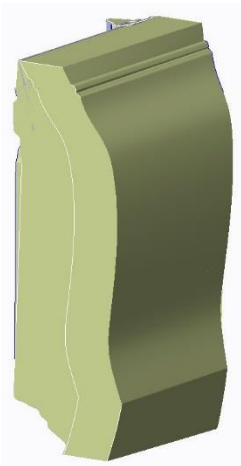
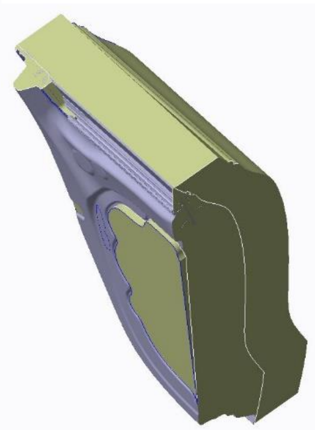
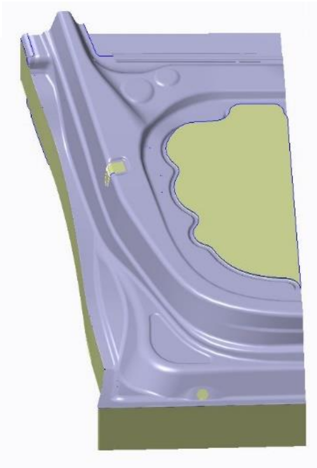


Figure 7 Inlet and outlet on the outer side of the door

To run the simulation, a closed volume is desired. Create contour surfaces to generate a closed volume. Be sure that the boundary conditions do not impact the water flow, but keep the volume as tight as possible to save simulation capacity. The closed volume is shown in three different views shown in Figure 8- Figure 10. All surfaces that are highlighted yellow are surfaces that are created in order to create a closed volume.

-

Figure 8 Closed volume, side view

Figure 9 Closed volume, top view

Figure 10 Closed volume, back view

Save the file as a STEP-file

2 CAD preparation inside SCCM

In order to make a realistic sealing in between the sealing and the window a surface is extruded from the sealing towards the window. The surface stops just before it hits the window which creates a narrow gap in between the window and the sealing. The surface is also modified to leave a gap in between the surface and the vertical bar dedicated for the main gap. This is shown in Figure 11.

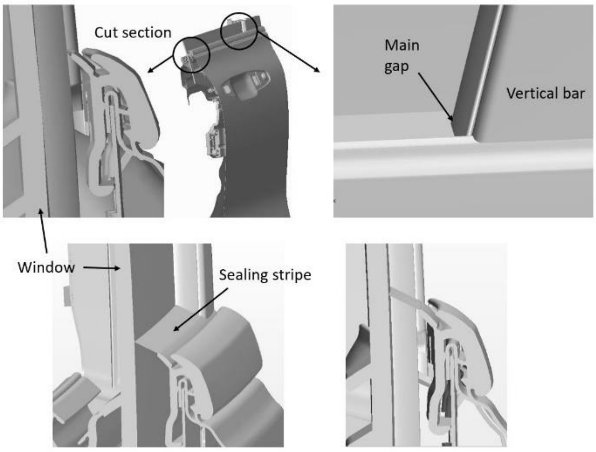


Figure 11 Creating a sealing close to the reality

3 Selecting parts and rename

Import the geometry from CATIA as a STEP file into STAR-CCM+. Select and rename parts to the purpose of the surface. It could be surfaces that will act as inlets, outlets or other surfaces with a special mesh treatment. In Figure 12 the surface highlighted in blue is acting as a water inlet and the red and the gray in between the red and blue is acting as an air inlet. The orange surfaces in Figure 12 and the purple surfaces in Figure 13 are acting as outlets.

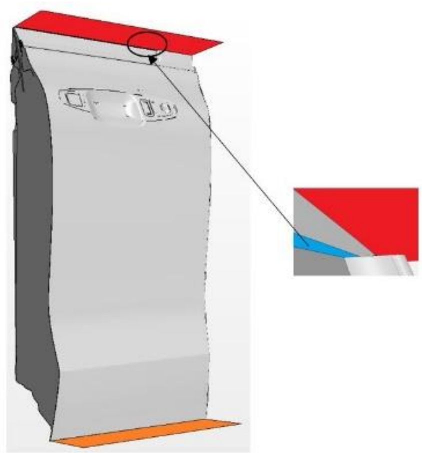


Figure 12 Showing inlets and outlets

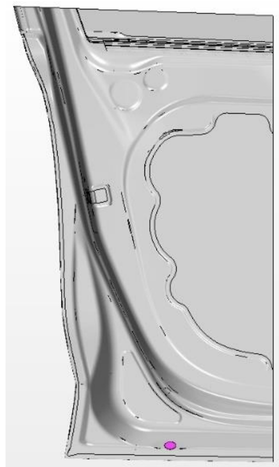
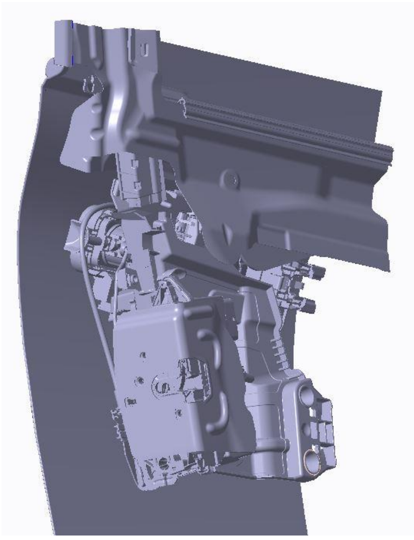


Figure 13 Outlet on the backside

To make further steps easier it is advantageous to rename parts related to the latch system shown in Figure 14. Renaming the window is also advantageous for the same reason, which is shown in Figure 15.



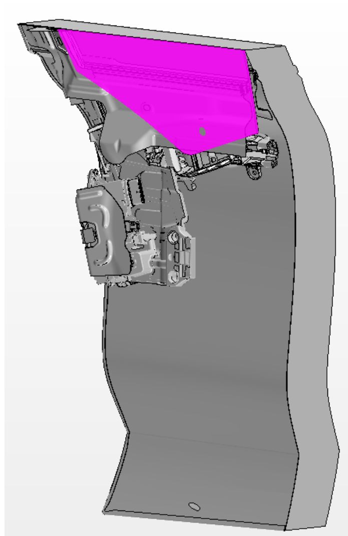


Figure 14 Parts related to the latch system **Figure 15 Highlighting the window**

Save simulation

4 Surface wrapper

The surface wrapper process needs to be done before generating the volume mesh.

4.1 Default Controls

Settings that differ from default controls are listed in Table 1 below.

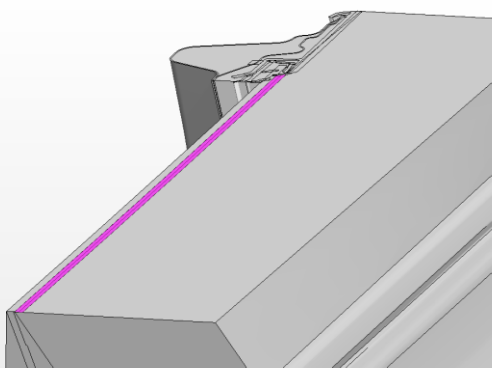
Table 1 Surface wrapper settings

Setting	**Value**
Base size	6 mm
Volume of interest	Seed point

4.2 Custom controls

Settings that differ from custom controls are listed below.

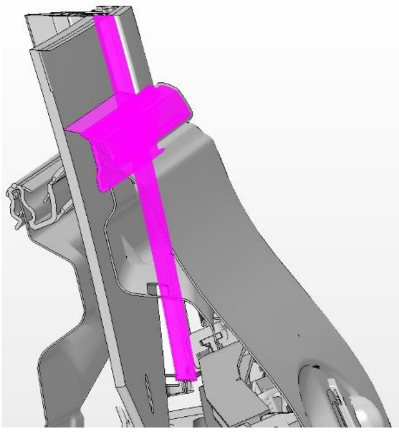
A surface control is set on both water inlets to make sure the surface quality is fine, which is illustrated in Figure 16.



Setting	Value (% of base size)
Target surface size	1 mm (16.66 %)
Minimum surface size	0.1 mm (1.66 %)

Figure 16 Separating water inlets

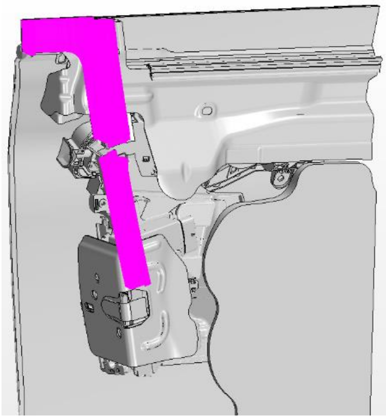
To prevent the air and water inlet surfaces from merging together a target surface is applied on the surface acting as air inlet. It is illustrated in Figure 17.



Setting	Value (% of base size)
Target surface size	2 mm (33.33 %)
Minimum surface size	0.2 mm (3.33 %)

Figure 17

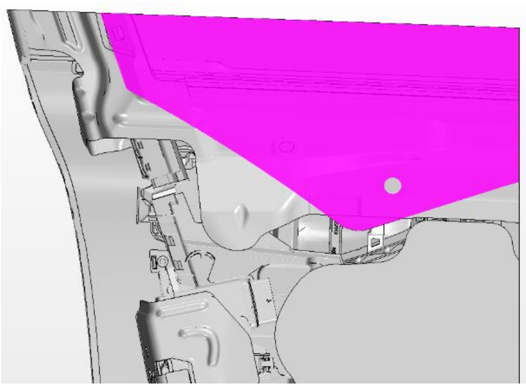
To ensure that parts which will be exposed to water have a fine surface quality, a custom control is set on those parts. These are colored purple in Figure 18.



Setting	Value (% of base size)
Target surface size	1 mm (16.66 %)
Minimum surface size	0.1 mm (1.66 %)

Figure 18 Target surface size of parts

The surface quality of the window is refined due to the fact that water will stream down the window. The window part is shown in Figure 19.



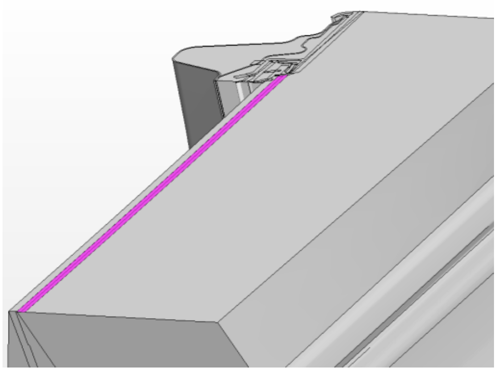
Setting	Value (% of base size)
Target surface size	3 mm (50 %)
Minimum surface size	0.5 mm (8.33 %)

Figure 19 Refining the window surface

4.3 Contact prevention

Settings that differ from the default contact prevention settings are listed below.

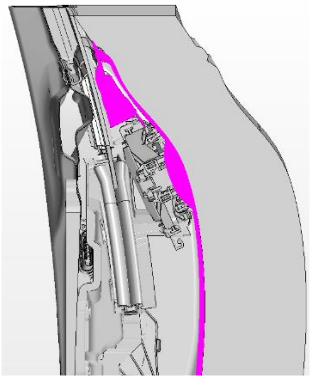
To make sure that the different water inlets are separated from each other a surface control is set, which is illustrated in Figure 20.



Setting	Value
Minimum size	1.0E-4 m

Figure 204.1 Contact prevention at the water inlet

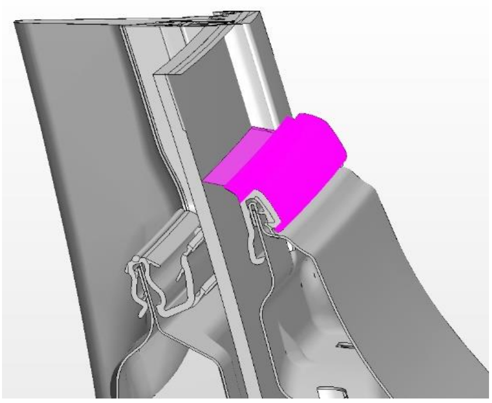
To prevent the outer metal sheet from merging into an inner metal sheet a contact prevention is established, shown in Figure 21.



Setting	Value
Target surface size	1 mm

Figure 21 contact prevention of two metal sheet

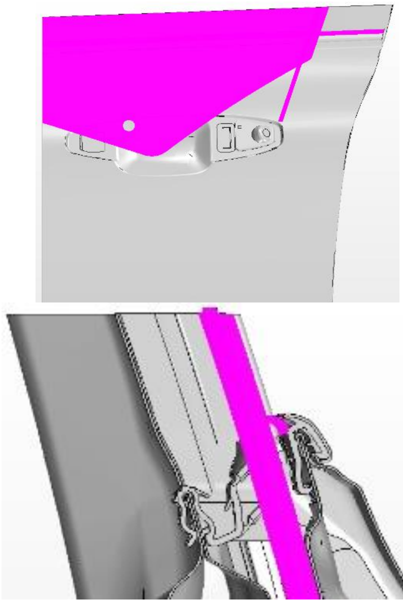
To prevent the sealings from merging together a contact prevention is created, which is shown in Figure 22



Setting	Value
Target surface size	1 mm

Figure 22 Contact prevention between the sealings

To prevent the sealings and window from merging together a contact prevention is created, which is shown in Figure 23.



Setting	Value
Target surface size	0.01 mm

Figure 23 Contact prevention between sealings and window.

Save the simulation

5 Volume Mesh

5.1 Meshers

A volume mesh is applied with the settings listed in Table 2.

Table2 Settings for volume mesh

Setting	**Value**
Surface remesher	Active
Automatic surface repair	Active
Trimmed cell mesher	Active
Prism layer mesher	Active

5.2 Default controls

The default values that have been used are displayed in Table 3.

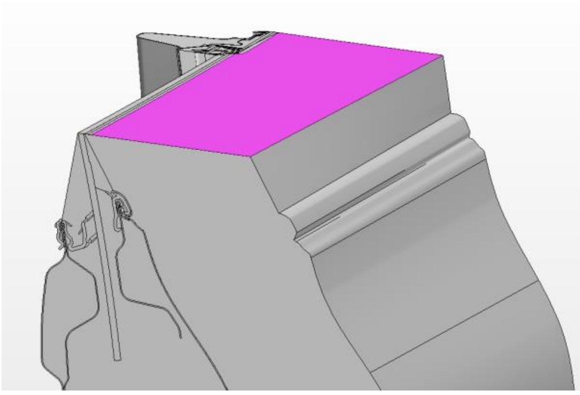
Table 3 Surface control settings regarding volume mesh

Setting	**Value (% of base size)**
Base size	9 mm
Surface growth rate	1.3
Number of prism layers	5
Prism layer stretching	1.5
Prism layer total thickness	1 mm (11.11 %)

5.3 Surface control

Settings of the surface control regarding the volume mesh that differ from standard are listed below.

The air inlet surface is set to disable prism layer in order to reduce the number of cells. This is illustrated in Figure 24.



Setting	Value
Disable prism layer	active

Figure 24 Surface control at the air inlet

An increased number of prism layers is set on the window in order to get a better calculation. This is shown in Figure 25.

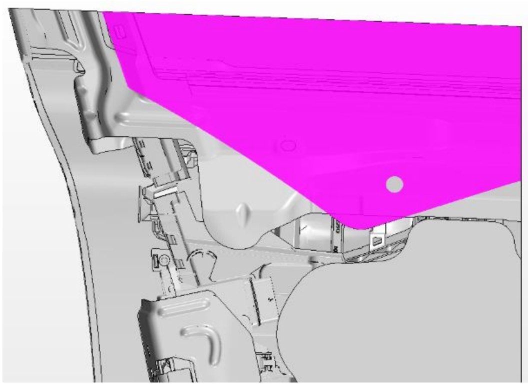
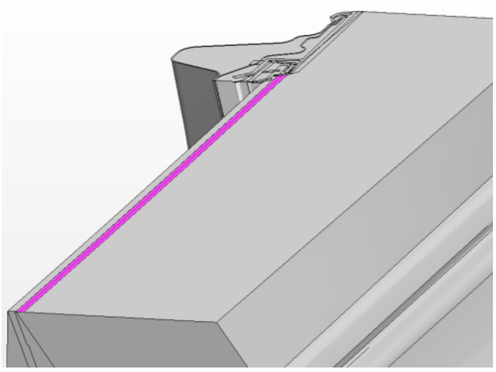


Figure 25 Surface control at the window

Setting	Value (% of base size)
Number of prism layers	8
Prism layer total thickness	2.5 mm (27.77%)

The surface of the water inlet is refined to get a continuous flow into the geometry. This is done by creating a target surface size which is shown in Figure 26.



Setting	Value (% of base size)
Target surface size	2.7E-4 m (3 %)

Figure 26 Surface control size at water inlet

5.4 Volumetric controls

Settings of the volumetric control regarding the volume mesh that differ from standard are listed below.

Boxes are created to cover volumes where the water flow is expected to stream. The cell size within these boxes are reduced compared to the global size. An illustration of the positions of the boxes is shown in Figure 27 and Figure 28.

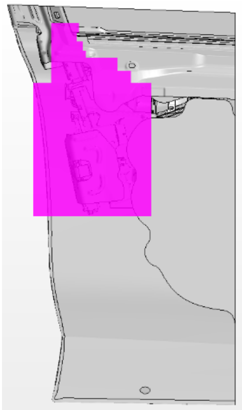


Figure 27 Refining cell, size, back view

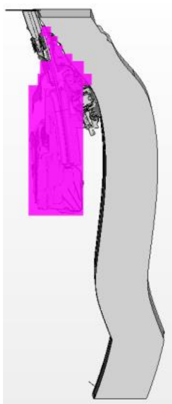


Figure 28 Refining cell size, side view

Setting	Value (% of base size)
Custom size	0.675 mm (7.5 %)

An even smaller cell size is created within the boxes as shown in Figure 29 and Figure 30.

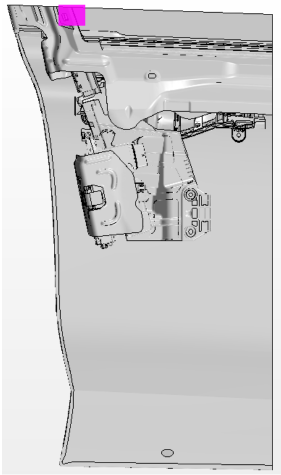


Figure 29 Refining cell size smaller boxes, back view

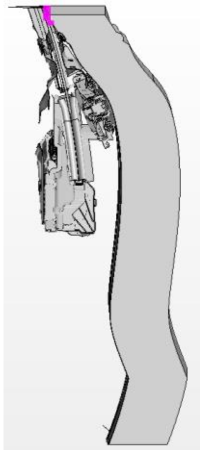


Figure 30 Refining cell size smaller boxes, side view

Setting	**Value (% of base size)**
Custom size	0.27 mm (3 %)

Save the simulation

6 Continua

Physics that are not default are listed in Table 4.

Table 6.1 Physics settings

I. Cell quality remediation
II. Convective CFL Time-Step Control
III. Eulerian multiphase
    a. Air - constant density, gas, turbulent
        i. Constant density
        ii. Gas
        iii. Turbulent
    b. Water - constant density, liquid, turbulent
        i. Constant density
        ii. Liquid
        iii. Turbulent
IV. Exact wall distance
V. Gradient
VI. Gravity
VII. Implicit unsteady
VIII. K-Epsilon turbulence
IX. Multiphase equation of state
X. Multiphase interaction
    a. Phase interaction

i. Multiphase material,
ii. Surface tension force
iii. VOF-VOF phase interaction

XI. Realizable K-Epsilon two-layer
XII. Reynolds-averaged navier-stokes
XIII. Segregated flow
XIV. Three dimensional
XV. Turbulent
XVI. Two-layer all y+ wall treatment
XVII. Volume of fluid (VOF)

The initial volume of the geometry is set to 100 % air and the gravity is acting in the negative z-direction.

7 Boundaries

All surfaces that are not explained below are set as default.

7.1 Water inlet

The surfaces which is acting as water inlet is set to inject 100 % water, with a water mass flow rate per second in kg/s. (This value is confidential and is therefore left out in this version of the report.)

7.2 Air inlets

Surfaces which is acting as air inlets are set to inject 100 % air with a velocity in m/s. (This value is confidential and is therefore left out in this version of the report.)

7.3 Outlets

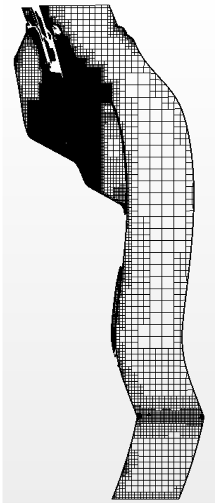
Outlets surfaces are set to pressure outlet with a backflow set to volume fraction of 100 % air.

Save the simulation

8 Derived parts

For the geometry an isosurface is created, which will visualize the water during simulation.

Create a plane section located in the middle of the geometry, with the ability to study the mesh. A resulting plane section is shown in Figure 31 and Figure 32.



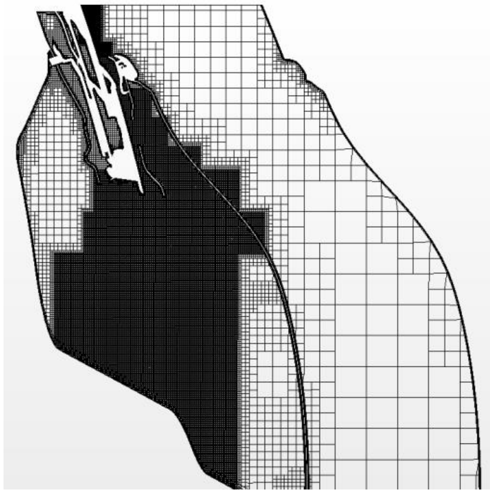


Figure 31 Overview of a mesh plane **Figure 32 Closer view of a mesh plane**

9 Solvers

In order to have a more flexible step time, time-step control is selected. This function automatically changes the time step depending on the CFL number. The values that have been used are shown in Table 5. The lowest acceptable time-step is set to 0.001s.

Table 5 Time-step control settings

Settings	**Values**
Target Mean CFL Number	5
Target Max CFL Number	6

10 Stopping Criteria

The simulation length is set as the stopping criteria and settings that differ from default are shown in Table 6.

Table 6 Stopping criteria

Settings	**Values**
Maximum inner iterations	5
Maximum physical time	30 s
Maximum steps	30000

11 Set up for scenes

Depending on what result is to be studied, different views are selected. Views that are recommended in this case are listed in Table 7.

Table 7 Suggested view

View
Geometry scene
Mesh scene
Scalar scene

The volume fraction of water that was used during the simulation to vizulize the water, was set to a range between 0.5 and 1.

When saving pictures of the simulaiton the trigger was set to “Time step” with a frequency of 10 to all scenes.

Save the simulation

12 Delete cells

When generating a volume mesh, it is possible that even improper cells are created. In order to delete cells that are invalid the function “Remove invalid cells” is used. Settings that are used with this function is shown in Table 8.

Table 8 Settings to remove invalid cells

Settings	Values
Contiguous cells	1000
Connected face	10e-10 m^2

Table 12.1 Settings to remove invalid cells

13 Run the simulation

When you reach this point, it is time to start the simulation.

14 Previous result

By using settings as described, previous simulations have generated the result shown in Figure 33 – Figure 34.

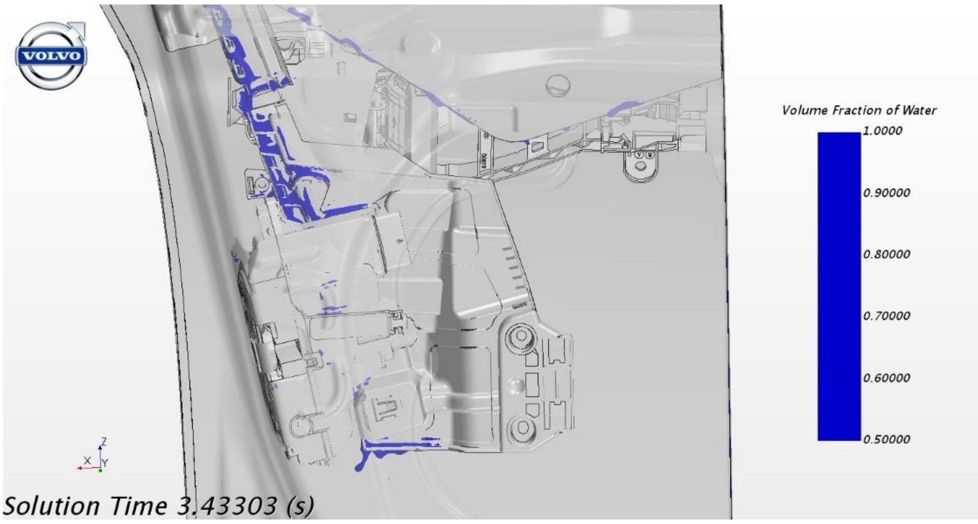


Figure 33 Water flow, back side

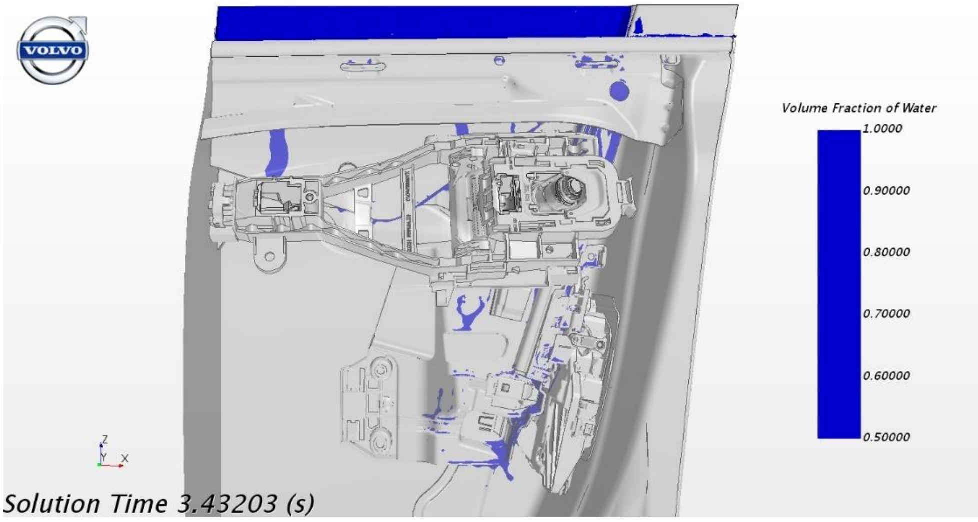


Figure 34 Water flow, front view

C.2 Template for the rear door

Rear door simulation using STAR-CCM+

General Information about the process

Project	model		**Created**	2018-02-26
Needed part	Parts regarding the structure of the rear doors.		**STAR-CCM+ version**	11.06.010
Purpose	Study if water is streaming on the latch system inside the door.		**Assumed performing time**	5 days
Requirement	It is not allowed to have a continuous water flow streaming in the latch system.		**Previous employers performed the process**	Thesis students, Erik Nilvé and Daniel Persson
Dept/name	Side doors	name	**Required software**	TeamCenter CATIA Star-CCM+

Timeline for the process

CAD (20%)	Surface wrapper (20 %)	Mesh (25 %)	Boundaries (5 %)	Scenes (10 %)	Running (20 %)

1 CAD preparation

Start by exporting the parts shown in Figure 1 and Figure 2 from Team Center. Insert the parts that make up the door structure.

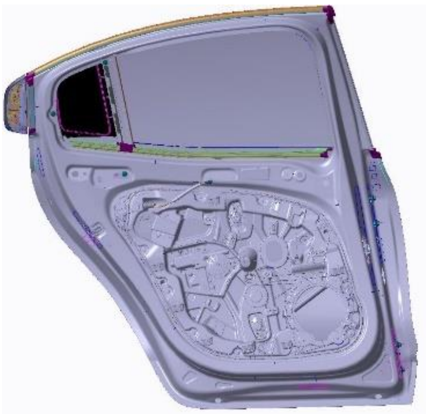


Figure 1 Overview of needed parts, view from outside.

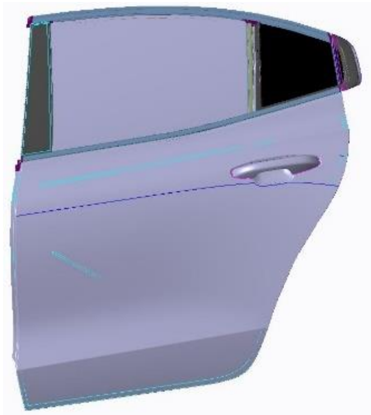


Figure 2 Overview of needed parts, view from outside

Inside CATIA delete parts that are unnecessary to the initial geometry and parts that are not of interest to the simulation. All parts inside the door can be deleted except for the parts related to the latch system. At the same time as the parts are deleted, cut down irrelevant volumes of the geometry. The result of the trimming can be studied in Figure 3 and Figure 4.

Figure 3 Trimmed geometry, view from inside

Figure 4 trimmed geometry, view from outside

A closer view of the parts that need to remain is shown in Figure 5.

Figure 1.1 Closer view of parts to remain

Surfaces acting as water inlets need to be created in order to prepare the geometry for simulation. The first surface (beige) is extruded from the edge of the top area of the window. It is extruded horizontally in a direction away from the window acting as a water inlet. The second surface (red) is extruded from the free edge of the first surface, in the same direction as the first. At the bottom of the geometry a surface outlet is extruded in the same direction as the two others. All these surfaces are shown in Figure 6.

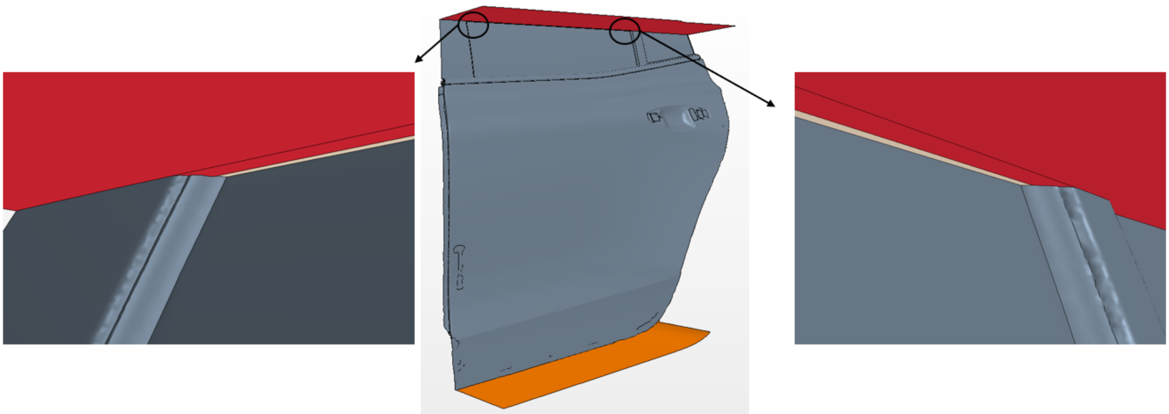
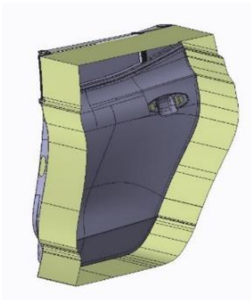
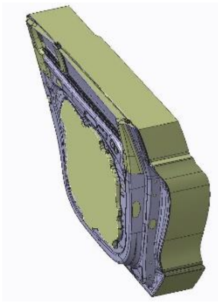


Figure 6 Inlet and outlet on the outer side of the door

To run the simulation a closed volume is desired. Create contour surfaces to generate a closed volume. Be sure that the boundary conditions do not impact the water flow, but keep the volume as tight as possible to save simulation capacity. The geometry is closed except for one surface which will be created inside SCCM. A result of the geometry from CATIA is shown in three different views in Figure 7- Figure 9. All surfaces that are highlighted in yellow are surfaces that are created in order to create a closed volume.





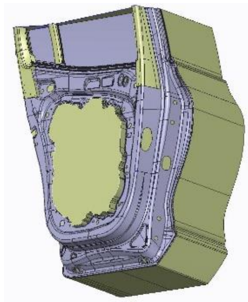


Figure 1.2 Closed volume, side view

Figure 1.3 Closed volume, top view

Figure 1.4 Closed volume, back view

To close the geometry completely, the last surfaces need to be created inside SCCM. This is done by using “Repair surface” and then filling the hole with the function “fill holes using selected edges”. To get a more proper mesh the function “remesh selected surfaces” is used. The result of the closed volume inside SCCM is shown in Figure 10

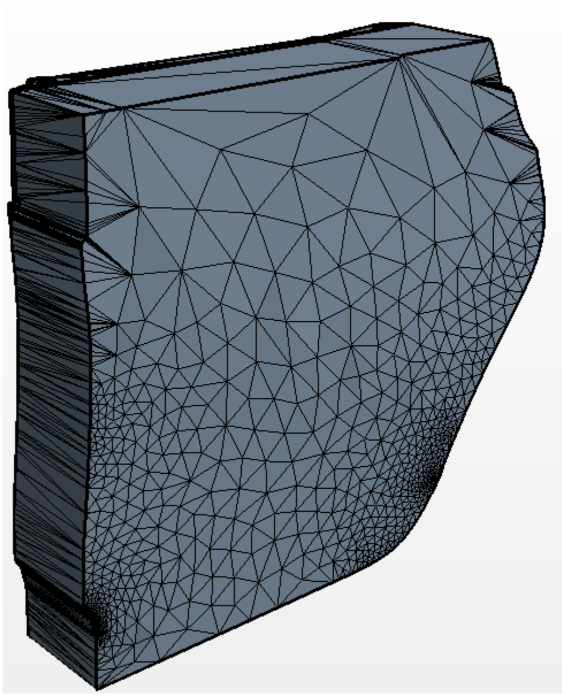
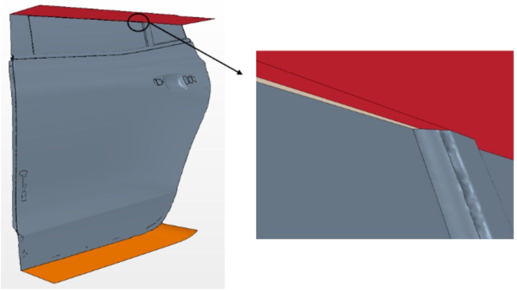


Figure 10 Closed volume

Save the file as a STEP-file

2 Selecting parts and rename

Import the geometry from CATIA as a STEP file into STAR-CCM+. Select and rename parts to the purpose of the surface. It could be surfaces that will act as inlets, outlets and other surfaces with a special mesh treatment. In Figure the surface highlighted in beige is acting as a water inlet and the red is acting as an air inlet. The orange surfaces in Figure 11 and Figure 12 are acting as outlets.



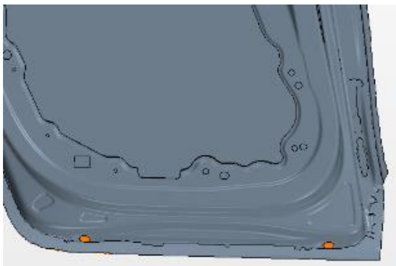
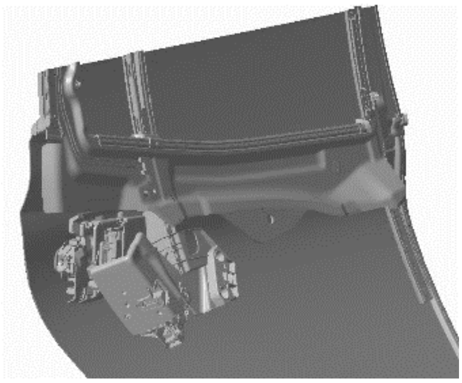


Figure 11 Showing inlets and outlets **Figure 12 Outlet on the backside**

To facilitate next steps, it is recommended to rename parts related to the latch system shown in Figure 13. It is also recommended to rename the window due to the same reason, which is shown in Figure 14.



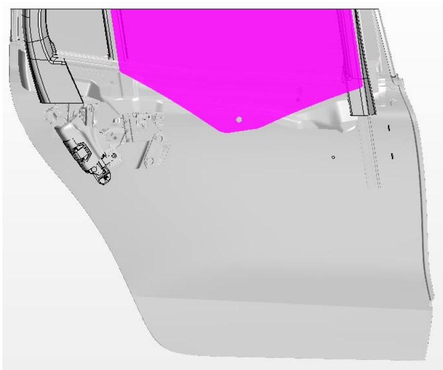


Figure 13 Parts related to the latch system **Figure 14 Highlighting the window**

<u>Save simulation</u>

3 Surface wrapper

The surface wrapper process needs to be done before generating the volume mesh.

3.1 Default Controls

Settings that differ from default controls are listed in Table 1 below.

Table 1 Surface wrapper settings

Setting	**Value**
Base size	10 mm
Volume of interest	Seed point

3.2 Custom controls

Settings that differ from default custom controls are listed below.

A surface control is set on the water inlet to make sure the surface quality is fine, which is illustrated in Figure 15.

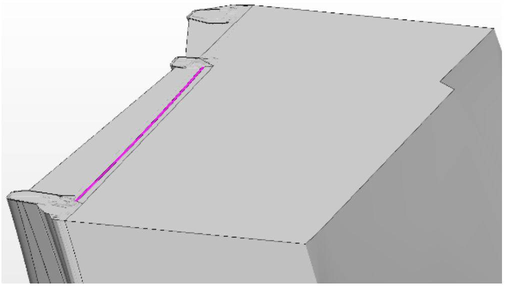


Figure 15 Separating water inlets

Setting	Value (% of base size)
Target surface size	1 mm (10 %)
Minimum surface size	0.1 mm (1 %)

To prevent the sealings from merging together a target surface is applied on the surfaces acting as sealings. Target surface is also applied on the latch system surfaces with the same purpose. Both cases could be illustrated in Figure 16.

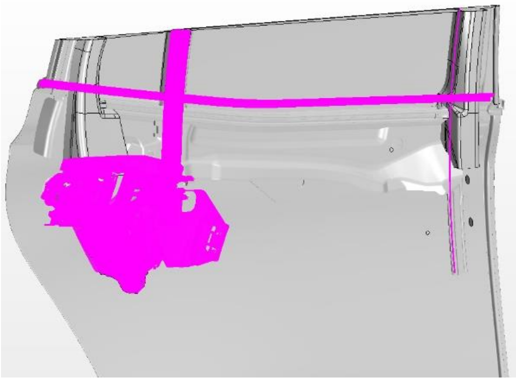
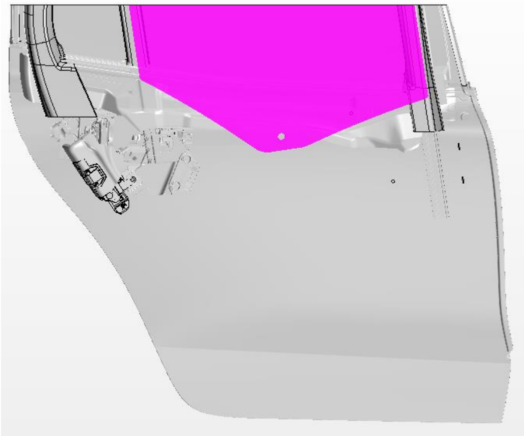


Figure 16

Setting	Value (% of base size)
Target surface size	2 mm (20 %)
Minimum surface size	0.2 mm (2%)

The surface quality of the window is refined due to water streaming down the window, which is shown in Figure 17.



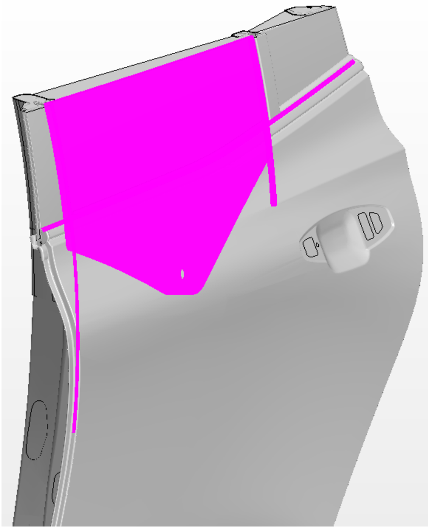
Setting	Value (% of base size)
Target surface size	3 mm (30 %)
Minimum surface size	0.3 mm (3 %)

Figure 17 Refining the window surface

3.3 Contact prevention

Settings that differ from default contact preventions are listed below.

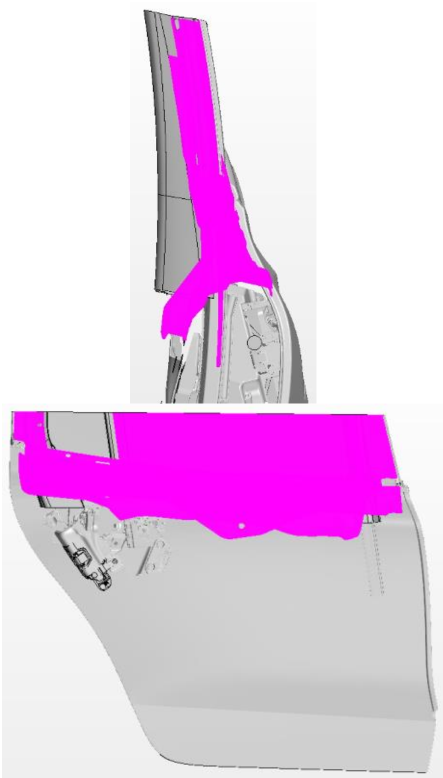
To prevent sealings and window from merging together a contact prevention is created, which is shown in Figure 18.



Setting	Value
Minimum size	0.1 mm

Figure 18 Contact prevention between sealings and window

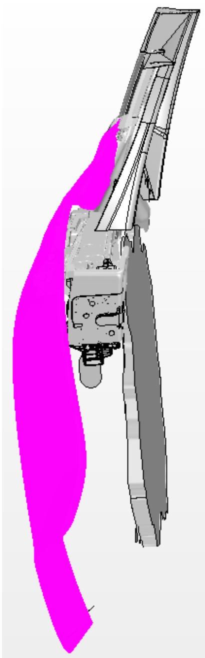
To prevent the outer metal sheet from merging together with an inner metal sheet a contact prevention is established, as shown in Figure 19.



Setting	Value
Target surface size	1 mm

Figure 19 contact prevention of two metal sheet

To prevent the outer metal sheet from merging together with an inner metal sheet a contact prevention is established, as shown in Figure 20



Setting	Value
Target surface size	2 mm

Figure 20 Contact prevention between outer and inner metal sheets

Save the simulation

4 Volume Mesh

4.1 Meshers

A volume mesh is applied with the settings listed in Table 2.

Table 2 Settings for volume mesh

Setting	Value
Surface remesher	Active
Automatic surface repair	Active
Trimmed cell mesher	Active
Prism layer mesher	Active

4.2 Default controls

Default values that have been used are displayed in Table 3.

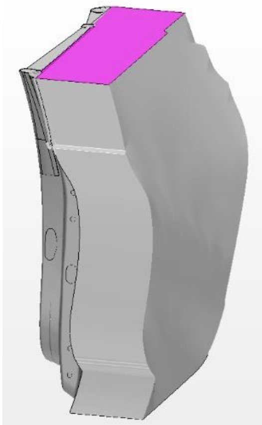
Table 3 Surface control settings regarding volume mesh

Setting	Value (% of base size)
Base size	9 mm
Surface growth rate	1.3
Number of prism layers	5
Prism layer stretching	1.5
Prism layer total thickness	1.5 mm (16.66 %)

4.3 Surface control

Settings of the surface control regarding the volume mesh that differ from standard values are listed below.

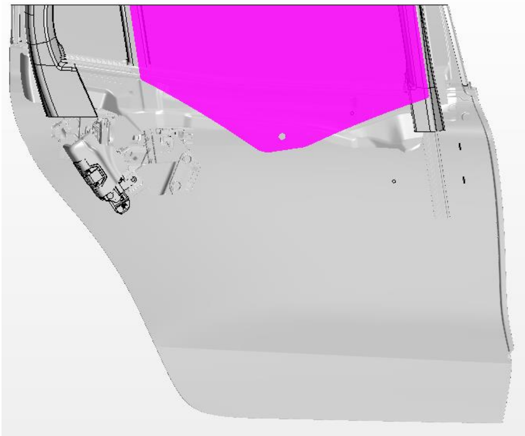
The air inlet surface is set to disable prism layer in order to reduce the number of cells, which is illustrated in Figure 21.



Setting	Value
Disable prism layer	active

Figure 21 Surface control at the air inlet

An increased number of prism layers is set on the window in order to get a better calculation, which is shown in Figure 22.



Setting	Value (% of base size)
Number of prism layers	8
Prism layer total thickness	2.5 mm (27.77%)

Figure 22 Surface control at the window

The surface of the water inlet is refined to get a continuous flow into the geometry. This is done by creating a target surface size which is shown in Figure 23.

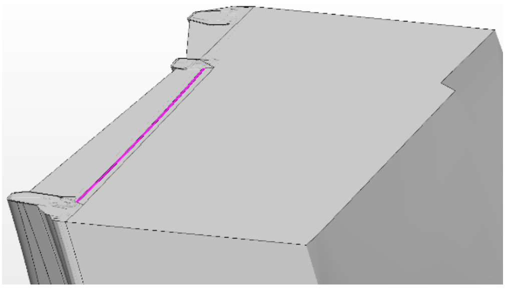


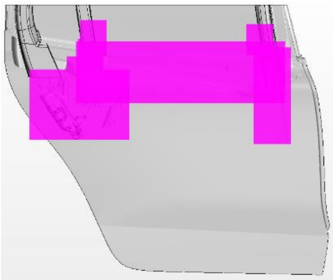
Figure 23 Surface control size at water inlet

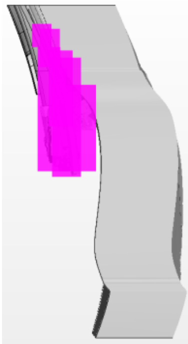
Setting	Value (% of base size)
Target surface size	2.7E-4 m (3 %)

4.4 Volumetric controls

Settings of the volumetric control regarding the volume mesh that differ from standard are listed below.

Boxes are created to cover volumes where the water flow is expected to stream. The cell size within these boxes are reduced compared to the global size. An illustration of the positions of the boxes are illustrated in Figure 24 and Figure 25.



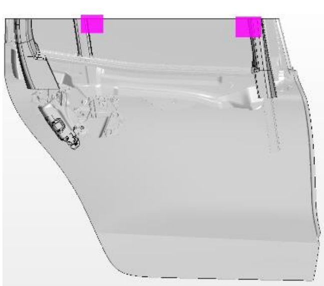


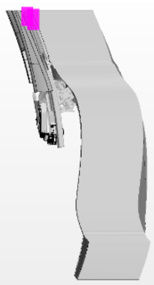
Setting	Value (% of base size)
Custom size	0.675 mm (7.5 %)

Figure 24 Refining cell, size back view

Figure 25 Refining cell, size side view

An even smaller cell size is created within the boxes as shown in Figure 26 and Figure 27.





Setting	Value (% of base size)
Custom size	0.27 mm (3 %)

Figure 26 Refining cell, size, smaller boxes, back view

Figure 27 Refining cell, size, side view side view

<u>Save the simulation</u>

5 Continua

Physics that are not default are listed in list Table 4.

Table 4 Physics settings

XVIII. Cell quality remediation
XIX. Convective CFL Time-Step Control
XX. Eulerian multiphase
    a. Air - constant density, gas, turbulent
        i. Constant density
        ii. Gas
        iii. Turbulent
    b. Water - constant density, liquid, turbulent
        i. Constant density
        ii. Liquid

iii. Turbulent
XXI. Exact wall distance
XXII. Gradient
XIII. Gravity
XIV. Implicit unsteady
XXV. K-Epsilon turbulence
XVI. Multiphase equation of state
XVII. Multiphase interaction
a. Phase interaction
i. Multiphase material,
ii. Surface tension force
iii. VOF-VOF phase interaction
VIII. Realizable K-Epsilon two-layer
XIX. Reynolds-averaged navier-stokes
XXX. Segregated flow
XXI. Three dimensional
XXII. Turbulent
XIII. Two-layer all y+ wall treatment
XIV. Volume of fluid (VOF)

The initial volume of the geometry is set to 100 % air and the gravity is acting in the negative z-direction.

6 Boundaries

All surfaces that are not explained below are set as default.

6.1 Water inlet

The surfaces which are acting as water inlet is set to inject 100 % water, with a water mass flow rate in kg/s. (This value is confidential and will therefore be left out in this version of the report).

6.2 Air inlets

Surfaces which is acting as air inlets are set to inject 100 % air with a velocity in m/s. (This value is confidential and will therefore be left out in this version of the report).

6.3 Outlets

Outlet surfaces are set to pressure outlets with a backflow set to volume fraction of 100 % air.

Save the simulation

7 Derived parts

For the geometry an isosurface is created, which will visualize the water during simulation.

Create a plane section located in the middle of the geometry, with the ability to study the mesh. A resulting plane section is shown in Figure 28 and Figure 29.

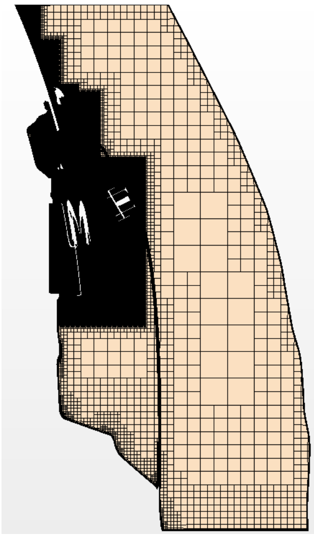


Figure 28 Overview of a mesh plane

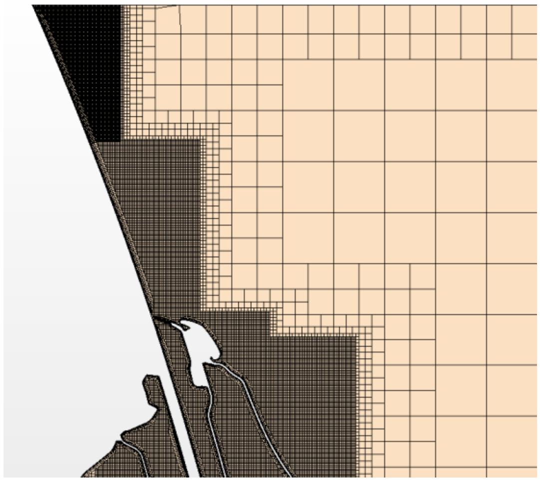


Figure 29 Closer view of a mesh plane

8 Solvers

In order to have a more flexible step time, time-step control is selected. This function automatically changes the time step depending on the CFL number. The values that have been used are shown in Table 5. The lowest acceptable time-step is set to 0.001s.

Table 5 Time-step control settings

Settings	Values
Target Mean CFL Number	5
Target Max CFL Number	6

9 Stopping Criteria

The simulation length is set as stopping criteria, settings that differ from default are shown in Table 6.

Table 6 Stopping criteria

Settings	Values
Maximum inner iterations	5
Maximum physical time	20 s
Maximum steps	30000

10 Set up for scenes

Depending on what result is to be studied, different views are selected. Views that are recommended in this case are listed in Table 7.

Table 7 Suggested views

View
Geometry scene
Mesh scene
Scalar scene

The volume fraction of water that was used during the simulation to vizulize the water, was set to a range between 0.5 and 1.

When saving pictures of the simulaiton the trigger was set to “Time step” with a frequency of 10 to all scenes.

Save the simulation

11 Delete cells

When generating a volume mesh, it is possible that even improper cells are created. In order to delete cells that are invalid, the function “Remove invalid cells” is used. Settings that are used with this function are shown in Table 8.

Table 8 Settings to remove invalid cells

Settings	Values
Contiguous cells	1000

Connected face	10e-10 m^2

12 Run the simulation

When you reach this point, it is time to start the simulation.

13 Previous result

By using settings as described, previous simulations have generated the result shown in Figure 30 – Figure 31.

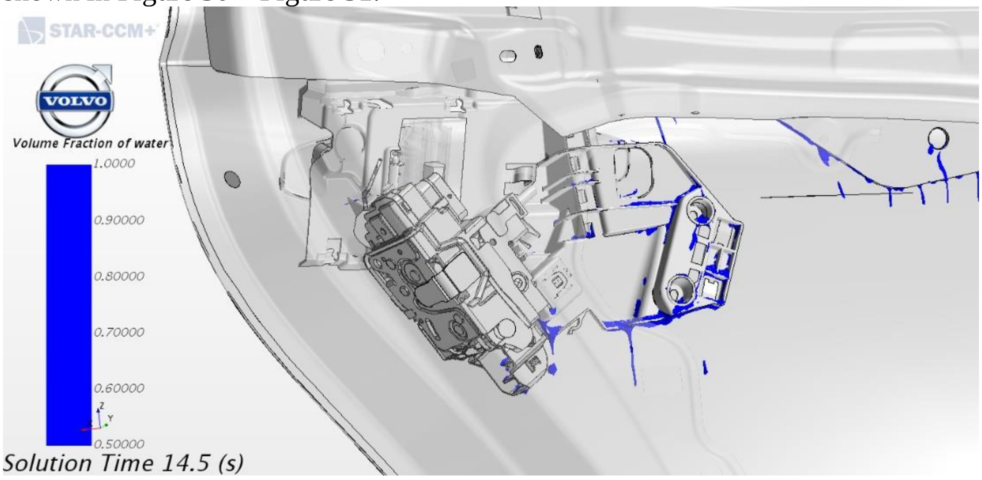


Figure 30 Water flow, back view

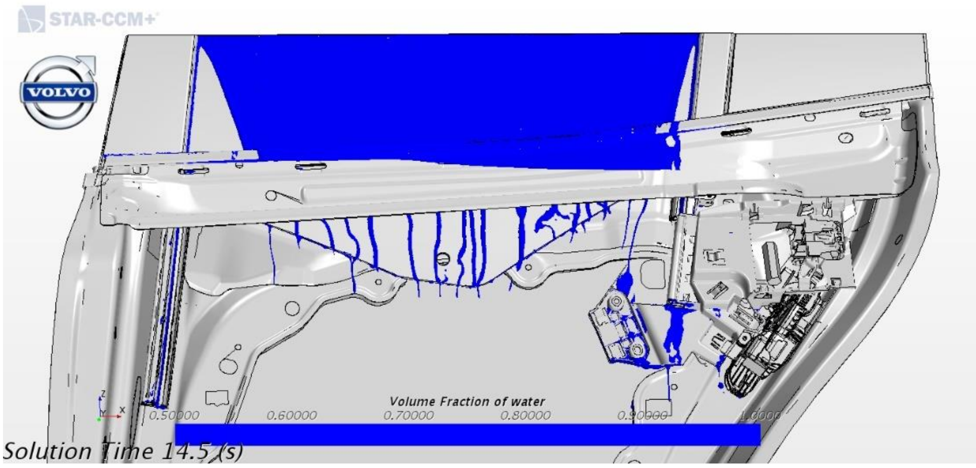


Figure 31 Water flow, front view

C.3 Template for the pipe

Pipe simulation using STAR-CCM+

General Information about the process

Project	model		**Created**	2018-02-26
Needed part	Drainage pipes with connections on both sides.		**STAR-CCM+ version**	11.06.010
Purpose	Study the pipes capacity and how the water is behaving inside the pipes.		**Assumed performing time**	1-2 days
Requirement	Each pipe should handle a volume of water per time unit.		**Previous employers performed the process**	Thesis students, Erik Nilvé and Daniel Persson
Dept/name	Roof systems and mirror	name	**Required software**	TeamCenter CATIA Star-CCM+

Timeline for the process

CAD (25 %)	Surface mesh (10 %)	Mesh (10 %)	Boundaries (5 %)	Scenes (10 %)	Running (40 %)

1 CAD preparation

Start by exporting parts from Team Center, that are shown in Figure 1. The focus is to import all parts that are connected to the pipes, which is the water collector at the top and the outlet nozzle at the bottom.

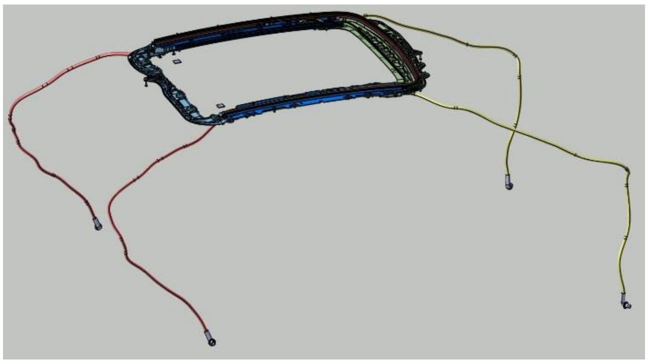


Figure 1, Parts from Team Center

Trim the geometry that is imported from Team Center by deleting parts that are unnecessary for the simulation. The result of a trimmed geometry should look like Figure 2.

Figure 2 Result after reduced the geometry size

A closer view of the ends is illustrated in Figure 3 and Figure 4. Cut the inlet pipe with the same angle as the water collector had before.

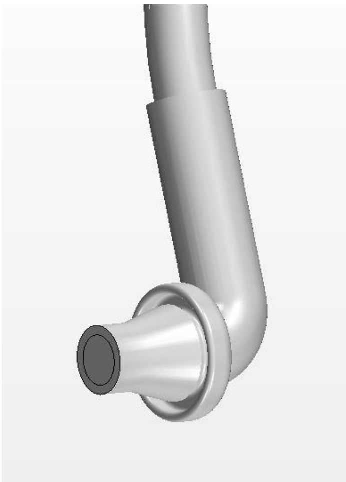


Figure 3 Closer view of the outlet

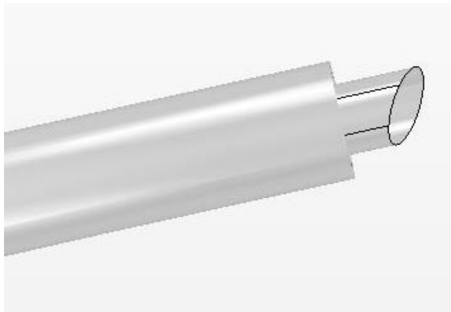


Figure 4 Closer view of the inlet

On the inlet, a box should be built with the same volume as the requirement. The box is constructed with an inclination towards the pipe inlet, to make sure that the water is streaming into the pipe. In Figure 5 an overview of the box is illustrated.

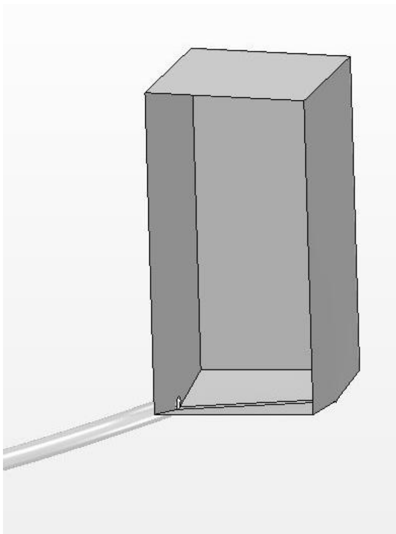


Figure 5 Overview of the box

The only area that needs to be checked in order to have a closed volume is the connection between the pipe and box pipe, since the ends are already closed. By extruding surfaces between the pipe and the box pipe the volume is closed. A closer view of the connection is shown in Figure 6.

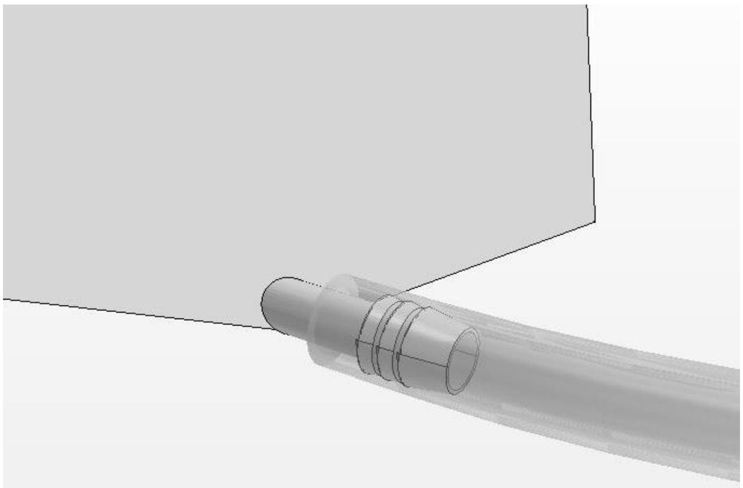


Figure 6 Closer view of the connection between the box and the pipe

Save the file as a STEP-file

2 Selecting parts and rename

Import the geometry from CATIA as a STEP file into STAR-CCM+. Select and rename parts to the purpose of the surface. It could be surfaces that will act as inlets, outlets and other surfaces with a special mesh treatment. Surfaces that are colored purple in Figure 7 are surfaces acting as outlets.

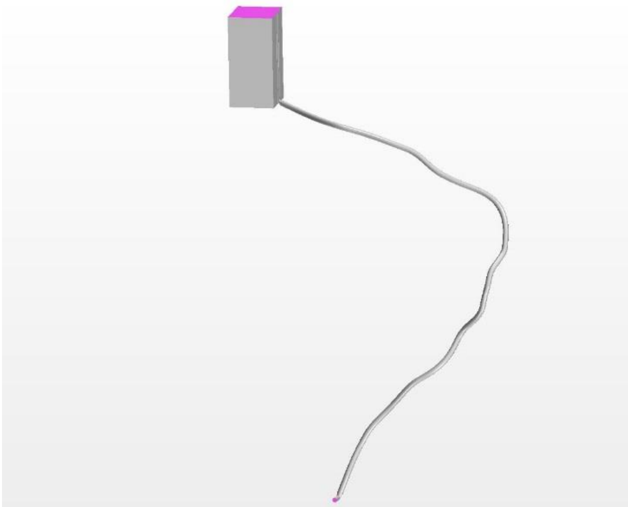


Figure 7 Highlighting inlet and outlets

Save simulation

3 Surface wrapper

The surface wrapper process needs to be done before generating the volume mesh. With surface wrapper it is possible to setup surfaces with custom controls. Settings that differ from default is explained in Table 1.

Table 1 Surface wrapper settings

Setting	**Value**
Base size	2 mm
Volume of interest	Seed point

Save the simulation

4 Volume Mesh

A volume mesh is applied with the settings listed in Table 2.

Table 2 Settings for the volume mesh

Setting	**Value**
Surface remesher	Active
Polyhedral mesher	Active
Prism layer mesher	Active

4.1 Default controls

Settings of the surface control regarding the volume mesh that differ from standard are listed in Table 3.

Table 3 Surface control settings regarding volume mesh

Setting	**Value (% of base size)**
Base size	2 mm
Surface growth rate	1.3
Number of prism layers	3
Prism layer stretching	1.5
Prism layer total thickness	6.665E-4 (33.33 %)

Save the simulation

5 Continua

Physics that are not default are listed in list Table 4.

Table 4 Physics settings

XXV. Cell quality remediation
XVI. Eulerian multiphase
- a. Air - constant density, gas, turbulent
- b. Constant density
- c. Gas
- d. turbulent
- e. Water - constant density, liquid, turbulent
- f. Constant density
- g. Liquid
- h. turbulent

XVII. Exact wall distance
VIII. Gradient
XIX. Gravity
XL. Implicit unsteady
XLI. K-Epsilon turbulence
XLII. Multiphase equation of state
I. Multiphase interaction
II. Phase interaction
- a. VOF-VOF phase interaction
  - i. Primary phase – water
  - ii. Secondary phase - air

LIII. Realizable K-Epsilon two-layer
LIV. Reynolds-averaged navier-stokes
XLV. Segregated flow
LVI. Three dimensional
LVII. Turbulent
VIII. Two-layer all y+ wall treatment
LIX. Volume of fluid (VOF)
- a. VOF waves
  - i. Flat vof wave

The initial volume inside the pipe is set to 100 % air, while the volume inside the box is set to 100 % of water. The gravity is acting in the negative z-direction with the value of 9.81 m/s.

6 Boundaries

Surfaces that are not explained below preserve default values.

6.1 Water inlet

The water during simulation is defined by an initial volume, and no water will be injected during the running of the simulation. The initial volume is defined by a field function that is referring to a new coordinate system, which makes it possible to select what volume that should contain water. The code for the field function is $${Position}(@CoordinateSystem("Laboratory.Cartesian"))[2] < 0 ? 1:0
which says that everything in the negative z-axis should be filled by 100 % water. An illustration of the coordinate system is shown in Figure 8.

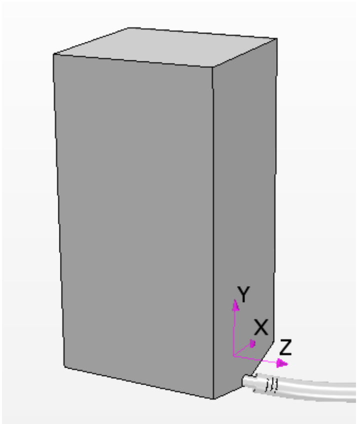


Figure 8 Creating a local coordinate system

6.2 Outlets

Outlet surfaces are set to pressure outlets with a field function called hydrostatic pressure of flat wave, and the backflow for each outlet is set to 100 % of volume fraction of air.

Save the simulation

7 Derived parts

Due to the tube being thin and not positioned along a plane, resampled volume is advantageous to use in order to visualize the volume fraction of water. To study the mesh, a plane sections is created. Two plane sections are created, one to study the mesh inside the box and one to study the mesh inside the pipe. A result of each plane section shows in Figure 9 and Figure 10.

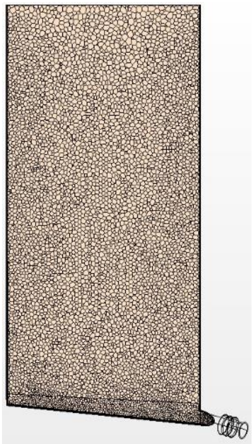


Figure 9 Overview of the mesh on the box

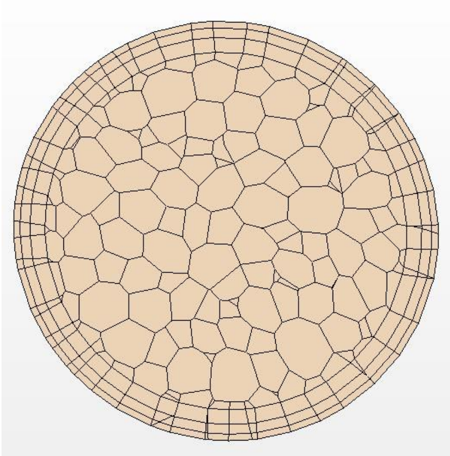


Figure 10 Overview of the mesh on the pipe

8 Solvers

Default settings are used for the trunk case.

9 Stopping Criteria

The simulation length is set as stopping criteria, settings that differ from default are shown in Table 5.

Table 5 Stopping criteria

Settings	Values
Maximum inner iterations	5
Maximum physical time	60 s
Maximum steps	60000

10 Set up for scenes

Depending on what result is to be studied, different views are selected. Views that are recommended in this case are listed in Table 6

Table 6 Suggested views

Geometry scene
Mesh scene

Scalar scene

In the simulation, the volume fraction of water has been set to 0.5 as mimimum and 1 as maximum.

The trigger is set to “Time step” with a frequency of 10 to all scenes that should document the simulation.

Save the simulation

11 Run the simulation

When you reach this point, it is time to start the simulation.

12 Previous result

By using settings as described, previous simulations have generated the result shown in Figure 11 – Figure 12.

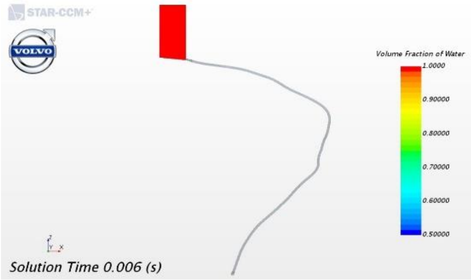


Figure 11 Initial step during simulation

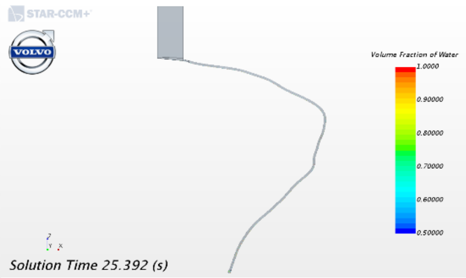


Figure 12 At the end of the simulation

C.4 Template for the trunk

Trunk simulation using STAR-CCM+

General Information about the process

Project	model		**Created**	2018-02-26
Needed part	Exterior parts of the rear section of the car.		**STAR-CCM+ version**	11.06.010
Purpose	Study if the water is entering the luggage area or not.		**Assumed performing time**	5 days
Requirement	No water flow is allowed to enter the luggage area.		**Previous employers performed the process**	Thesis students, Erik Nilvé and Daniel Persson
Dept/name	Side doors	name	**Required software**	TeamCenter CATIA Star-CCM+

Timeline for the process

CAD (20%)	Surface wrapper (20 %)	Mesh (25 %)	Boundaries (15 %)	Scenes (20 %)	Running (20 %)

1 CAD preparation

Start by exporting the shown in Figure 1 from Team Center. The focus is to include all the exterior surfaces where it is possible that water might stream.

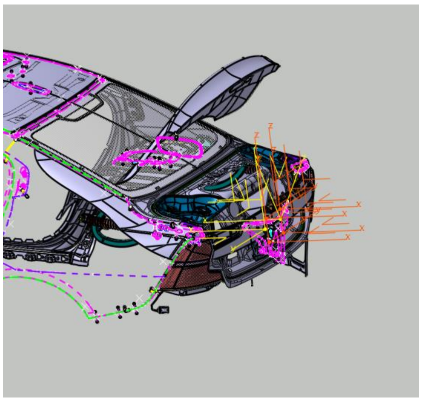


Figure 1, Parts from Team Center

Set the angle of the trunk to the maximum opening angle by using a mechanism tool in CATIA. Divide the geometry from Team Center into two regions, by a plane. Keep the region closest to the trunk. A result of all operations can be studied in Figure 2.

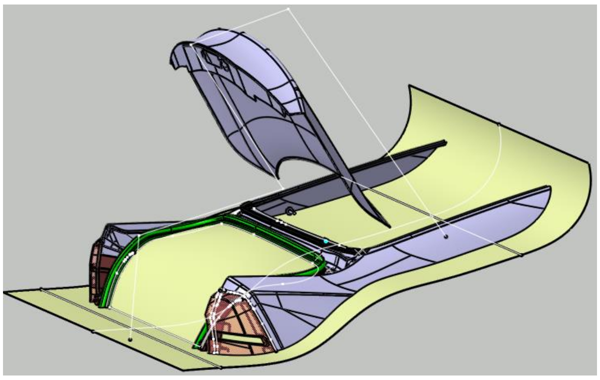


Figure 2, Select needed area of the trunk

Due to symmetry in the geometry it is possible to cut the geometry into half, which will save simulation capacity. Even the window could be reduced once more, by cutting the geometry. The result of an optimized geometry is illustrated in Figure 3 and Figure 4.

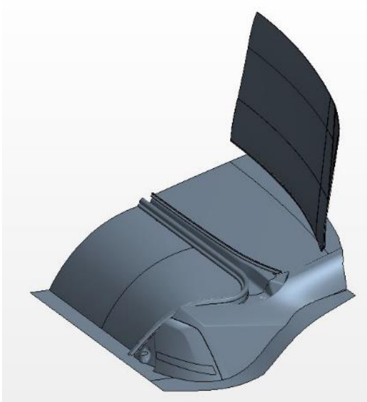


Figure 3 Cut and trimmed geometry, view 1

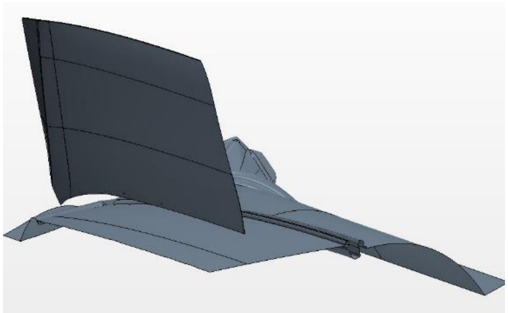


Figure 4 Cut and trimmed geometry, view 2

To prepare the geometry for the simulation, surfaces acting as water inlets are needed. The first surface is extruded from the edge of the top area of the trunk. The direction of the first surface is placed normal to the trunk surface. Next surface is extruded from the edge of the first surface, in the same direction as the first. Both surfaces are divided into two sections along the edge. One section is called middle and the other is called side. These sections will be used differently when injecting the water.

Figure 5 Two water inlets

To run the simulation a closed volume is desired. Create contour surfaces to create a closed volume. Be sure that the boundary conditions do not impact the water flow, but keep the volume as tight as possible to save simulation capacity. The result of the closed volume is shown in Figure 6.

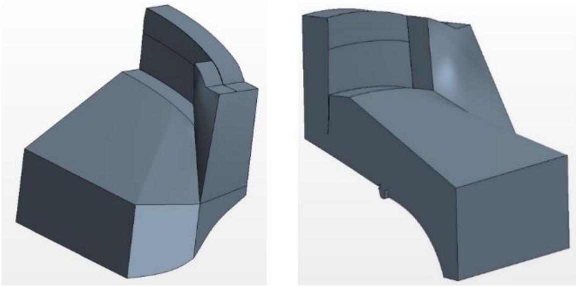


Figure 6 Closed volume

<u>Save the file as a STEP-file</u>

2 Selecting parts and rename

Import the geometry from CATIA as a STEP file into STAR-CCM+. Select and rename parts with to the purpose of the surface. It could be surfaces that will act as inlets, outlets and other surfaces with a special mesh treatment. Surfaces that are colored orange in Figure 7 are surfaces acting as outlets. Surfaces with a red color are acting as air inlets, and surfaces colored beige are acting as water inlets.

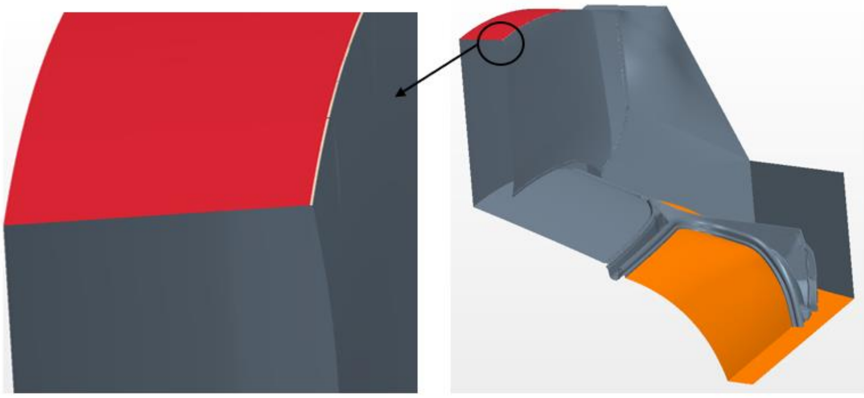


Figure 7 Highlighting inlet and outlets

Surfaces that will have a special treatment are highlighted in Figure 8.Figure 8 Surfaces with special treatment

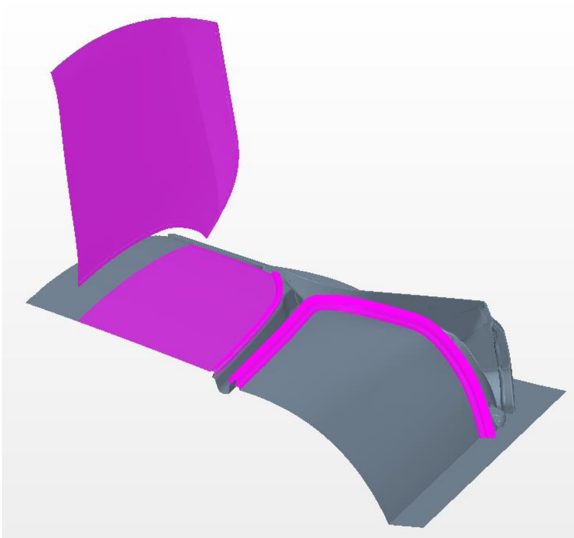


Figure 8 Surfaces with special treatments

Save simulation

3 Surface wrapper

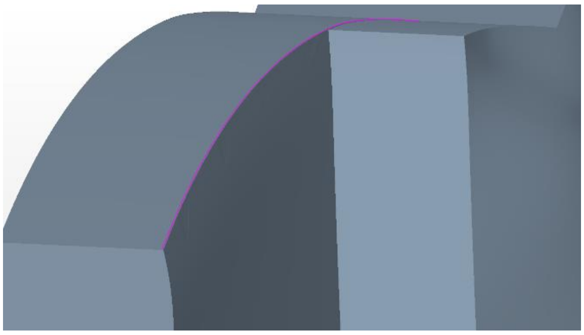
The surface wrapper process needs to be done before generating the volume mesh. With surface wrapper it is possible to setup surfaces with custom controls. Settings that differ from default are explained in Table 1.

Table 1 Surface wrapper settings

Setting	Value
Base size	4 mm
Volume of interest	Seed point

3.1 Custom controls

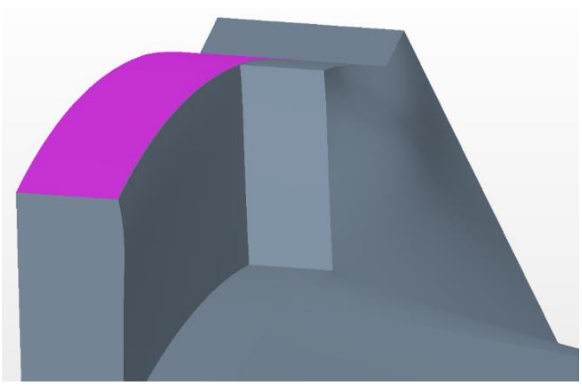
Make sure that water inlets are separated from each other by creating a curve control, which is illustrated in Figure 9.



Setting	Value (% of base size)
Target surface size	4.0E-4 m (10 %)

Figure 9 Curve control in between water inlets

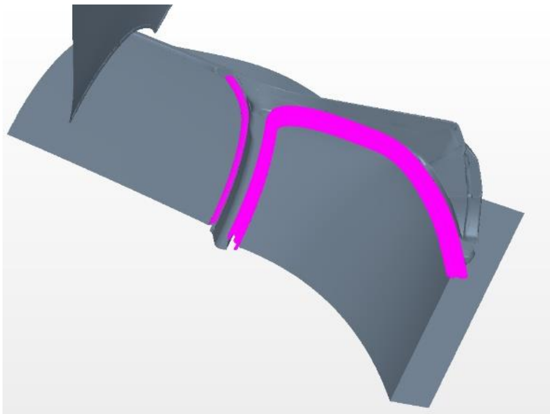
To prevent the air and water inlet surfaces from merging together a target surface is applied on the surface acting as air inlet. It is illustrated in Figure 10.



Setting	Value (% of base size)
Target surface size	2.0E-4 m (5 %)

Figure 10 Target surface size at air inlet

To avoid the rubber sealings from merging together a target surface is applied at surfaces marked purple in Figure 11.



Setting	Value (% of base size)
Target surface size	0.001 m (25 %)

Figure 11 Refines surface at the sealings

Save the simulation

4 Volume Mesh

A volume mesh is applied with the settings listed in Table 2.

Table 2 Settings for volume mesh

Setting	Value
Surface remesher	active
Trimmed cell mesher	active
Prism layer mesher	Active

4.1 Surface control

Settings of the surface control regarding the volume mesh that differ from standard are listed in Table 3.

Table 3 Surface control settings regarding volume mesh

Setting	Value (% of base size)
Base size	8 mm
Surface growth rate	1.3
Number of prism layers	3
Prism layer stretching	1
Prism layer total thickness	0.002666 (33.33 %)

The size of the cells in the prism layer will be smaller while the thickness of the prism layer is reduced. This is illustrated in Figure 12.

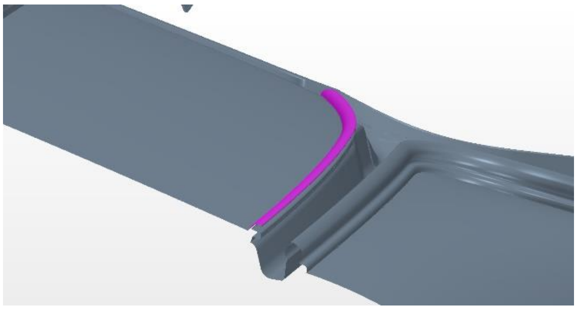


Figure 12 Reducing the prism layer thickness

Setting	Value (% of base size)
Number of prism layers	3
Prism layers stretching	1.5
Prism layer total thickness	8.0E-4 m (10%)

In order to get a more accurate result, more prism layers are applied on the outer surface of the trunk, which is illustrated in Figure 13.

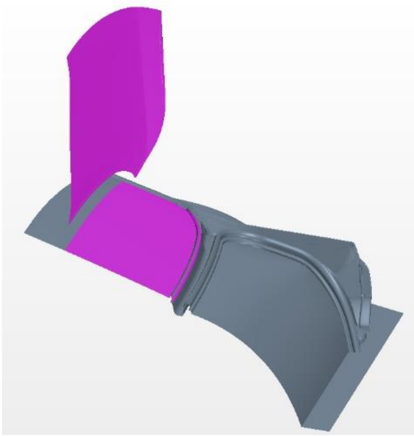
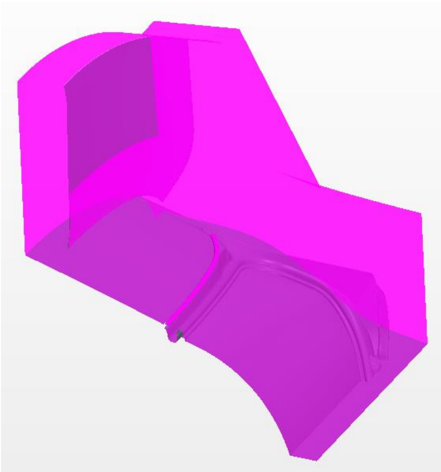


Figure 13 More prism layers is applied on areas of importance

Setting	Value (% of base size)
Number of prism layers	8
Prism layers stretching	1.5
Prism layer total thickness	0.004 m (50%)

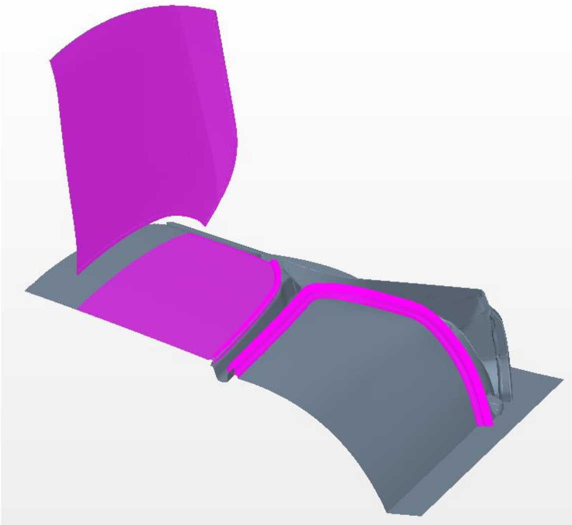
By disabling the prism layer at all boundary surfaces, the number of cells is slightly reduced, which is shown in Figure 14.



Setting	Value
Prism layers	disable

Figure 14 Disable prism layers to reduce number of cells

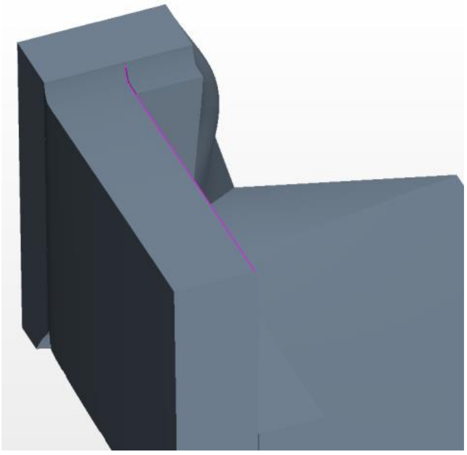
The result will become more accurate when refining the mesh size at areas that are of interest. This is shown in Figure 15



Setting	Value (% of base size)
Target surface size	8.0E-4 m (10%)
Minimum surface size	4.0E-4 m (5%)

Figure 15 Refining mesh size at important areas

Both water inlet surfaces are refined in order to get a proper surface quality in the area of the water injection, which is illustrated in Figure 16

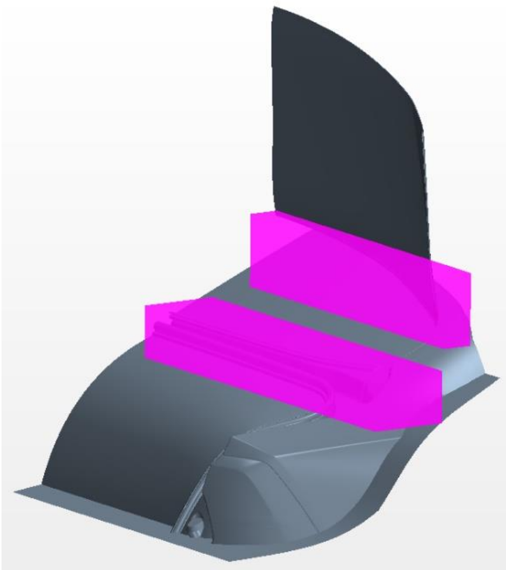


Setting	Value (% of base size)
Target surface size	5.6E-4 m (7%)

Figure 16 Refinment at the water injection

4.2 Volumetric controls

Create boxes that cover the water flow from the trunk to the window and at the area of the gap. Use the created boxes to define volumetric controls, which is shown in Figure 17.



Setting	Value (% of base size)
Custom size (trimmed)	8.0E-4m (10%)

Figure 17 Volumetric controls

Save the simulation

5 Continua

Physics that are not default are listed in list Table 4.

Table 4 Physic settings

L. Cell quality remediation
LI. Eulerian multiphase
   a. Air - constant density, gas, turbulent
   b. Water - constant density, liquid, turbulent
LII. Exact wall distance
LIII. Gradient
LIV. Implicit unsteady
LV. K-Epsilon turbulence
LVI. Multiphase equation of state
LVII. Multiphase interaction
LVIII. Phase interaction - Multiphase material, surface tension force, VOF-VOF phase interaction
LIX. Realizable K-Epsilon two-layer
LX. Reynolds-averaged navier-stokes
LXI. Segregated flow
LXII. Three dimensional
LXIII. Turbulent
LXIV. Two-layer all y+ wall treatment
LXV. Volume of fluid (VOF)
   a. CFL_l = 200
   b. CFL_u = 800

The initial volume of the geometry is set to 100 % air and the gravity is acting in the negative z-direction.

6 Boundaries

All surfaces that are not explained below are set as default.

6.1 Water inlet

The surfaces which is acting as water inlet is set to inject 100 % water, but the water volume is not the same along the edge. The change of the water volume is controlled by a field function that adjusts the water mass per time unit for each surface.

6.2 Air inlets

Surfaces which are acting as air inlets are set to inject 100 % air with a velocity in m/s. (This value is confidential and will therefore be left out in this version of the report).

6.3 Outlets

Outlet surfaces are set to have a backflow with a volume fraction of 100 % air.

<u>*Save the simulation*</u>

7 Derived parts

Create a plane section located in the middle of the geometry, with the ability to study the mesh. A resulting plane section is shown in Figure 18.

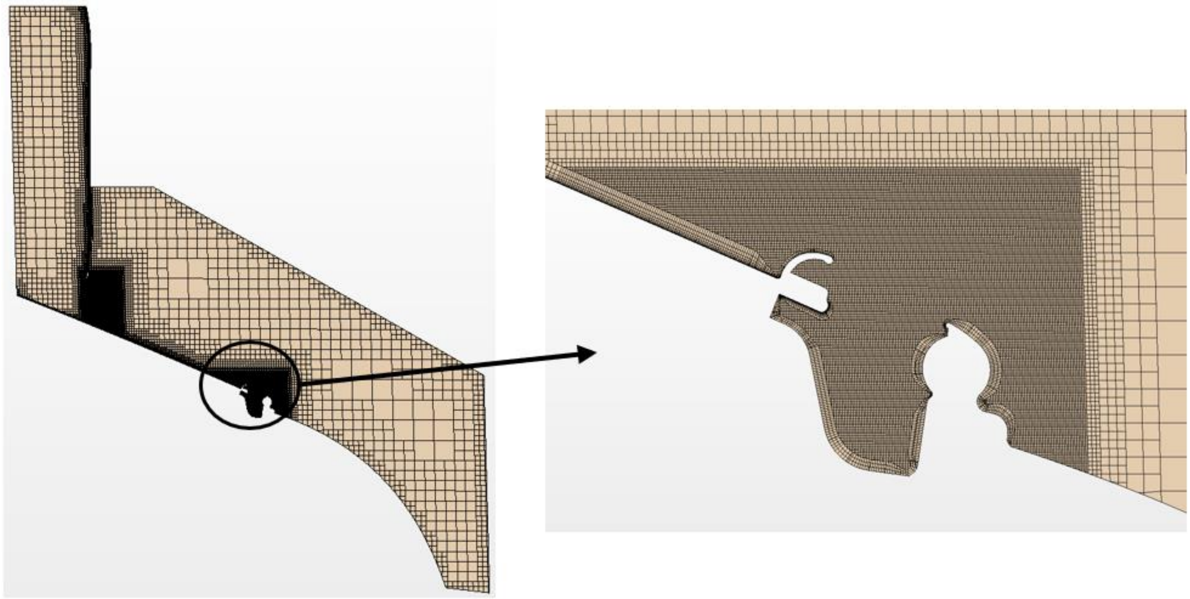


Figure 18 Plane section

Inside the geometry an isosurface is created, which is set to visualize the water during the simulation.

8 Solvers

Default settings are used for the trunk case.

9 Stopping Criteria

The simulation length is set as stopping criteria, settings that differ from default are shown in Table 5.

Table 5 Stopping criteria

Settings	Values
Maximum inner iterations	5
Maximum physical time	30 s
Maximum steps	30000

10 Set up for scenes

Depending on what result is to be studied, different views are selected. Views that are recommended in this case are listed in Table 6.

Table 6 Suggested view

Geometry scene
Mesh scene
Scalar scene

In the simulation the volume fraction of water has been set to 0.5 as minimum and 1 as maximum.

The trigger is set to "Time step" with a frequency of 10 to all scenes that should document the simulation.

Save the simulation

11 Run the simulation

When you reach this point, it is time to start the simulation.

12 Previous result

By using settings as described, previous simulations have generated the result shown in Figure 19 – Figure Figure 20.

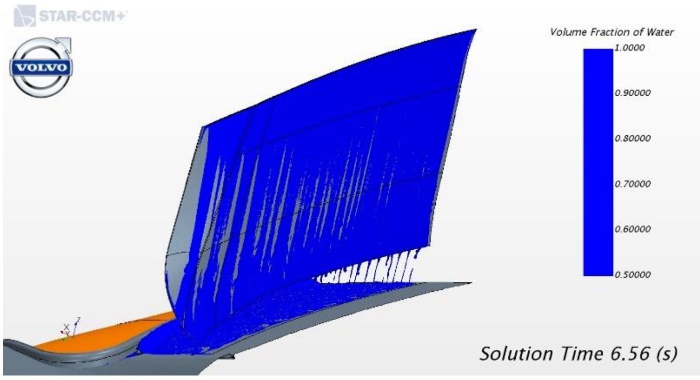


Figure 19 Water flow on trunk

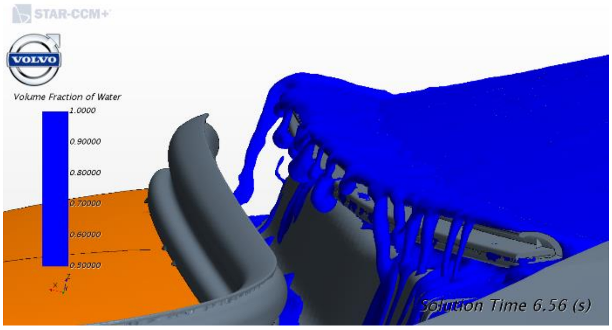


Figure 20 Water flow at the gap

Appendix D

Appendix D contains templates for all cases related to the software PreonLab.

D.1 Template for the front door

Front door simulation using PreonLab

General Information about the process

Project	Model	**Created**	2018-02-26
Needed part	Parts regarding the structure of the front doors.	**STAR-CCM+ version**	11.06.010
Purpose	Study if water is streaming on the latch system inside the door.	**Assumed performing time**	2-3 days
Requirement	It is not allowed to have a continuous water flow streaming in the latch system.	**Previous employers performed the process**	Thesis students, Erik Nilvé and Daniel Persson

Dept/name	Side doors	name	Required software	Team Center CATIA Star-CCM+

Timeline for the process

CAD (25%)	Boundaries (15 %)	Views (10 %)	Running (50 %)

1 CAD preparation inside CATIA

Start by exporting the parts shown in Figure 1 and Figure 2 from Team Center. Insert the parts that make up the door structure.

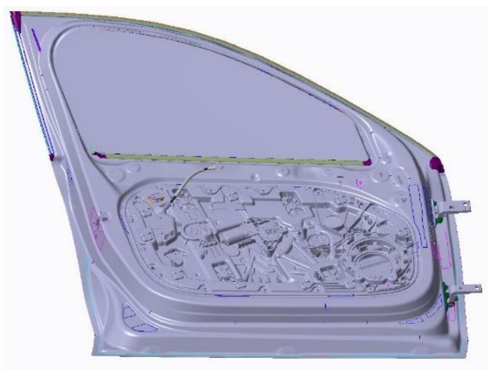


Figure 1 Overview of needed parts, view from inside

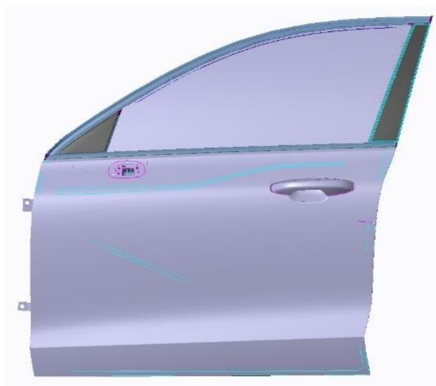


Figure 2 Overview of needed parts, view from outside.

Inside CATIA delete the parts that are intersecting with the window within the area in between the window and the sealing. This is shown in Figure 3.

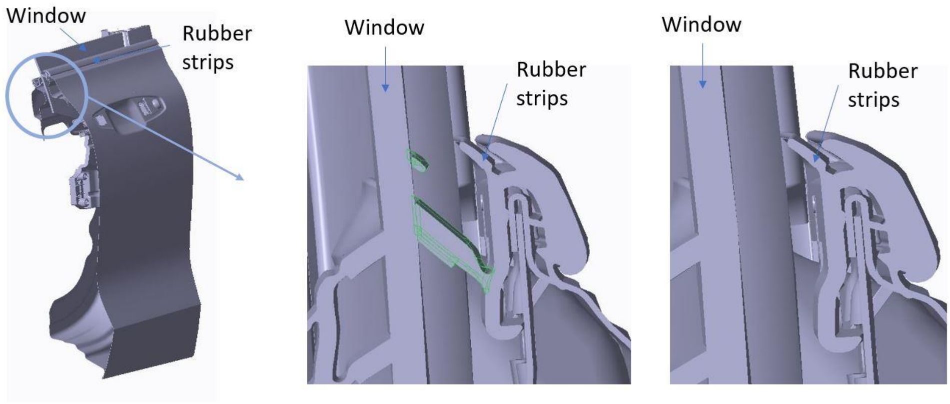


Figure 3 Area in between the window and the sealing

In order to make a realistic sealing in between the sealing and the window a surface is extruded from the sealing towards the window. The surface stops just before it hits the window which creates a narrow gap in between the window and the sealing. The surface is also modified to leave a gap between the surface and the vertical bar dedicated for the main gap. This is shown in Figure 4.

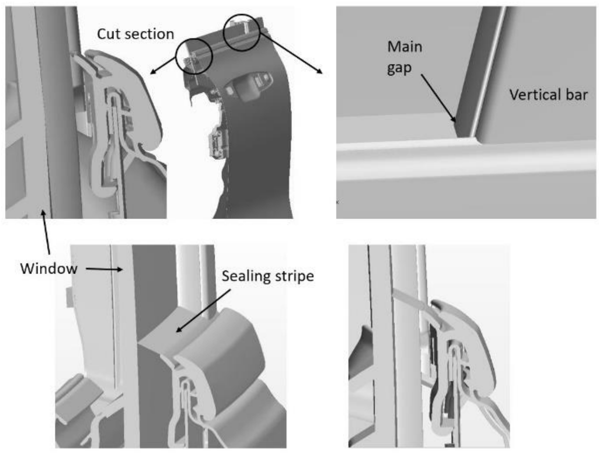


Figure 4 Creating a sealing close to the reality

When saving the geometry, it is advantageous to save parts within different regions as separated since it is not possible to separate or edit the CAD file inside PreonLab. With separated regions of the geometry, it is possible to use different setting on different regions.

Save the file as a STL-file

2 Setup inside PreonLab

2.1 Orient the geometry

First thing to do after importing is to orient and scale each part, so that they are creating a unity together. The scale could be controlled by using the measurement tool inside PreonLab.

2.2 Set the boundary

Next step in the setup is to create outer domain boxes and the inlet. Domain boxes are created only to preserve particles that bring important information to the simulation and delete the particles that are not of interest.

Water inlets are created by square sources located close to the inlets.

Both the water inlet and the outer boxes are shown in Figure 5.

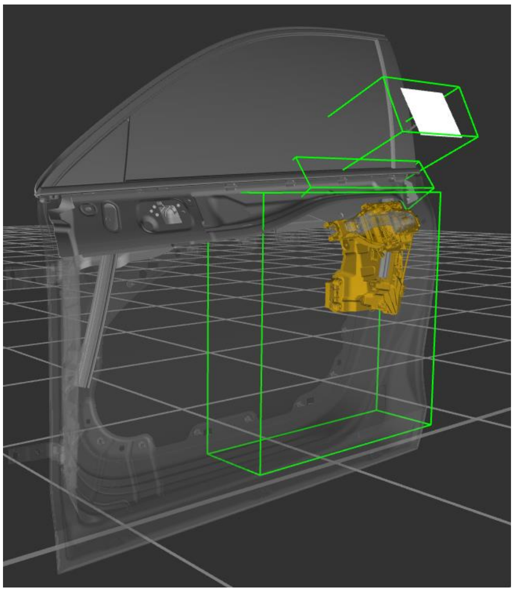


Figure 52.1 Outer domain boxes and water inlet

2.3 Simulation settings

In the simulation the particles size is set to less than half of the size of the gap in order to get a proper flow through the main gap.

To make sure that the water is sticking to the surface, the adhesion is changed.

During the setup the simulation time is also set. This is done by the simulation recorder.

2.4 Set views

View settings can be set according to the users' desire. Features that are recommended to use are functions like opacity which can make parts that are not important transparent, and coloring the geometry with a different color than the particles. Views that are recommended could be shown in chapter 4.

3 Run the simulation

When you reach this point, it is time to start the simulation.

4 Previous result

By using settings as described above, previous simulations have generated the result shown in Figure 6– Figure 9.

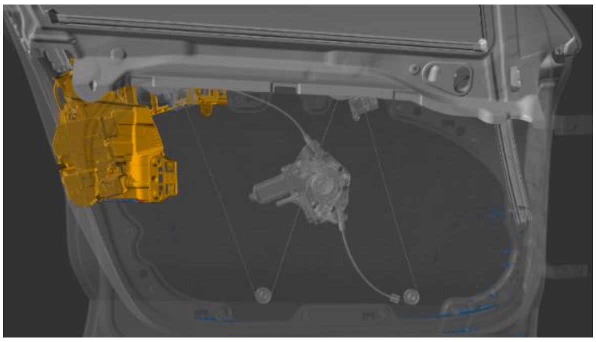


Figure 6 Overview, back view

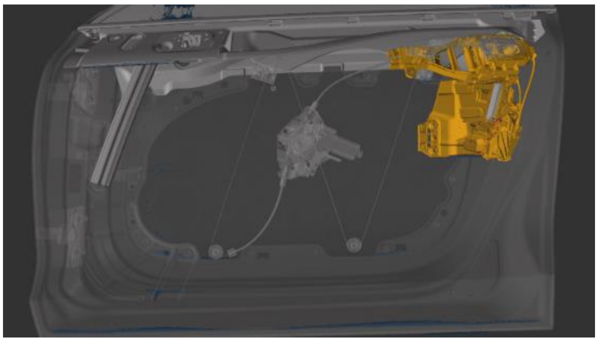


Figure 7 Overview, front view

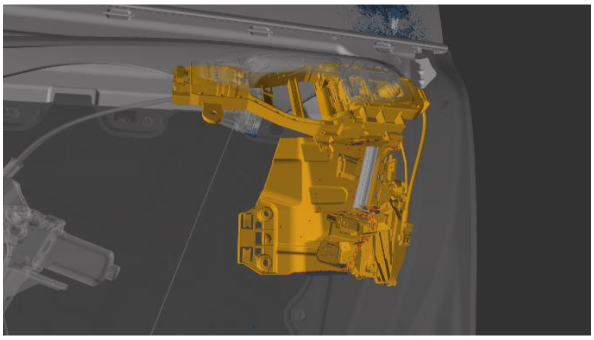


Figure 8 Latch zoom, front view

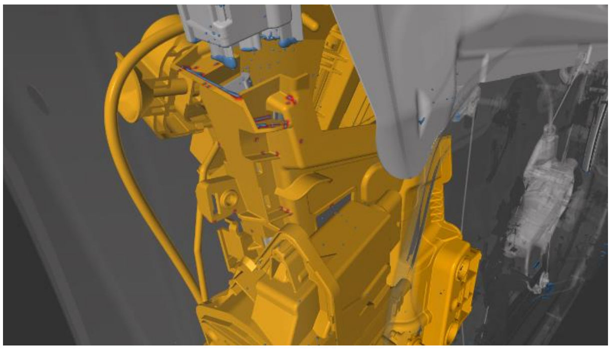


Figure 9 Latch zoom, back view

D.2 Template for the rear door

Rear door simulation using PreonLab

General Information about the process

Project	model		**Created**	2018-02-26
Needed part	Parts regarding the structure of the rear doors.		**STAR-CCM+ version**	11.06.010
Purpose	Study if water is streaming on the latch system inside the door.		**Assumed performing time**	2 - 3 days
Requirement	It is not allowed to have a continuous water flow streaming in the latch system.		**Previous employers performed the process**	Thesis students, Erik Nilvé and Daniel Persson
Dept/name	Side doors	name	**Required software**	Team Center CATIA Star-CCM+

Time line for the process

CAD (25%)	Boundaries (15 %)	Views (10 %)	Running (50 %)

1 CAD preparation inside CATIA

Start by exporting the parts shown in Figure 1 and Figure 2 from Team Center. Insert the parts that make up the door structure.

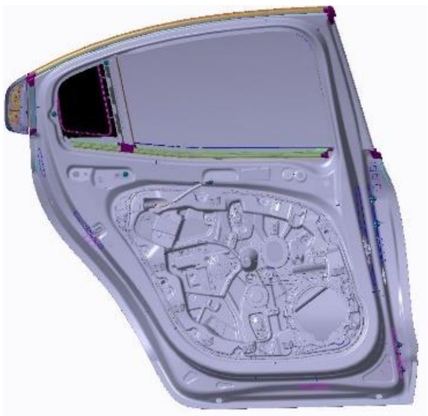


Figure 1 Overview of needed parts, view from inside

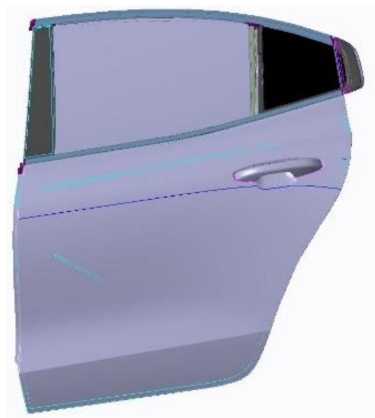


Figure 2 Overview of needed parts, view from outside.

Inside CATIA delete the parts that are intersecting with the window within the area in between the window and the sealing. This is shown in Figure 3.

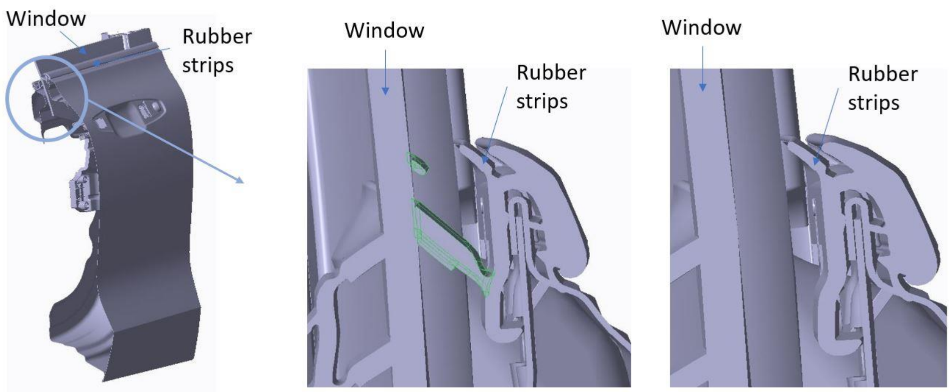


Figure 3 Area in between the window and the sealing

In order to make a realistic sealing in between the sealing and the window a surface is extruded from the sealing towards the window. The surface stops just before it hits the window which creates a narrow gap in between the window and the sealing. The surface is also modified to leave a gap in between the surface and the vertical bar dedicated for the main gap. This is shown in Figure 4.

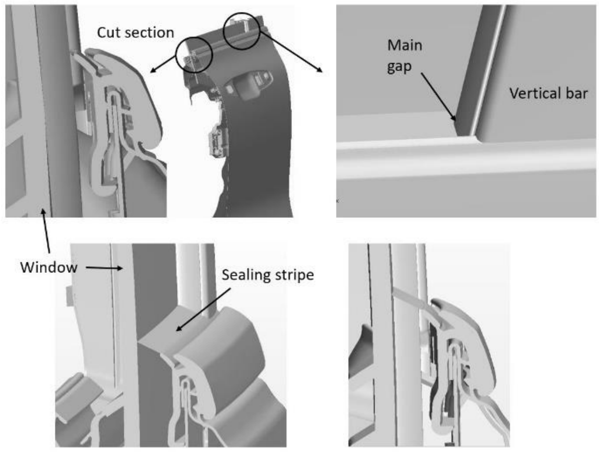


Figure 4 Creating a sealing close to the reality

When saving the geometry, it is advantageous to save parts within different regions as separate files since it is not possibly to separate or edit the CAD file inside PreonLab. With separated regions of the geometry, it is possible to use different setting on different regions.

<u>Save the file as a STL-file</u>

2 Setup inside PreonLab

2.1 Orient the geometry

First thing to do after importing is to orient and scale each part, so that they are creating a unity together. The scale could be controlled by using the measurement tool inside PreonLab.

2.2 Set the boundary

Next step in the setup is to create outer domain boxes and the inlet. Domain boxes are created only to preserve particles that bring important information to the simulation and delete the ones that are not of interest.

Water inlets are created by square sources located close to the inlets.

Both the water inlet and the outer boxes is shown in Figure 5

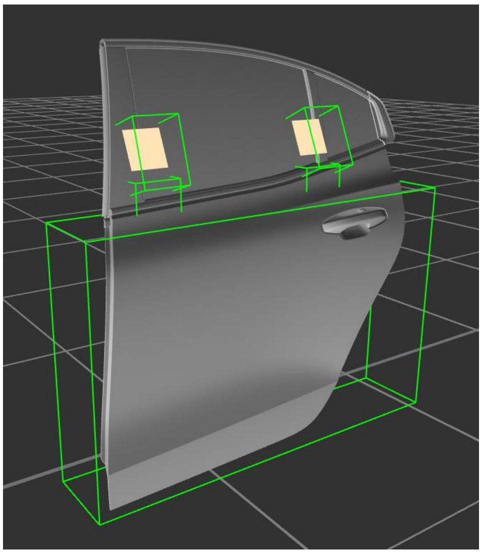


Figure 5 Outer domain boxes and water inlet

2.3 Simulation settings

In the simulation the particle size is set to less than half the size of the gap in order to get a proper flow through the main gaps.

To make that the water is sticking to the surface a little more, the adhesion is changed in the simulation.

During the setup the simulation time is also set, this is done by the simulation recorder.

2.4 Set views

View settings are possible to set according to the users' desire. Features that are recommended to use are functions as opacity which can make parts transparent, and coloring the geometry with a different color than the particles. Views that are recommended could be shown in chapter 4.

3 Run the simulation

When you reach this point, it is time to start the simulation.

4 Previous result

By using settings as described above, previous simulations have generated the result shown in Figure 6- Figure 9.

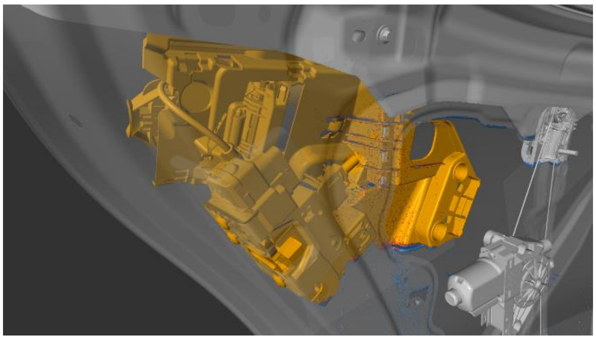


Figure 6 Latch zoom, back view

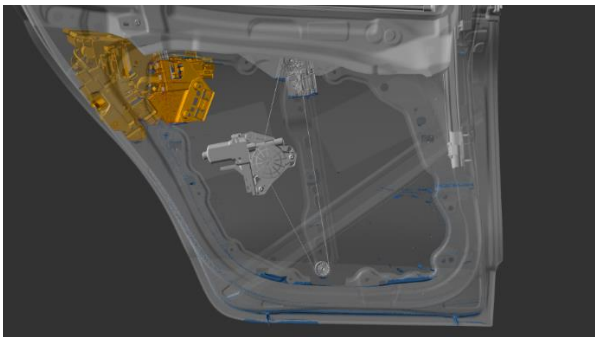


Figure 7 Overview, back view

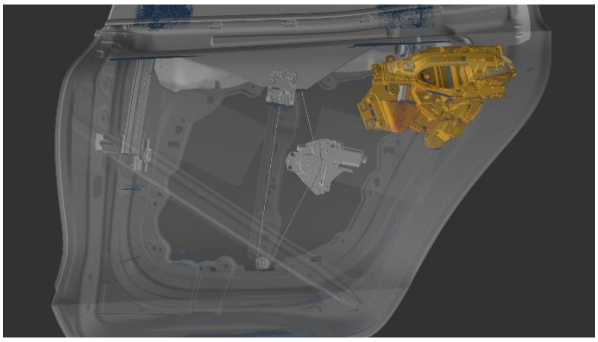


Figure 8 Overview, front view

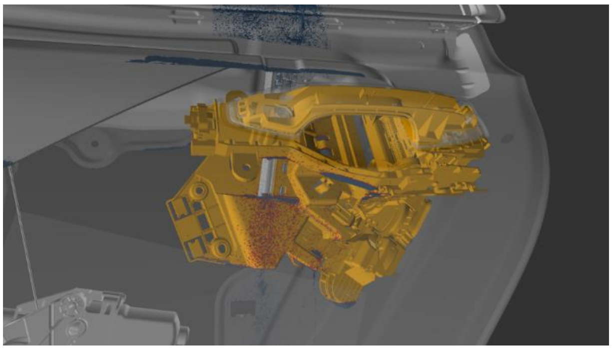


Figure 9 Latch zoom, front view

D.3 Template for the pipe

Drainage pipe simulation using PreonLab

General Information about the process

Project	model		**Created**	2018-02-26
Needed part	Drainage pipes with connections on both sides.		**STAR-CCM+ version**	11.06.010
Purpose	Study the pipes capacity and how the water is behaving inside the pipes.		**Assumed performing time**	4-5 days
Requirement	Each pipe should handle a volume of water per time unit.		**Previous employers performed the process**	Thesis students, Erik Nilvé and Daniel Persson
Dept/name	Side doors	name	**Required software**	TeamCenter CATIA Star-CCM+

Timeline for the process

CAD (20%)	Boundaries (5 %)	Views (5 %)	Running (70 %)

1 CAD preparation inside CATIA

Start by exporting the parts shown in Figure 1 from Team Center. All parts that are connected to the pipe need to be inserted.

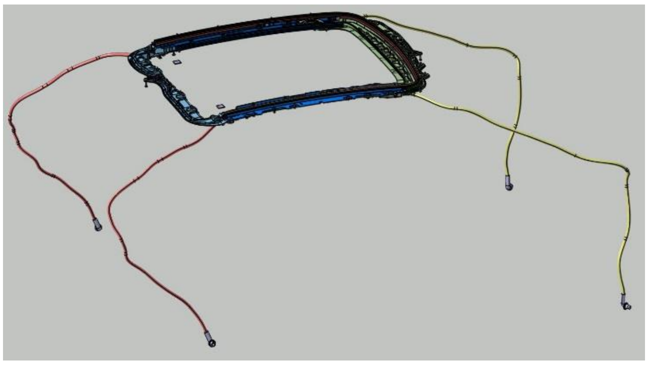


Figure 1 Overview of needed parts, view from inside

Trim the geometry that is imported from Team Center by deleting parts that are unnecessary for the simulation. The result of a trimmed geometry should look like Figure 2.

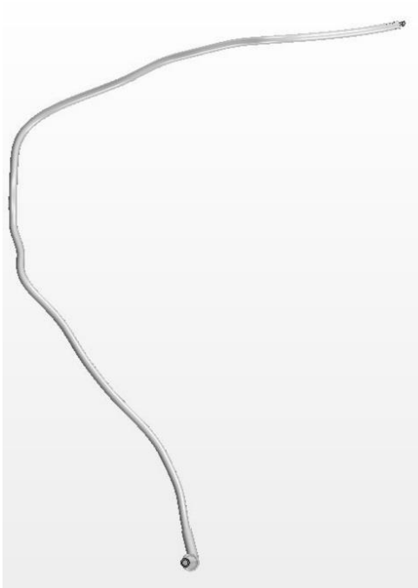


Figure 2

A closer view of the ends is illustrated in Figure 3 and Figure 4. Cut the inlet pipe with the same angle as the water collector had before.

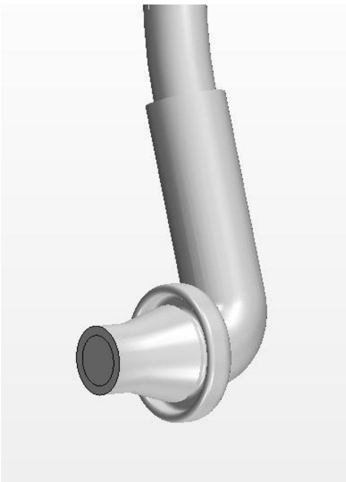


Figure 3 Closer view of the outlet

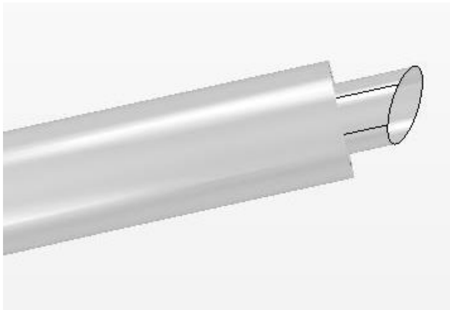


Figure 4 Closer view of the inlet

On the inlet, a box should be built with the same volume as the requirement. The box is constructed with an inclination towards the pipe inlet, to make sure that the water is streaming into the pipe. In Figure 6 an overview of the box is illustrated.

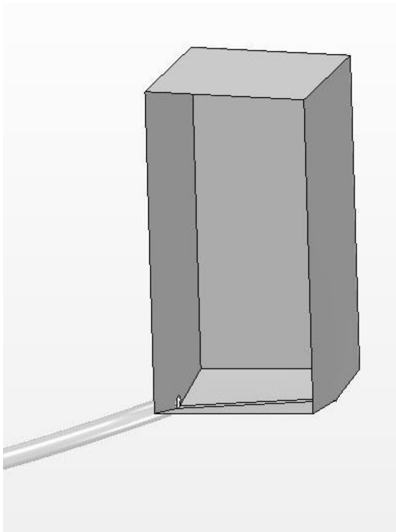


Figure 6 Overview of the box

The only area that needs to be checked in order to have a closed volume is the connection between the pipe and box pipe, since the ends are already closed. By extruding surfaces between the pipe and the box pipe the volume is closed. A closer view of the connection is shown in Figure 7.

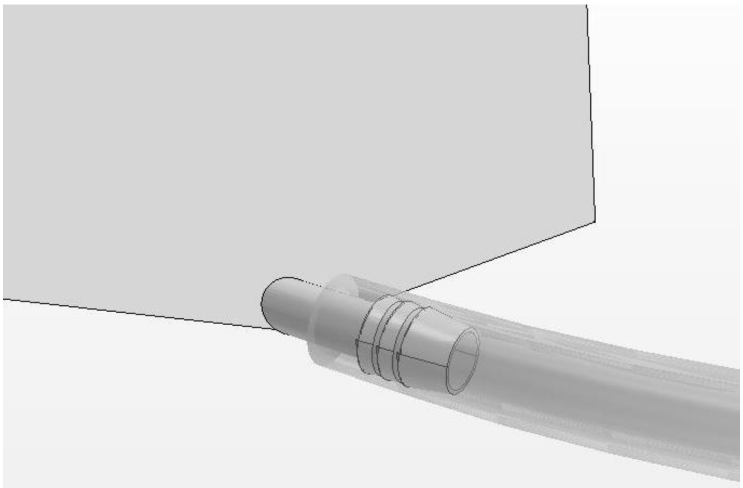


Figure 7 Closer view of the connection between the box and the pipe

Save the file as a STL-file

2 Setup inside PreonLab

2.1 Orient the geometry

The first thing to do after importing is to orient and scale each part, so that they are creating a unity together. The measurement tool inside PreonLab can be used when controlling the scale of the geometry inside PreonLab.

2.2 Set the boundary

Next step in the setup is to create outer domain boxes and the inlet. Domain boxes are created to only preserve particles that bring important information to the simulation and delete the ones that are not of interest.

Water inlets are created by square sources located close to the inlets.

Both the water inlet and the outer boxes is shown in Figure 5.

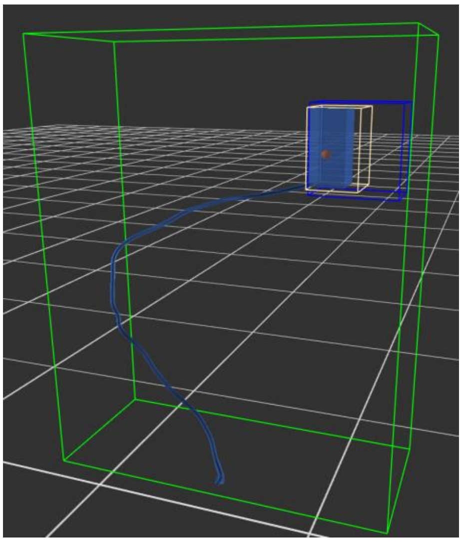


Figure 5 Outer domain boxes and water inlet

2.3 Simulation settings

In the simulation the particle size set to less than half the size of the tube order to get a proper flow through the main gap.

To make the water stick to the surface more than before, the adhesion is changed.

During the setup the simulation time is also set. This is done by the simulation recorder.

2.4 View settings

View settings can be set according to the users' desire. Features that are recommended to use are functions such as opacity which can make parts transparent, and coloring the geometry with a different color of the particles. Views that are recommended could be shown in chapter 4.

3 Run the simulation

When you reach this point, it is time to start the simulation.

4 Previous result

By using settings as described above, previous simulations have generated the result shown in Figure 6 and Figure 8.

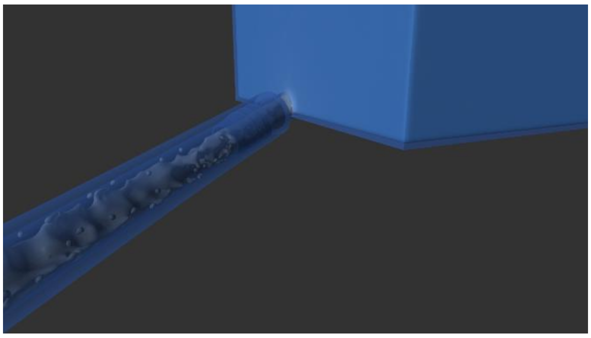


Figure 6 Connections between pipe and box

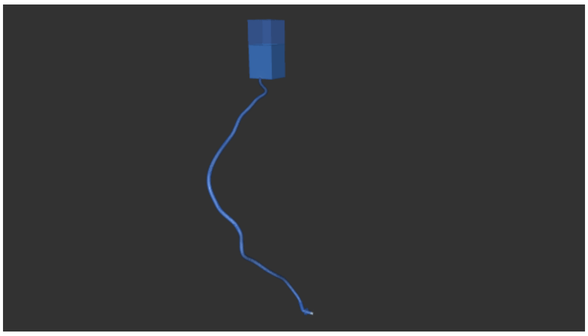


Figure 7 Overview

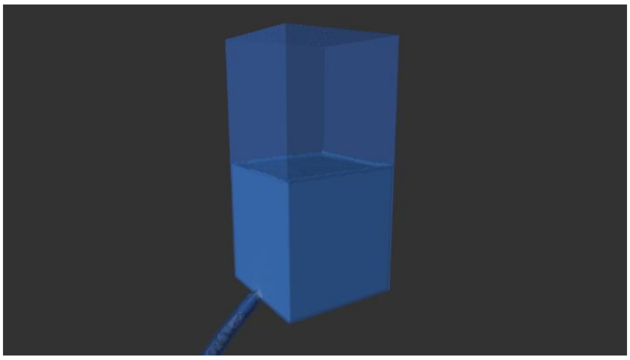


Figure 8 Tank during simulation

D.4 Template for the trunk

Trunk simulation using PreonLab

General Information about the process

Project	model		**Created**	2018-02-26
Needed part	Exterior parts of the rear section of the car.		**STAR-CCM+ version**	11.06.010
Purpose	Study if the water is entering the luggage area or not.		**Assumed performing time**	1-2 days
Requirement	No water flow is allowed to enter the luggage area.		**Previous employers performed the process**	Thesis students, Erik Nilvé and Daniel Persson
Dept/name	Side doors	name	**Required software**	TeamCenter CATIA Star-CCM+

Time line for the process

CAD (10%)	Boundaries (20 %)	Scenes (20 %)	Running (50 %)

1 CAD preparation

Start by exporting the parts shown in Figure 1 from Team Center. The focus is to include all the exterior surfaces where it is possible that water might be streaming.

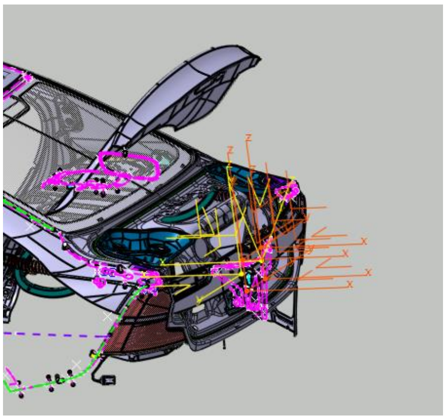


Figure 1, Parts from Team Center

Inside CATIA, save parts from the geometry as separate files. This will have a specific setting inside PreonLab. For the trunk case the trunk with associated parts is set as one region and the inner part of the luggage area will create one region.

Save the file as a STL-file

2 Setup inside PreonLab

2.1 Orient the geometry

First thing to do after importing is to orient and scale each part, so that they are creating a unity together. The measurement tool inside PreonLab can be used when controlling the scale of the geometry inside PreonLab.

2.2 Set the boundary

Next step in the setup is to create outer domain boxes and the inlet. Domain boxes are created to preserve only the particles that bring important information to the simulation and delete those that are not of interest.

Water inlets are created by square sources located close to the inlets.

Both the water inlet and the outer boxes are shown in Figure 2.

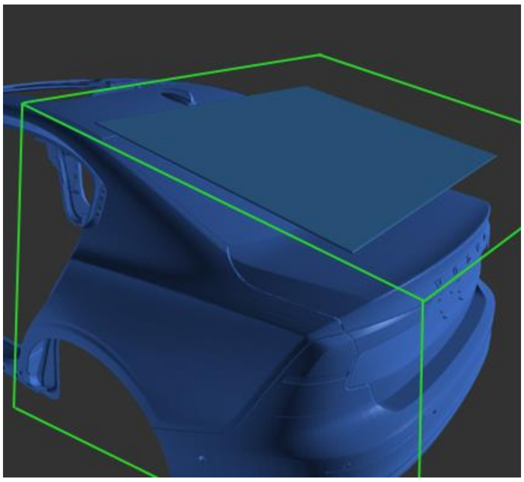


Figure 2 Outer domain boxes and water inlet

2.3 Simulation settings

Due to exterior water flow the size of the particle is not that important compared to interior water flow. There is no requirement that the water should be able to flow through a gap.

2.4 View settings

View settings can be set according to the users' desire. Features that are recommended to use are functions as opacity which can make parts transparent, and coloring the geometry with a different color than the particles. Views that are recommended could be shown in chapter 4.

3 Run the simulation

When you reach this point, it is time to start the simulation.

4 Previous result

By using settings as described, previous simulations have generated the result shown in Figure 3-Figure 6.

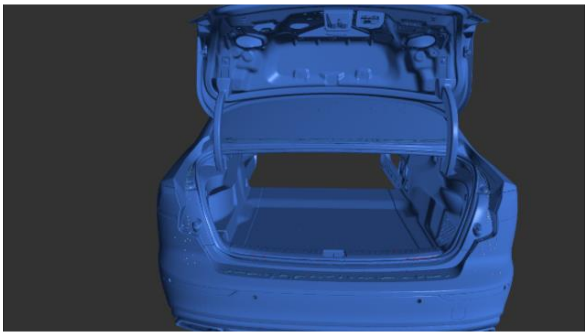


Figure 3 Open trunk, back view

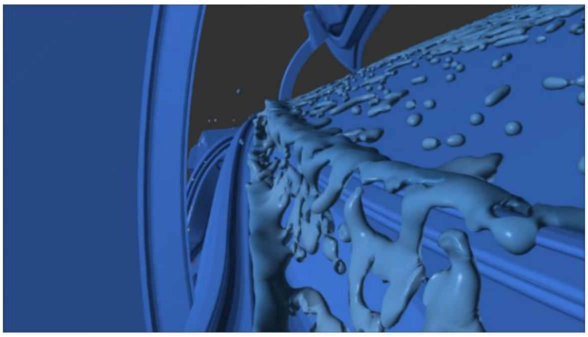


Figure 4 Gap area

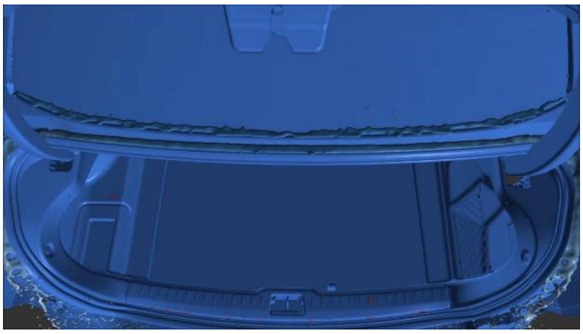


Figure 5 Open trunk, top view

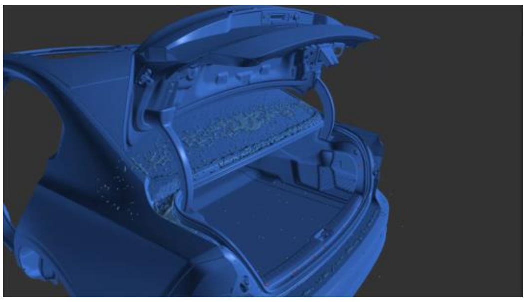


Figure 6 Open trunk, side view