A Sensitivity Study of Some Numerical and Geometrical Parameters Affecting Lift

Volvo Car Corporation

Petter Ekman

LIU-IEI-TEK-A--14/02020—SE

Master thesis
Department of Management and Engineering
Linköping University, Sweden

Linköping, June 2014
A sensitivity study on the numerical and geometrical parameters affecting lift

Volvo Car Corporation

Petter Ekman

Supervisor: Professor Simone Sebben
Volvo Car Corporation, Göteborg

Ph.D. Roland Gårdhagen
IEI, Linköpings Universitet

Examiner: Professor Matts Karlsson
IEI, Linköpings Universitet
Linköping, June 2014
Abstract
Volvo Car Corporation (VCC) uses Computational Fluid Dynamics (CFD) and wind tunnel during the aerodynamic development of new vehicles. In the past VCC main focus has been on the drag force correlation to the wind tunnel measurements but in recent years improved methods for lift force correlations has been highly wanted. Three objectives were considered in this study to improve the lift force correlation between the CFD simulations and wind tunnel measurements for geometrical configurations of the V60 and S60 models.

Poor mesh resolution for the wall bounded flow existed for the VCC mesh method and therefore prisms layers were considered in this thesis to increase the mesh resolution inside the boundary layer.

As slick tyres generally were used in the CFD simulations better geometrical correlation was wanted to be studied as it could improve the lift force correlation between CFD simulations and wind tunnel measurements. Therefore detailed tyres were considered in this study.

As the coarsest surface mesh size was used for the underbody and the components inside the engine bay, where some of the highest flow velocities occurred, mesh refinements were investigated for engine bay and underbody in this study.

The prisms layers improved the predicted behavior for the boundary layer as it captured the large velocity gradients more accurately. Due to this, the skin friction prediction was also improved. Different flow behavior around the front wheels and rear wake occurred due to earlier separation. The different flow field caused an improved correlation for the lift force but worsened correlation for the drag force due to increased pressure at the rear of the cars. However, the front lift force trend correlation for the considered configurations was improved with the prisms layer mesh method.

The detailed tyres caused slight more disturbances for the underbody flow which caused more attached flow around the rear of the car hence lowered pressure. Earlier separation around the front wheels also occurred for the detailed tyre geometry as the disturbed flow around the wheels was increased. Slight improved correlation for the front and rear lift forces to the wind tunnel measurements could be seen with the detailed tyre compared to the slick tyre.

The mesh refinements for the engine bay and underbody showed significant differences for the flow at the underbody which had significant impact on the flow at the rear wake for the V60 model. Minor differences could be seen for the aerodynamic forces for the baseline configuration for the V60 model while great differences occurred for the configurations affecting the underbody. Due to this significant improved correlation for the front and rear lift force trends were achieved for the underbody configurations with the refined engine bay and underbody mesh method.

Conclusions could be drawn that the prisms layer caused earlier separation due to its increased mesh resolution for the wall bounded flow. However, finer mesh resolution was needed inside the boundary layer to ensure consistent separation behavior for both the considered models. Improved correlation for the front lift force could however be seen. The detailed tyre only had minor effects on the flow field and aerodynamic forces and therefore not so important to include for further studies. The refined engine bay and underbody caused significant improved lift force trend correlation to the wind tunnel measurements and should be considered for future studies. To improve the correlation between CFD simulations and wind tunnel measurements increased mesh resolution for the wall bounded flow should be considered to better capture the large velocity gradients close to the wall.

Keywords: CFD, Mesh, Boundary layer, Prisms layer, Harpoon, Ansys Fluent, Vehicle aerodynamics.
Preface
In this thesis a sensitivity study of some numerical and geometrical parameters affecting the aerodynamic forces has been performed to improve the general correlation between CFD simulations and wind tunnel measurements. The thesis was performed at the aerodynamics group at Volvo Car Corporation in Torslanda, Sweden.

Acknowledgements
I want to thank all the people at the aerodynamics department for their kindness and helpfulness with this thesis.

I especially want to thank my supervisor at VCC professor Simone Sebben for the opportunity, support and help to perform my master thesis at VCC. Also a special thanks to thank Mattias Hejdesten and PhD student Lennert Sterken for help and support with software but also for interesting discussion which helped to achieve the results of this thesis.

Many thanks to my supervisor at Linköping University Roland Gårdhagen and opponent Anton Jonzén for important and good feedback on the report and presentation performed at Linköping University.

Göteborg June 2014
Petter Ekman
Nomenclature

Greek letters

\( \Delta t \) Time step  
\( \Delta C_D \) Drag force coefficient difference  
\( \Delta C_Lf \) Front lift force coefficient difference  
\( \Delta C_{Lr} \) Rear lift force coefficient difference  
\( \delta \) Boundary layer thickness  
\( \varepsilon \) Turbulent dissipation rate  
\( \mu \) Viscosity  
\( \mu_t \) Turbulent viscosity  
\( \rho \) Density  
\( \rho_\infty \) Free stream density  
\( \tau_w \) Wall shear

Latin letters

\( A_{ref} \) Reference area  
\( C_D \) Drag force coefficient  
\( C_L \) Lift force coefficient  
\( C_{Lf} \) Front lift force coefficient  
\( C_{Lr} \) Rear lift force coefficient  
\( k \) Turbulence kinetic energy  
\( l \) Characteristic length  
\( M \) Mach number  
\( Re_{crit} \) Critical Reynolds number  
\( t \) Time  
\( U^* \) Dimensionless velocity  
\( u_t \) Friction velocity  
\( u^+ \) Dimensionless velocity  
\( v \) Velocity  
\( v_\infty \) Free stream velocity  
\( X_i \) Volume force  
\( y \) Wall distance  
\( y^+ \) Dimensionless wall distance  
\( y^* \) Dimensionless wall distance

Abbreviations

CAD Computer Aided Design  
VCC Volvo Car Corporation  
CFD Computational Fluid Dynamics  
RANS Reynolds Average Navies Stokes  
PID Property Identification  
FMG Full Multi Grid  
CPU Central Processing Unit  
MRF Moving Reference Frame
Contents

1. Introduction ........................................................................................................................................... 1
   1.1 Objective ........................................................................................................................................ 2
      1.1.1 Implementation of prisms layers ......................................................................................... 2
      1.1.2 Effects of detailed tyres ...................................................................................................... 3
      1.1.3 Effects on aerodynamic forces of mesh refinement for engine bay and underbody .......... 3
   1.2 Limitations ....................................................................................................................................... 3

2. Background .......................................................................................................................................... 5
   2.1 Geometry description ..................................................................................................................... 5
      2.1.1 Geometrical differences between CAD and wind tunnel models ..................................... 6
      2.1.2 Considered configurations of the models .............................................................................. 6
   2.2 Description of VCC wind tunnel ................................................................................................... 7

3. Theory .................................................................................................................................................. 11
   3.1 Fundamental aerodynamics ........................................................................................................... 11
   3.2 Road vehicle aerodynamics ........................................................................................................... 13
      3.2.1 Aerodynamic coefficients ..................................................................................................... 14
   3.3 Governing equations ....................................................................................................................... 14
      3.3.1 Turbulence ............................................................................................................................. 14
      3.3.2 Reynolds Average Navier-Stokes ......................................................................................... 15
      3.3.3 Turbulence modelling ............................................................................................................. 15
      3.3.4 Wall treatments ....................................................................................................................... 17
   3.4 Numerical methods ......................................................................................................................... 18
      3.4.1 Solver scheme ......................................................................................................................... 18
      3.4.2 Numerical discretization ....................................................................................................... 18
   3.5 Numerical mesh elements techniques ............................................................................................ 18

4. Method ................................................................................................................................................. 20
   4.1 Numerical set-up ............................................................................................................................ 20
   4.2 Computational mesh ....................................................................................................................... 22
   4.3 Implementation of prisms layers .................................................................................................... 24
      4.3.1 Geometrical work .................................................................................................................. 26
4.3.2 Prisms layer settings .................................................................................................. 28
4.3.3 General mesh method and geometrical dependency ........................................... 33
4.4 Effects of detailed tyres ............................................................................................ 35
  4.4.1 Special consideration for the groove plateau ....................................................... 39
4.5 Mesh refinement for the engine bay and underbody ................................................ 39
5. Results and Discussion ............................................................................................... 42
  5.1 Prisms layer generation ............................................................................................ 42
    5.1.1 V60 ..................................................................................................................... 42
    5.1.2 S60 ..................................................................................................................... 49
    5.1.3 Force comparison between the mesh methods and wind tunnel measurements ... 54
    5.1.4 General discussion about the prisms layer mesh method .................................. 60
  5.2 Effects of detailed tyres ............................................................................................ 61
    5.2.1 General discussion about the detailed tyres ........................................................ 68
  5.3 Detailed tyres and prisms layer mesh ..................................................................... 69
  5.4 Mesh refinement for engine bay and underbody .................................................... 71
    5.4.1 General discussion about the refined engine bay and underbody mesh method ... 77
  5.5 Effects on processing and computational cost ......................................................... 78
6. Conclusions .................................................................................................................. 79
7. Future work .................................................................................................................. 81
References ......................................................................................................................... 82
Appendix A ....................................................................................................................... 83
  A.1 Geometry problems .................................................................................................. 83
  A.2 Additional figures for the configurations ................................................................. 83
  A.3 Harpoon version 5.4beta17 .................................................................................... 84
  A.4 Additional simulations ............................................................................................. 85
  A.5 Surface thickness creation procedure ..................................................................... 86
  A.6 Updated tyre morphing procedure ......................................................................... 89
  A.7 Harpoon script for prisms layer generation ............................................................ 99
  A.8 Removal of negative volumes in Ansys Fluent ....................................................... 103
  A.9 PID set up for the refined engine bay and underbody method ............................... 104
1. Introduction
Volvo Car Corporation (VCC) uses Computational Fluid Dynamics (CFD) and wind tunnel measurements during the aerodynamic development of new vehicles. For many years VCC’s main focus in the aerodynamic research and development has been on the drag force which have resulted in better prediction for the drag force. However, in recent years the demand of accurate results from the CFD simulations for all aerodynamic forces on the vehicles have become more important to ensure better correlation between simulation and experimental results for improved possibilities of virtual aerodynamic development. Upcoming EU (European Union) rules about CO2 emissions requires that VCC in earlier stages of a project can assure that the results from the CFD simulations are valid in order to meet the requirements for the emissions and fuel consumption.

During the design process of a new vehicle a lot of simulations and configuration works are made in CFD where the selected model is validated with a clay model in the wind tunnel for some configurations before a real car is produced and tested in the wind tunnel. VCC usually get good correlation between the CFD simulations and wind tunnel for the drag force. However, the front and rear lift forces often has poor correlation to the wind tunnel and put a lot of uncertainties to the results and also lowers the engineer’s confidence for the CFD results. Poor correlation for the force trends also hampers the virtual development of the cars and thereby increases the work effort in the wind tunnel. These poor correlations for the front and rear lift forces produce a lot of uncertainties in the results as it indicates that different pressure distribution and hence different flow fields are achieved in the simulations compared to the wind tunnel measurements.

There exist many possible reasons for the poor correlation between the CFD simulations and wind tunnel experiments. One problem is that the vehicle geometry not always is identical between the car measured in the wind tunnel and the CFD simulated car. This is especially true for the underbody where aerodynamic forces acts on the vehicle during the experimental measurements and deforms the wind tunnel models geometry which influences the results. These effects are very hard to reproduce for the CFD simulations as it would require more complex simulations methods which would be more time consuming than it is today. The real geometry can however be achieved by scanning the real wind tunnel model but can only be made for existing models which sometimes may be too late for drastically design changes. Differences in the measurements can also occur in the wind tunnel as for example the pitch of the car affects the lift force.

For the CFD simulation simplified tyre geometries are used to decrease the computational costs. These tyres are however, modified into the same shape (due to the loading of the vehicle and the rotational forces) as in the wind tunnel in order to capture the effects of the contact patch, increased radius and wider bulge of the tyre. The VCC wind tunnel is also not represented in the CFD simulations as simulations are performed to replicate road like conditions. Different flow behavior to CFD domain has been seen in the wind tunnel [1] as the geometry in the wind tunnel causes asymmetrical flow behavior around the models.

However, probably the main reason for the poor correlation is the turbulence modelling in CFD. VCC uses today a two equation Reynolds Average Navier-Stokes (RANS) model which has its limitations as the turbulent flow is modeled instead of resolved. It can therefore be hard to capture all the effects of the turbulent flow from a modeled behavior as for example, complex flow features occur when the turbulent flow travels through engine bay and the underbody of a vehicle. It is also known from earlier studies [2] and [3] that two-equation RANS turbulence models have problems to model correct behavior of turbulent flow around geometries similar to vehicles.
The mesh and its quality can also play a major role in order to achieve better accuracy from the CFD simulations. In order to model the behavior of the boundary layer as accurate as possible many cells in the mesh needs to cover it. Today a hexahedral dominant mesh is used which not covers the boundary layer with several cells but instead trust is put on the wall function treatment to model the behavior of the boundary layer to an acceptable level. However, creating a mesh which covers the boundary layer is demanding, especially for complex geometries which are the case for VCC. The detail of the geometries has also increased the recent years while the demand on faster results have led to need of not only faster CFD simulations but also need of faster CAD assembly and mesh generations where as little manual input as possible would be needed. Therefore is the meshing technique used today simplified so it can be created by use of scripts for all types of vehicle geometries and configurations.

VCC is keen to believe that better lift force correlation to the wind tunnel measurements can be achieved with CFD simulation with an updated CFD procedure.

1.1 Objectives
VCC has with the current CFD procedure problem to achieve good correlation between CFD and the wind tunnel for their lift forces, especially for the rear lift force which has important effects on the high speed handling. This makes it hard for the engineers to ensure confidence to meet the lift requirements set by the chassis dynamic department. As it is too costly for VCC to increase the mesh size much and switch to more advanced turbulence models it would be desirable to achieve better results with slight modifications of their current simulations method. Due to the CFD simulation method is used on a large variety of cars it is important that the method is robust and not take much longer time than the current simulation method in order to keep the processing time as low as possible.

Earlier study [4] had been performed where CFD simulations according to the current CFD procedure had been compared to wind tunnel measurements. In order to see if better correlation could be obtained the results of the new developed methods was to be compared with these results.

1.1.1 Implementation of prisms layers
A hexahedral dominant mesh created in Harpoon is used for the CFD simulations. Harpoon have in previous versions not provided the possibilities to generate cells to specific cover the boundary layer of the flow field but in the newly released version the possibilities exist.

As no specific cells were used to cover the boundary layer in the current meshing method the number of cells covering the boundary layer was heavily affected by the set surface mesh size. The surface mesh size was set after the geometrical shape in order to achieve a good geometry representation but where complex flow files where expected. For example finer surface mesh was used for the rear spoiler as separation at its rear edge was expected.

Due to the first node height was affected by the surface mesh size the first node height for the cells varied over the surfaces which in turn generated a varying y+. Varying y+ would not been a problem if the variations of the first node height were small. However, the surface mesh size could vary from 1.25 mm to 5 mm which resulted in large y+ variations. This resulted in unphysical flow behavior and thereby poorer correlation to the wind tunnel measurements. As VCC used the realizable $k − \epsilon$ RANS turbulence model with the standard wall function the y+ should be in the region of 30 to values of hundreds which corresponded to the log-law region, based on the Reynolds number for a car [5]. To achieve better control of y+ and better mesh resolution for the boundary layer implementation of prisms layers into the current meshing technique was wanted in this thesis.
Implementing prisms layer generation into the meshing method was believed to improve the modelling of the boundary layer. Due to this, improved correlation to the wind tunnel measurements and better captured flow fields was to be expected. Even though fully correct values may not be achieved with the prisms layers hopes were set to force trends for configurations would correlate better which would improve the development of cars. There was also a future need of the prisms layer implementation as VCC in the future wanted to switch to more advanced turbulence models which needed increased resolution of the wall bounded flow.

Earlier study [6] showed promising results but was only made for vehicles without engine bay due problems caused when meshing. Therefore a robust and geometry insensitive meshing technique was wanted.

1.1.2 Effects of detailed tyres
VCC used slick tyres on their CFD models in the simulations. Earlier studies [7], [8] and [9] had shown the importance of simulating the correct shape and detail level of the tyres as large effects on the aerodynamic forces have been seen. Due to this, one objective in this thesis was to see if better correlation to the wind tunnel could be achieved with simulations with detailed tyres corresponding to the same geometry and similar shape as the tyres used on the wind tunnel models.

1.1.3 Effects on aerodynamic forces of mesh refinement for engine bay and underbody
Seen in many studies [10], [11] and [12] the flow through the engine bay and at the underbody has large effects on the aerodynamic forces. VCC used the largest surface mesh size at the underbody where also the highest flow velocities were obtained for the considered geometries due to the ground effect. Due to this poor resolution of the wall bounded flow and geometry was obtained. Complex geometry and thereby complex flow behavior occurred at the engine bay and underbody. Different mesh approach for the engine bay and underbody was therefore investigated in order to see if improved capturing of the flow field could be obtained and thereby improved correlation between CFD and wind tunnel measurements.

1.2 Limitations
- VCC wanted to keep the pre-processing and simulation costs as low as possible and thereby were only steady state simulations considered in this thesis even though flow behavior for passenger cars are known for its unsteady behavior [2], [3] and [10].

- In order to keep the computational costs small too large mesh sizes could not be used which meant that a full mesh independency study could not be considered. However, VCC’s meshing procedure have been updated and evaluated over time and thereby the results should be reasonably mesh independent.

- Prisms layers for all the external surfaces would be wanted but was not possible due to the complexity of the given CAD geometry. Therefore, were prisms layers mainly considered for the external surfaces of the vehicles as the exterior geometry was simpler and easily possible to simplify for the prisms generation, hence more suitable for prisms layers.

- The mesh was done in Harpoon which is used for mesh generation for external aerodynamics simulations at VCC. As Harpoon generates the prisms layers after the initial mesh was created the control of the number of prisms layers and its quality was limited by the software.

- No accurate measurements existed for the tyre geometry for the wind tunnel models thus no geometrical values existed for the detailed tyres shape. Instead the values were achieved from
[7] for the tyre morphing and the deformation of the grooves were based on assumptions made in [8].

- In the CFD simulations the vehicle geometry was completely solid while in the wind tunnel measurements flexing and small deformation of panels and components may have occurred. These geometrical errors were measured in static conditions in [4] and may have increased during the wind tunnel measurements.

- Errors and limitations will always occur when simulating advanced physics. As only steady state simulations were considered RANS turbulence modelling was used in order to generate a steady state solution for turbulent flow. When turbulent flow is simulated as a steady state the turbulent behavior is modeled instead of resolved as it partly would be in more advanced turbulence models. The modelling of the turbulent flow may cause errors as steady state turbulence models which corresponded well with all types of flow behaviors not existed at the time performing the thesis.

- One of the most limiting factors of this thesis was that no flow visualization figures from the wind tunnel experiments existed (except some wake plots), making it hard to understand if the flow behavior correlated well to the flow in the wind tunnel experiments. Therefore more trust on the correlation of the aerodynamic forces had to be made.
2. Background
In this chapter are CAD geometry, VCC current CFD procedure and the software’s used for this thesis presented.

2.1 Geometry description
CFD simulations and wind tunnel experiments performed in the VCC wind tunnel had earlier been done [4] for two Volvo car models, the S60 sedan and the V60 sportswagen seen in Figure 2.1. Both models had the same five cylindrical diesel engines and four wheel drive system. They were identical from the front to the A-pillars. In order to be able to compare the new methods created in this thesis the same models used in [4] were used.

![Figure 2.1. Left: The V60 model used in this thesis. Right: The S60 model used in this thesis. Note that the car models are identical until the A-pillars.](image)

The vehicle dimensions can be seen in Table 2.1.

Table 2.1. The dimensions for the considered models. Note that the S60 was slightly higher due to the shark fin (antenna) was placed more forward on the roof than on the V60 which increased the height.

<table>
<thead>
<tr>
<th>Car model</th>
<th>Length [mm]</th>
<th>Width [mm]</th>
<th>Height [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>V60</td>
<td>4633</td>
<td>1866</td>
<td>1449</td>
</tr>
<tr>
<td>S60</td>
<td>4633</td>
<td>1866</td>
<td>1452</td>
</tr>
</tbody>
</table>

The models were finished production cars and were thereby highly detailed in order to replicate the wind tunnel models as close as possible. Some simplifications of the geometry however existed to make the models possible to mesh and simulate. The CAD models consisted of two separated files, underbody with engine bay and the body work. These two parts were merged in ANSA in order to ensure gaps did not exist in the complete models. The cooling package consisted of radiators, condensers, charge air coolers, fans and shrouds. The radiator, condenser and charge air cooler were modeled as separate fluid zones with different viscosities in order to be able to simplify the geometry as much as possible but still capture the effects of it. The viscosities were obtained from the suppliers of the cooling package components.

Slick tyres were fitted to the geometries which were morphed into a shape with the contact patch corresponding to the cars static load. In order to replicate rotation of the rims a volume between the rim was set to a separate fluid zone which adds a rotational component to the passing flow and thereby makes the flow passing through the rims achieve the effects of rotational rims [7]. This was needed as the rims are stationary in steady state simulations. These separate fluid zones can be seen in Figure 2.2 as the covering surfaces between the rims.
The detailed underbody and engine bay parts for the V60 model. Note the surfaces between the rims which defined the separate fluid zones to capture the effects of the rotating wheels.

The cabin of the cars were closed off at the firewall (placed after the engine bay), underbody and the exterior which then created a closed volume for the cabin.

The coordinate system for the car can be seen in Figure 2.1 where the positive x-direction corresponded to the rear of the car, the positive y-direction to the right side of the car and the positive z-direction to the top of the car.

The front driveshafts were removed in the CAD model as they also were removed for the wind tunnel measurements. This was due to both cars having automatic gearboxes which could be damaged if run with the engine turned off. The cars were also four wheel drive and thereby the prop shaft was removed as it could be damaged.

2.1.1 Geometrical differences between CAD and wind tunnel models
In [4] the CAD model geometries was compared to the wind tunnel test models with distance measurements at certain points at the underbody. The measurements were performed in static conditions on the wind tunnel model and compared with the CAD geometry in ANSA. Differences between 13 and 22 mm were measured at the front under shields while variations up to 24 mm were measured at the tank panels for the S60 model. The differences were quite similar for the V60 except differences around 30 mm was measured for the tank panels and muffler. Note that these measurements are measured in static conditions with the CAD model and corresponded to a lower ground clearance for the wind tunnel model. Variation of these measurements could therefore have occurred as the geometry might have differed slightly after the volume mesh had been generated. The panels on the physical test objective might also had changed shape during the wind tunnel measurements, as the pressure distribution may had caused panels to flex and deform. These geometrical differences may have occurred due to manufacturing faults and tolerances for the fastener devices. Differences may also be due to simplifications in the CAD model to be more suitable for CFD simulations.

2.1.2 Considered configurations of the models
In order to ensure that the new developed methods were robust, a number of geometrical configurations were considered. The underbody configurations can be seen in Figure 2.3 and are described in Table 2.2.
Figure 2.3. Considered underbody configurations for the V60 and S60 models. Note the marked underbody configurations.

Table 2.2. Configuration description.

<table>
<thead>
<tr>
<th>Nr.</th>
<th>Description</th>
<th>Abbreviation</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Without front wheel deflectors</td>
<td>w/o fwDEFL</td>
</tr>
<tr>
<td>2</td>
<td>Without front undershield and extension</td>
<td>w/o FUS</td>
</tr>
<tr>
<td>3</td>
<td>Without engine undershield and extension</td>
<td>w/o EUS</td>
</tr>
<tr>
<td>4</td>
<td>Without right side underbody panel and left tank panel</td>
<td>w/o TP RH &amp; TP LH</td>
</tr>
<tr>
<td>5</td>
<td>Without left and right tank panel</td>
<td>w/o TP RH &amp; TP LH</td>
</tr>
<tr>
<td>6</td>
<td>With large triangle</td>
<td>LTR</td>
</tr>
<tr>
<td>7</td>
<td>With small triangle</td>
<td>STR</td>
</tr>
<tr>
<td>8</td>
<td>With separation edge on D-pillar</td>
<td>SED</td>
</tr>
<tr>
<td>9</td>
<td>Rim covers (flat rims)</td>
<td>RC</td>
</tr>
<tr>
<td>10</td>
<td>Closed cooling (grill and spoiler intake closed)</td>
<td>CC</td>
</tr>
</tbody>
</table>

Configurations 1-5 and 9-10 were considered for both the V60 and S60 models while configurations 6-8 only were considered for the V60 model as they appeared around the rear roof spoiler and D-pillars. Configuration 6 and 7 were a triangle surface fitted between the rear light and spoiler while the separation edge was a sharpened edge on the rear lights. See Appendix A.2 for figures of configurations 6-8. The rim cover configuration was only considered for the detailed tyre simulations as the effect of the detail level on the tyres had shown significant importance on the rims flow behavior [7], [8] and [9].

2.2 Description of VCC wind tunnel

VCC wind tunnel is a fully automatic closed wind tunnel with semi slotted walls at the test section. It was built in the 1980s but was upgraded in 2006 to the current configuration. The wind tunnel is a multi-purpose wind tunnel as it has possibilities to do aerodynamic, thermodynamic and contamination experiments. Just before the test section a heat exchanger, honeycomb and turbulence nets are placed to be able to control the temperature but also to reduce the turbulence level at the test section. The wind tunnel layout can be seen in Figure 2.4.
The test section has a length of 15.8 m, width of 6.6 m and a height of 4.1 m which gives it a cross-sectional area of 27.06 m². However, the cross-sectional area gets an increased effect of the slotted walls which has a 30 % open-area ratio in order to reduce the blockage effects of the surrounding walls in the test section. In order to be able to replicate road like conditions the test section has a five belt moving ground system. The moving ground is placed on a turntable which makes it possible to do yaw angles up to 30 degrees. In front of the turntable a boundary layer control system removes the boundary layer at the test section floor to better correspond to road like conditions. The test section with its turntable and slotted walls can be seen in Figure 2.5. The test section is not entirely symmetric as four vertical support beams are placed at the right side of the tunnel to support the slotted walls which are made of Plexiglas in order to make the users of the wind tunnel able to see the test section.
2.3 Software
The software’s used in this thesis are presented in this section.

2.3.1 ANSA
ANSA is a pre-processor software created by CAE Systems which was good to use for modifying existing CAD geometries and can also generate meshes suitable for simulations. VCC uses ANSA to clean-up the CAD geometry from the designers and to add or modify the geometry for configuration simulations. In this thesis ANSA was used to clean the geometry, thickening the surfaces for the front and rear of the cars and also to morph the surface mesh of the detailed tyre into correct shape corresponding to the load of the car and the rotational forces.

2.3.2 Sharc Harpoon
Harpoon is a mesh generator created by Sharc. It is known for its fast mesh generation but also for its capabilities to create high quality body-fitted hex meshes. It is designed to cope with very complex geometries which makes it very useful for industrial applications were geometries with a lot of details needs to be simulated.

Harpoon uses a Cartesian octree meshing technique to generate the mesh. With an octree approach the whole domain is meshed and then the parts where no mesh is wanted is removed. As the Cartesian grid cuts the geometrical surfaces and thereby results in a “stair step” mesh which not is desirable for all types of simulations. To prevent this Harpoon uses a boundary-fitting technique where the nodes of the hexahedral elements are moved onto the surface. If however the cell quality (skewness) becomes too poor special algorithms and pseudo-integral calculations are performed to split the cell into pyramid, tetrahedral or wedge elements to ensure better cell quality. Due to this mesh method good quality meshes can be achieved fast and also to low memory usage.

Prisms layers can be generated at the surfaces for the newer version of Harpoon. However, it can only be generated after the initial mesh is created as it replaces the cells closest to the surfaces. This limits the possible space for the prisms layers and thereby the number of layers, layer thickness and growth rate.

In order to keep the quad dominance of the mesh Harpoon uses mainly quads as surface mesh elements but also triangle elements. Instead of using an O-grid approach to keep the quad dominance it splits quad elements into triangle elements which can cause circles to appear in the mesh and cause effects on the volume cells and thereby flow features. This may also be due to how Harpoon defines surfaces as the quad splits increases with the curvature of the surface as can be seen in Figure 2.6.

Figure 2.6. Circles consisting of triangle elements created at the surface mesh caused by Harpoon to ensure a hexahedral dominant mesh and due to Harpoons method to define surfaces.

In this thesis Harpoon was used for volume mesh generation.
2.3.3 **Ansys Fluent**
Ansys Fluent is a commercial cell based CFD solver owned by Ansys Inc. Ansys Fluent has capabilities to simulate and model turbulent flow, heat transfer and chemical reactions for industrial and research applications. Fluent is widely used in the automotive industry due to its fast, robust solver which in many cases been able to achieve good correlation to experimental data. All the simulations in this thesis have been solved in Ansys Fluent version 14.5.0.

2.4 **VCC CFD procedure**
The standard CFD procedure at VCC consist of several steps which includes generation of geometry, numerical mesh and solution.

The CAD handling is done in ANSA where the obtained geometries are cleaned and simplified to better suit the demands of the CFD simulation. Before the geometry is exported into the mesh generator the geometry needs correct PID (Personal Identification) to be set for the surfaces as it later decides the surface mesh size. From ANSA a geometry representable Fluent surface mesh is created and exported to Harpoon as it better can handle surface meshes than full CAD geometries.

The volume mesh is created in Harpoon where the base level of the cell size is set and then the surface mesh sizes are set to the wanted levels by the naming of the PID. The domain size and surface mesh expansion and refinements boxes are also set for the volume mesh generation in Harpoon. The created volume mesh is a hexahedral dominant mesh with pyramids and tetrahedral cells to achieve good quality.

The volume mesh is then exported into Ansys Fluent where the boundary conditions and solver settings are set. When the simulation is completed an output file is written which is imported into an Excel document where the aerodynamic forces and its deviations are calculated and presented. For post-processing Excel and Python is used for aerodynamic force description while Ensight is used for flow visualization.
3. **Theory**
This chapter briefly presents vehicle aerodynamics theory and some of the governing equations for fluid mechanics.

3.1 **Fundamental aerodynamics**
Aerodynamic is the study of fluid mechanics for air. The aerodynamic forces acting on an object when air travels over it is due to the pressure and shear stress distribution over the surfaces on the object. The force caused by the pressure acts normal to the surface while the shear stress force acts tangential to the surface.

For flow problems the dimensionless Reynolds number should be considered. It is defined in Equation 3.1 and is a function of the flow velocity, density, kinematic viscosity and a characteristic length.

\[
Re = \frac{\nu_0 \rho l}{\nu}
\]  

(3.1)

The Reynolds number gives a measure of the inertial forces and viscous forces of the flow which makes it useful for predicting if the flow mainly will be laminar or turbulent, where the transition from laminar to turbulent will occur and the boundary layer thickness, \( \delta \).

When the air moves over surfaces it is affected by viscosity effects due to the frictional force between the air and the surface which makes the velocity at the surfaces equal to zero. Looking at the flow just adjacent to the surface an increase of the flow velocity until the free stream velocity can be seen. This generates a velocity profile which defines the velocity boundary layer which is illustrated in Figure 3.1. The distance from the surface to the point where 99 % of the free stream velocity is achieved is defined as the velocity boundary layer thickness. This only affects a thin part of the flow adjacent to the surface. The boundary layer thickness is dependent on the Reynolds number and therefore increases with the travel over surface.

![Figure 3.1. Creation of velocity boundary layer over a flat plate. Note how the boundary layer thickness increases over the plate.](image)

Inside the boundary layer the viscous effects are significant while outside the boundary layer the flow behaves inviscid except at separations where viscous effects may occur. The boundary layer can be divided into several parts by use of the non-dimensional variable \( y^+ \) which is defined in Equation 3.2

\[
y^+ = \frac{\rho u_\tau y}{\mu}
\]  

(3.2)

where \( \rho, u_\tau \) and \( \mu \) is the density, frictional velocity and viscosity for the flow close to the wall while \( y \) is the normal distance from the wall. The boundary layer consists of a viscous sublayer, buffer layer...
and the log-law region which can be seen in Figure 3.2. The viscous sublayer is the closest part to the wall and is dominated by the viscous effects and is in practice very thin as it reaches to around $y^+ = 5$. In the viscous sublayer a linear relationship between the mean velocity and the distance from the wall can be found. Outside the viscous sublayer the buffer layer and log-law region exist. In the log-law region the viscous effects is still important but also the turbulent effects are significant. The name log-law region comes from the logarithmic relationship between $u^+$ and $y^+$. $u^+$ is defined in Equation 3.3.

$$u^+ = \frac{U}{u_\tau}$$  \hspace{1cm} (3.3)

$u_\tau$ is the friction velocity and is a function of shear force, $\tau_w$ and density, $\rho$. The log-law region reaches from $y^+ = 30$ to values of hundreds as the upper limit depends on the Reynolds number. The buffer layer between the viscous sublayer and the log-law region is a blending region and occurs approximately between $y^+ = 5$ to 30.

![Diagram of boundary layer](image)

Figure 3.2. Parts of the boundary layer. Note the linear behavior in the log-law region as the x-axis is in logarithmic scale.

Depending on the Reynolds number the boundary layer will behave laminar or turbulent. Critical Reynolds number exists from experiments where the flow goes from laminar to turbulent. The transition zone where the flow goes from laminar to turbulent occurs around $Re_{crit} = 5 \cdot 10^5$.

Laminar flow is characterized by low Reynolds number where the flow behaves smoothly with no or negligible fluctuations. For laminar flow over a flat plate the flow can be seen as 2-dimensional as the flow only moves parallel and perpendicular to the plate. Turbulent flow is characterized by chaotic random behavior in all directions. For turbulent flow over a flat plate the flow moves in all the directions due to the irregularities of the turbulent flow structures.

The pressure distribution imposed by the external flow strongly influences the boundary layer. The flow separation starts in the boundary layer and can be due to an increased pressure distribution in the flow direction or due to that the flow cannot follow the surface as a too large decrease of energy in the boundary layer leads to separation.

The density of the air can change in the flow as gases are compressible. The compressibility effects (change of density) can generate differences in the flow and hence the aerodynamic forces when traveling over surfaces. However, at lower speeds ($M < 0.3$) the compressibility effects are negligible.
3.2 Road vehicle aerodynamics

When talking about road vehicle aerodynamics the flow can be divided into three categories, the flow around the body, through the body and within the machinery. The first and second is highly connected as for example the cooling flow in the engine bay often is released into the flow at the underbody, thereby causing disturbances to the external flow. For road vehicles often two aerodynamic forces are considered, drag and lift. The drag force acts in the opposite direction of the vehicles travel direction and thereby tries to slow down the vehicle while the lift force tries to lift or press the car to the ground. The flow around passenger cars is similar to flow around bluff bodies where the main drag is caused by the pressure drag force. The pressure drag force can be described as the difference between the pressure at the front and rear surfaces of the car. The drag caused by the friction between the air and surfaces are usually small and corresponds to 5-10 % of the total drag force of a passenger car. For vehicles traveling with velocities over 70 km/h the aerodynamic drag force is the main source of the energy consumption which proves the importance of aerodynamic efficiency in order to reduce the energy consumption for vehicles [10] and [13]. The main drag force sources for a passenger car can be seen in Figure 3.3.

![Figure 3.3. Typical drag force sources and contribution for a passenger car [13].](image)

The lift force is caused by the pressure difference between the upper and bottom surfaces of the vehicle. For passenger cars the lift force can cause concerns for the high speed handling.

The flow around passenger cars can rarely be assumed as symmetrical as external features of the body work may differ from the sides but mainly because the geometry in the engine bay and for the underfloor rarely are symmetrical. Crosswind is typical in road like condition which also makes the flow behave asymmetrical.

The flow field around a passenger car usually consists of accelerated turbulent flow at the underbody due to the ground effect. The underbody of a passenger car often has protruding components which disturb the flow greatly and have therefore for many years been seen as an extremely rough flat plate causing a lot of turbulence and drag. In recent years the knowledge about the flow at the underbody has increased and today many car manufacturers focus on covering most of the protruding components at the underbody in order to reduce the drag caused by the underbody.

As seen in Figure 3.3 the wheels contribute of around a quarter of the total drag for a passenger car. The flow around rotating tyres is very complex and especially when it is inside a wheelhouse. It has been seen in studies [9], [14] and [15] that the rotational effect generates lower drag and reduces lift for a partly faired wheel. Generally a high pressure zone occurs at the front of the wheel as the flow hits the front side of the tyres while at the rear a low pressure wake occurs. This low pressure wakes give rise to vortices starting at sidewalls edges of the contact patch [15].
3.2.1 Aerodynamic coefficients
In order to be able to compare the aerodynamic efficiency for different vehicles the use of aerodynamic force coefficients are useful. By normalizing the force with the bulk flow dynamic pressure and a reference area for the vehicle a force coefficient is achieved. This is usually done for the drag and lift forces which are presented in Equation 3.4 and 3.5.

\[
C_D = \frac{\text{Drag force}}{0.5 \cdot \rho_\infty \cdot v^2_\infty \cdot A_{\text{ref}}} \quad (3.4)
\]

\[
C_L = \frac{\text{Lift force}}{0.5 \cdot \rho_\infty \cdot v^2_\infty \cdot A_{\text{ref}}} \quad (3.5)
\]

For road vehicles the frontal area is often used as the reference area. The aerodynamic coefficients are functions of the Reynolds number and Mach number as different flow field hence different force distribution may occur. However, for external aerodynamics for passenger cars the aerodynamic coefficients usually are quite insensitive for Reynolds number changes [10]. Counts are often used when describing changes of the force coefficients. A count is referred as a thousand of a force coefficient.

3.3 Governing equations
CFD is based upon some governing equations of fluid mechanics of what are mathematical assumptions of some conservation laws of physics. All the essential equations are in some manner based upon these equations in order to describe the fluids behavior. The equations can be divided into three parts which represents different physical laws.

The first equation called the continuity equation describes the conservation of mass, that matter cannot be created nor destroyed. This simply requires that the rate of change of mass in control volume is equal to the mass flux crossing through the surface of the control volume. The continuity equation described in Cartesian coordinates can be seen in Equation 3.6.

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0 \quad (3.6)
\]

The momentum equation describes Newton’s second law, that the rate of change of momentum equals the sum of forces acting on the fluid. Both surface and body forces acts on the fluid control volume. However, the volume forces are usually added as an additional source outside of the surface forces. The momentum equation also known as the Navier-Stokes equation is presented in Equation 3.7.

\[
\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} + \frac{1}{\rho} \frac{\partial p}{\partial x_i} = \mu \left( \frac{\partial^2 u_i}{\partial x_j \partial x_j} \right) + X_i \quad (3.7)
\]

The first term describes the local acceleration, the second the advection, the third the pressure gradient and the fourth terms the diffusion for the fluid. The last term correspond to the added body forces in the specific direction.

The energy equation is needed to be solved when temperatures and compressible flow are to be simulated. It corresponds from the first law of thermodynamics and describes the time rate of change of energy which is equal to the net rate of heat added plus the net rate of work done by the fluid.

3.3.1 Turbulence
Almost all flow problems in the industry are turbulent flow which is characterized by its chaotic random motion. Turbulent flow occurs at higher Reynolds number as the inertia forces are large and
amplifies disturbances. This behavior causes fluctuations in velocity and pressure with time which thereby always makes a three dimensional behavior for the turbulent flow. This can be compared to laminar flow which can behave in one, two or three dimensions due to its lack of fluctuations.

Turbulent flow consist of so called turbulent eddies which is (rotational) flow structures. Due to these structures particles which originally where separated by a long distance can be brought closer which makes turbulent flow a good exchanger of momentum, heat and mass.

3.3.2 Reynolds Average Navier-Stokes

In order to reduce the computational power required to solve the Navier-Stokes equations (as it require enormous amount of computational power) RANS modelling was invented. By use of Reynolds decomposition (seen in Equation 3.8) which is splitting the flow variables in the equations into two components, one time-averaged and one fluctuating component the mean quantity of the flow can later be achieved

\[ \Phi = \Phi^\prime \quad \text{(3.8)} \]

In Equation 3.8 the \( \Phi \) and \( \Phi' \) are functions of the position and the time while the \( \Phi^\prime \) is the time-averaged steady quantity. The definition of the time-average components can be seen in Equation 3.9.

\[ \Phi_i^\prime = \frac{1}{\Delta t} \int_{t}^{t+\Delta t} \Phi_i \, dt \quad \text{(3.9)} \]

Inserting the Reynolds decomposition for all the velocity components and pressure component into the incompressible continuity equation and the Navier-Stokes equation results in the RANS-equations seen in Equation 3.10 and 3.11.

\[ \frac{\partial(u_i^\prime)}{\partial x_i} = 0 \quad \text{(3.10)} \]

\[ \rho \frac{\partial \Phi_i^\prime}{\partial t} + \rho \frac{\partial \Phi_i^\prime u_j^\prime}{\partial x_j} = - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial \Phi_i^\prime}{\partial x_j} + \frac{\partial \Phi_j^\prime}{\partial x_i} \right) - \rho u_i^\prime u_j^\prime \right] \quad \text{(3.11)} \]

From the last terms in Equation 3.11 the Reynolds stresses can be extracted and are defined as \( \rho \Phi_i^\prime \Phi_j^\prime \).

As these Reynolds stresses are unknown they need to be appropriately modeled by use of turbulence models in order to be solve able.

3.3.3 Turbulence modelling

Turbulence modelling is one of the key features in CFD and there exist many levels of it. The turbulence models usually consist of some extra set of equations which are responsible to model parts or the full turbulent flow behavior. It is also possible to simulate flow problems without use of
turbulence models by directly solving the continuity and Navier-Stokes equations. However, due to the computational cost this is not possible to use in the industry, not even for smaller applications. Instead of resolving all the turbulent structures models exist where parts of the structured are modeled. LES and DES are examples of turbulence models where the large structures are resolved while the smaller structures are modeled. However, these models are still in the edge of feasibility for the industry and can often only be used for smaller applications where the Reynolds number is low due to its required computational power. Therefore does today’s industry rely more on turbulence models that models all the turbulent structures as much less computational power is needed.

One of the most commonly used turbulence models in the industry is the standard $k$-$\varepsilon$ model. It is a two equation turbulence model initially created by Launder and Spalding [16] in 1974. The model is a semi-empirical model based on observations from several experiments with shear flow, mixing layers and jets. It is known for its robustness, economy and reasonable accuracy for many types of flows [5] and [16]. The model is an Eddy viscosity model as it is based on the Boussinesq assumption which relates the Reynolds stresses to the mean flow velocity gradients by use of the turbulence viscosity. The turbulence viscosity is an introduced variable and is defined by Equation 3.12 for the $k$-$\varepsilon$ models.

\[
\mu_t = \frac{\rho C_{\mu} k^2}{\varepsilon},
\]  
(3.12)

Two transport equations exist, one for the turbulent kinetic energy, $k$ and another for turbulent dissipation rate, $\varepsilon$. In the derivation of the model it is assumed that fully turbulent flow occurs and the molecular viscosity has no effect. This means that no consideration of the turbulence level exist in the model which limits the accuracy of the model.

Various modified formulations for the $k$-$\varepsilon$ model exist, thereby the realizable $k$-$\varepsilon$ model [17] proposed by Shih. The realizable $k$-$\varepsilon$ model differs in two major ways from the standard $k$-$\varepsilon$ model, it has a different formulation of turbulent viscosity, $\mu_t$ and a different transportation equation for the turbulence dissipation rate. Due to certain mathematical constraints for the Reynolds stresses it achieves it name, realizable.

Same equation (3.12) as for the standard $k$-$\varepsilon$ model is used for the turbulent viscosity, $\mu_t$. However, instead of using a constant value for $C_{\mu}$ it is variable and is defined in Equation 3.13.

\[
C_{\mu} = \frac{1}{A_0 + A_5 \frac{k U^*}{\varepsilon}},
\]  
(3.13)

$A_0$ and $A_5$ are constants while $U^*$ depend on the mean rate-of-rotation tensor and strain rate tensor. The definition of the turbulent kinetic energy, $k$ can be seen in Equation 3.14.

\[
k = \frac{1}{2} \left( u_i' u_i' \right)
\]  
(3.14)

The transport equations for the turbulent kinetic energy and dissipation rate can be seen in Equation 3.15 and 3.16

\[
\frac{\partial}{\partial t} (p k) + \frac{\partial}{\partial x_i} (p k u_i) = \frac{\partial}{\partial x_i} \left[ (\mu + \mu_t) \frac{\partial k}{\partial x_i} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k
\]  
(3.15)
\[
\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right) \frac{\partial \varepsilon}{\partial x_i}\right] + \rho C_1 \varepsilon S - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\nu \varepsilon}} + C_{1\varepsilon} \frac{\varepsilon}{k} C_{3\varepsilon} G_b + S_\varepsilon
\]

where \(G_k\) and \(G_b\) represents the turbulent kinetic energy caused by the velocity gradients and buoyancy. \(Y_M\) represents the influence of expansion in compressible turbulence to the overall turbulence dissipation rate. \(C_{1\varepsilon}, C_{3\varepsilon}, C_2, S_k\) and \(S_\varepsilon\) are constants while the \(\sigma_k\) and \(\sigma_\varepsilon\) are the turbulent Prandtl numbers for \(k\) and \(\varepsilon\) [5] and [17].

These differences make the realizable \(k-\varepsilon\) model outperform the standard \(k-\varepsilon\) model for rotational flow and boundary layer exposed to strong pressure gradients. The differences also improve the modeled behavior of separated flow and vortices [5].

3.3.4 Wall treatments

As the turbulent flow is heavily affected by the walls it is important to capture the effects of the wall bounded flow. Due to the friction between the fluid and the surface high velocity gradients is caused which are important to resolve or model accurate. However, as the Reynolds number increases the boundary layer thickness decreases and the cost to resolve the wall flow increases greatly. Instead of using a fine mesh resolution for the boundary layer wall functions can be used to model the closest parts of the wall bounded flow and thereby a coarser mesh can be used. The wall functions in Ansys Fluent are built from semi-empirical equations and functions. As the boundary layer can be divided into several regions (see Chapter 3.1 and Figure 3.2) it is important to model the behavior from the correct region. Ansys Fluent uses the wall unit \(y^*\) instead of \(y^+\) to decide its wall modelling mode. Its value describes the dimensionless distance from the wall and its definition can be seen in Equation 3.17.

\[
y^* = \frac{\rho C_u k^{1/4} y_p^{1/2}}{\mu} \quad (3.17)
\]

\(k_p\) and \(y_p\) is the turbulent kinetic energy and distance from the wall to the centroid of the cell in contact with the wall. However, the \(y^*\) and \(y^+\) are approximately the same value in equilibrium turbulent boundary layers and depends mainly on the local Reynolds number and the node distance from the wall. Depending on the \(y^*\) value the mean velocity is estimated differently. Depending on an if statement the mean velocity is calculated differently for the standard wall function. If \(y^* > 11.225\) the mean velocity is calculated with Equation 3.18

\[
U^* = \frac{1}{\kappa} \ln(E y^*) \quad (3.18)
\]

where \(\kappa\) is the von Kármán constant and \(E\) a constant from empirical experiments. However, if the \(y^* < 11.225\) the mean velocity is estimated with Equation 3.19 which is an assumption for laminar flow with the stress-strain relationship.

\[
U^* = y^* \quad (3.19)
\]

In Ansys Fluent it is recommended to have \(y^*\) and \(y^+\) lower than 5 or over 15 as accuracy problems may occur for the standard and enhanced wall treatments.

The enhanced wall treatment is similar to the standard wall function in many ways except it is less sensitive of the \(y^*\) values. This is due to that the enhanced wall treatment combines a two-layer zonal
model with enhanced wall functions. If the mesh is sufficient fine at the wall so the viscous sublayer can be resolved the enhanced wall treatment is equal to the two-layer zonal model.

For the two-layer zonal model the domain is divided into a viscosity affected region and a fully turbulent region. The viscosity affected region is fully resolved to the viscous sublayer. The two-layer zonal model uses integration of the enhanced wall treatment to specify the turbulent dissipation rate and the turbulent viscosity for the near wall cells. In the fully turbulent regions the ordinary turbulence model is used. The turbulent viscosity for the viscous region is then blended in to fully turbulent regions turbulent viscosity (which is the turbulence models definition) by a blending function [5].

The enhanced wall treatment also blends the linear and logarithmic parts of the boundary layer by use of a blending function [5]. By use of this approach effects in the fully turbulent region (e.g. pressure gradients) can be accounted for in the wall modelling. It also reduces the uncertainty when the $y^*$ falls inside the buffer layer.

3.4 Numerical methods
Several solver and discretization schemes exist in Ansys Fluent to solve the fluid mechanics equations. Here are the solver and discretization schemes used in this thesis presented.

3.4.1 Solver scheme
The coupled solver scheme solves the momentum and continuity equations simultaneously compared to the segregated solver schemes who solves the equations separately. Due to this, the coupled solver scheme can in many cases generate faster convergence than the segregated solver schemes.

3.4.2 Numerical discretization
For discretization of scalar transport the first and second order Upwind schemes were used in this thesis. The name Upwind results from the derivation from quantities from cells upstream. For first order Upwind schemes the face value, $\phi_f$ for the cell is set equal to the cell center of the upstream cells cell center value, $\phi$. The second order Upwind schemes uses a multidimensional linear reconstruction approach which considers the upstream cells centered value plus the gradient of the scalar in the upstream cell. This generates better accuracy but increases the computational cost slightly.

3.5 Numerical mesh and elements
As the fluid mechanics equations are solved numerically the domain needs to be divided into non-overlapping subdomains, usually called control volumes or cells. A set of subdomains can be referred as a mesh. These subdomains represent geometry in the domain and are needed in order to be able to solve the fluid mechanics equations numerically for the domain. In these cells the flow and its fluid properties is described. Due to this the accuracy of the solution is heavily affected by the mesh resolution and its distribution. A finer mesh resolution typically increases the accuracy of the solution as more details of the flow can be captured. However, the computational cost also increases with the mesh resolution. The mesh resolution and its distribution can also affect the convergence of a solution greatly.

Five types of cell elements exist for the meshing in Harpoon, hexahedral, tetrahedral, pyramid, wedge and polyhedral. All the elements have advantages and disadvantages. The hexahedral element is usually the one to prefer as a more structured mesh can be achieved which reduces the cell count compared to tetrahedral cells. Hexahedral cells can however be hard to generate for complex geometries which tetrahedral cells are better suited for.
The quality of the mesh is also an important factor for the solution as bad cell quality can cause numerical issues as convergence problems and numerical diffusion. An often used measure for cell quality is the cell skewness which is defined in Equation 3.20.

\[
Skewness = \frac{Optimal \ cell \ size - cell \ size}{Optimal \ cell \ size}
\] (3.20)

Numerical diffusion due to skewness is a problem for tetrahedral cells as it occurs in the discretization with increased skewness. This effect is however less in hexahedral cells but can occur when hexahedral cells have high aspect ratio perpendicular to the flow direction. Extreme levels of skewness \((skewness \geq 1)\) can be achieved for cells which then cause negative volumes due to the cell growths into itself. This can often cause convergence problems as the discretization becomes poorly and may result in unphysical values for the scalars.

Wedges and thin hexahedral cells are usually used near the wall to form prisms layers to resolve high velocity gradients of the boundary layer close to the wall. Cells in prisms layer can have large aspect ratios in order to capture the gradients as the flow in the boundary layer often is parallel to the surface.
4. Method

In this section is the processes presented which were used in this thesis in order to obtain the results. First will the numerical set-up be presented which were used for the simulations and then the used meshing methods. The used methods for the implementation of prisms layers, effects of detailed tyres and effects on the aerodynamic forces of mesh refinements for engine bay and underbody are then presented in order.

Several geometry and mesh problems occurred during the study and are presented in Appendix A.1.

The methods generally followed the VCC CFD procedure with geometry processing in ANSA, volume meshing in Harpoon, solving in Ansys Fluent and post-processing in Ensight.

4.1 Numerical set-up

As the results of the developed methods in this thesis was to be compared to results from CFD simulations performed in [4] the numerical set-up needed to be identical or at least similar to ensure that possible differences came from the new developed methods.

Due to only 0° yaw conditions were considered for the simulations the domain size presented in Table 4.1 was used for all the simulations.

Table 4.1. Used domain size for both the models for the simulations.

<table>
<thead>
<tr>
<th>Length</th>
<th>Width</th>
<th>Height</th>
</tr>
</thead>
<tbody>
<tr>
<td>50 m</td>
<td>9.5 m</td>
<td>10 m</td>
</tr>
</tbody>
</table>

The domain size for the simulations corresponds to almost 11 car lengths where the inlet was placed around 3.5 car lengths from the car models to not affect the solution around the model. The domain size can be seen in Figure 4.1 for the V60 model. The simulated domains front blockage area at the model was 2.39 % which well meets the requirement of maximum 5 % front area coverage in order to not affect the flow behavior over the car [18].

Figure 4.1. Used domain for the CFD simulations where only the car model, ground, right wall and outlet can be seen. Due to the considered geometries had the almost the same dimensions the same domain size was used for both car models.

The used boundary conditions are presented in Table 4.2.
Table 4.2. Used boundary conditions for the CFD simulations.

<table>
<thead>
<tr>
<th>Boundary Condition</th>
<th>Condition</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>Velocity-inlet</td>
</tr>
<tr>
<td>Outlet</td>
<td>Zero pressure outlet</td>
</tr>
<tr>
<td>Ground</td>
<td>Moving wall</td>
</tr>
<tr>
<td>Domain sides and top</td>
<td>Symmetry (free slip condition)</td>
</tr>
<tr>
<td>Exterior, underbody</td>
<td>No slip wall</td>
</tr>
<tr>
<td>Wheel and rim</td>
<td>Rotating wall</td>
</tr>
<tr>
<td>Air between the rim spokes</td>
<td>MRF (Moving Reference Frame) fluid zone</td>
</tr>
<tr>
<td>Radiator, charged air cooler, condenser</td>
<td>Fluid zone with different viscosity</td>
</tr>
<tr>
<td>Cooling package fans</td>
<td>Rotating wall and MRF fluid zones</td>
</tr>
</tbody>
</table>

The inlet velocity was set to 27.778 m/s (100 km/h) which corresponded to a Reynolds number of 8591500 based on the length of the cars. At the inlet 0.1 % turbulent intensity was used as it corresponded to the turbulence level at the inlet of the test section in the wind tunnel. A viscosity ratio of 200 was also set as it corresponded to values calculated for $k$ and $\varepsilon$ for the wind tunnel but remained constant with the inlet velocity.

The used air properties can be seen in Table 4.3 and was the air material properties for a temperature of 20°C which normally was operating temperature for the wind tunnel.

Table 4.3. Used air properties for the CFD simulations which corresponded to the air material properties at a temperature of 20°C.

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>1.205 kg/m³</td>
</tr>
<tr>
<td>Viscosity</td>
<td>1.805 · 10⁻⁵ kg/ms</td>
</tr>
</tbody>
</table>

Due to only steady-state simulations were considered the realizable $k$-$\varepsilon$ RANS model was used as in [4] due to its suitability for high Reynolds number flows. It has been used at VCC for many years as good drag correlation with the wind tunnel measurements have been obtained with it. The Standard wall function was mainly used as in [4] but Enhanced wall treatment was used for a single simulation in order to see if any noticeable difference occurred.

The simulations were initialized with the FMG (Full Multi Grid) which used the Euler equations (inviscid Navier-Stokes equations) to solve the pressure and velocity field (but no turbulence or transport equations) on a set of multi grids. As it was reasonably fast to generate and achieves a flow field similar to a RANS simulation decreased simulation time was obtained when compared to an initial solution of constant values.

The pressure-based solver was used for the simulations in Ansys Fluent as it was developed for incompressible low speed flows and therefore was suitable for road vehicle aerodynamics [5]. The coupled scheme was used as solver scheme due to it provides better performance than the segregated schemes [5]. The used discretization schemes for the momentum and turbulence transportations are presented in Table 4.4.

Table 4.4. Used discretization schemes for the CFD simulations in Ansys Fluent.

<table>
<thead>
<tr>
<th>Scheme</th>
<th>Standard</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure</td>
<td>Second Order Upwind</td>
</tr>
<tr>
<td>Momentum</td>
<td>First Order Upwind</td>
</tr>
<tr>
<td>Turbulent Kinetic Energy</td>
<td>First Order Upwind</td>
</tr>
<tr>
<td>Turbulent Dissipation Rate</td>
<td>First Order Upwind</td>
</tr>
</tbody>
</table>
Under-relaxation was used to minimize the risk of divergence during the start of the simulations and to reduce the amplitude of the fluctuations for the aerodynamic forces at the end of the simulations. 3500 iterations were set for the simulations due to that several iterations was needed for the flow to become fully converged inside the complex engine bay and the underbody.

No convergence criteria were set during the simulations for the residuals or the aerodynamic forces. The solutions were instead deemed converged if the residuals had dropped below $5 \cdot 10^{-4}$ and the standard deviation of $C_D, C_L, F$ and $C_L, R$ were equal or less than 0.001 for the last 500 iterations. From identical simulations the numerical error was estimated to be around 0.001 for the aerodynamic force coefficients.

A typical simulation took less than six hours to run on 480 CPU’s at VCC calculation clusters.

### 4.2 Computational mesh

The computational mesh consisted of a hexahedral dominant mesh created in Harpoon. Due to the initial mesh was built as a non-conformal mesh the cell size was divided into levels. The base level was set to 40 mm as it had been used in the simulations performed in [4] and corresponded to VCC’s currently used mesh method AEDCAE02. The surface mesh levels were set after the current meshing method but modifications were done for some of the investigated objectives. Largest surface mesh size was set to 5 mm while refinements consisting of surface mesh size of 2.5 and 1.25 mm were used where good geometrical representation was deemed important and complex flow expected. This was typically around the cooling package, mirrors, D-pillars and small curvatures.

In this thesis two main mesh methods have been used. The first was based on VCC’s current meshing method AEDCAE02 and used refinement boxes to refine the volume mesh. The second was based on the proposals in [6] and mainly used surface mesh expansion to refine the volume mesh. However, both methods used the same method for the engine bay where refinement boxes where added at the cooling package as a fine mesh was needed for the separate fluid zones to not obtain numerical issues.

The AEDCAE02 method includes refinement boxes for the mirrors, underbody and the rear wake to ensure good mesh resolution of the separated flow. The refinement box structure can be seen in Figure 4.2 and 4.3 and it can be clearly seen how mesh refinements are mainly considered for the rear of the car.

![Figure 4.2. AEDCAE02 mesh methods refinement box structure seen from the side. Here can also the pyramids connecting the non-conformal mesh be seen.](image)
Figure 4.3. AEDCAE02 refinement box structure seen from above with the front of the car pointing to the left.

The second meshing method considered mainly of surface mesh expansion which can be seen in Figure 4.4 and 4.5. This method still needed to use the refinements boxes used in AEDCAE02. However, the surface mesh size affects the volume mesh more as it grew into the volume mesh. The surface mesh expansion values were obtained from [6] but were reduced as too large mesh sizes occurred. The original values were designed so no refinement box for the underbody was needed but due to the modified values of the surface mesh expansion the refinement box for the underbody used in AEDCAE02 was implement again to ensure good mesh resolution at the underbody. The mesh expansion generated a smoother growth of the mesh out in the domain and thereby minimized the risk for the mesh to direct the flow behavior, making the solution less mesh dependent.

Figure 4.4. The new mesh method consisting of surface mesh expansion to generate smoother behavior of the mesh growth which also adapts after the car geometry. Refinements boxes still needed to be used at the cooling package, mirror, underbody and rear wake.
The new mesh method mainly consisted of surface mesh expansion to refine the volume mesh. Here is the mesh seen from above with the front of the car pointing to the left. Note the smoother growth of the volume mesh compared to Figure 4.3.

The new meshing method with surface mesh expansion was used for the objectives including prisms layer implementation and mesh refinement for the engine bay and underbody. The effects of the different meshing methods have been investigated in Section 4.3.3. The AEDCAE02 mesh method was used for the detailed tyre simulations except when detailed tyre and prisms layer were used as the surface mesh expansion technique then was used.

4.3 Implementation of prisms layers

In order to achieve a finer mesh resolution of the wall bounded flow and thereby better capture the effects of it, implementation of prisms layers in to the current mesh procedure was wanted. This would also help to get a better control of the $y^+$ distribution as the first layer thickness can be controlled more efficient.

As the AEDCAE02 mesh method not included any prisms layers the first node height varied with the surface mesh size. This led to few cells covered the wall bounded flow. Due to Harpoon generates a hexahedral dominant mesh stair step like $y^+$ distribution was caused as the node height varied over curved surfaces. With use of prisms layers the first node height variation would be smaller and result in smoother $y^+$ distribution.

Generally not less than ten cells covering the boundary layer was wanted in order to capture the wall bounded flow properly [5]. However, achieving ten or more layers for complex geometries is challenging as mesh quality problems often occurs. The mesh size may also increase with increased number of prisms layers which increase the computational costs but are still more cost effective compared to using finer surface mesh to increase the mesh resolution at walls.

Prisms layers were only considered for the external body work of the models as the engine bay and underbody was deemed too complex and detailed, hence not suitable for several prisms layers. There was also no need to generate prisms layers where no boundary layer was expected or where low speed flow appeared. However, tests of prisms layers on the tyres were performed and are presented in Section 4.5.

In previous study [6] prisms layers were generated for a V60 with full engine bay but due to problems with collapsed prisms layers unphysical flow behaviors were obtained in the solution. This happened due to Harpoon (version 5.2a7 and earlier) only could manage to build prisms layers on both sides of surfaces. The prisms layers behavior on both sides was highly connected and thereby if prisms layer collapse occurred on one side of the surface it caused the other sides to collapse too. This problem
occurred in [6] at the front of the car (as the CAD models only consisted of surfaces) as for example the prisms layers on the hood collapsed due to limited space inside the engine bay which can be seen in Figure 4.2 and 4.4. However, for rear of the exterior which was in contact with the cabin of the car this was not a problem as the created mesh inside the cabin was removed before prisms layer generation and thereby no prisms layers were generated on the inside of these surfaces. This can also be seen in Figure 4.6 where some parts of the bonnet were connected to a mesh free zone and thereby only the external side of the surface was in contact to the volume mesh and could achieve the prisms layers.

![Figure 4.6. Prisms layer collapse (inside the red square) at the bonnet due to prisms generation on both sides of the bonnet.](image)

From this the conclusion could be drawn that Harpoon (version 5.2a7) only is able to build prisms layers on one side of surfaces if the surface mesh only has contact to the bulk mesh on one side. Thereby thickening the front of the car's surfaces and removing the mesh inside the thickened surface, prisms layers would be possible to generate on the external side of the car's front.

Tests were performed on only a bonnet floating in the air to ensure correct behavior of the prisms layers. By copying the bonnet surface and placing it a distance from the original bonnet and connecting the outer edges with surfaces a thickened surface was created. For the thickened surface it could be seen that if the surface mesh size of the surface was larger than the thickness of the surface collapses of the thickened surface could occur. This caused zero thickness of the surface at the collapsed position which thereby caused prisms layer collapse as prisms layers were created at both sides of the collapsed region. By instead having a thickness larger than the surface mesh size the thickened surface was kept intact. This phenomenon occurred due to Harpoon uses the octree approach which builds a mesh for the whole domain (thereby even inside closed volumes). The volume cells were then forced to fit the surfaces in order to not create a stair-step mesh. When the cells were moved the nodes snapped on the closest surface and if the distances of surfaces were smaller than the volume mesh size in that region risks were that the nodes snapped on wrong surfaces. As some nodes may not snapped on the correct surface parts of the missed surface was removed.

In the given CAD geometries full engine bays existed but the models cabin were sealed off at the engine bay firewall, exterior and underfloor. Therefore was surface thickness creation only needed for the front of the car to the sealed of region. It was however later found that the lower part of the rear bumper also needed to be thickened as prisms layer collapse occurred at it.

As the surface mesh size mainly followed the AEDCAE02 mesh method the front external surfaces of the vehicles consisted of a surface mesh size of 5 mm which meant that the thickness of the front surfaces needed to be larger than 5 mm. The surface mesh size for the rear bumper was set to 2.5 mm for the major parts while for some cases a smaller surface mesh size was used. This meant that the thickness of the rear bumper surface needed to be larger than 2.5 mm.
4.3.1 Geometrical work

By using the scale function in ANSA it was possible to create a smaller copy of the external surfaces and place it on the inside of the original surfaces. By connecting the outer edges of the two set of external surfaces with surfaces it was possible to create the thickness of the front surfaces which can be seen in Figure 4.7. A complete description of the surface thickening creation method is presented in Appendix A.5. Due to the complexity of the geometry at the number plate holder, grill intake and spoiler intake it was better to not include those parts in the front surface thickening while for the rear of the car the cars towing system and its fasteners were also not thickened.

![Figure 4.7. The green and burgundy surfaces were the external side of the thickened front while the blue surfaces were the interior sides.](image)

In order to achieve at least a 5 mm thickness of the front surfaces a scaling factor of 0.992 was used. The scaling center was set to the front wheels center for the x and z-coordinates while it was set to zero for the y-coordinate in order to make the interior part scaled in all directions and be placed with correct distance from the original surface. A different PID (colored in blue in Figure 4.7) was set for the newly created inside of the front surfaces and the surfaces connecting the front exterior surfaces. This was needed to ensure that no prisms layers would be created on the inside of the front exterior surfaces. The thickness of the front exterior surface varied from 3 to 8 mm.

After the thickened surfaces had been created surfaces from other parts where sticking through the inner front exterior surface, seen in Figure 4.8. Due to these surfaces Harpoon sometime snapped on these surfaces, causing prisms layer collapses. By use of the intersection tool in ANSA penetrating surfaces could be removed to ensure that Harpoon not snapped on wrong surface and thereby minimizing the risk for prisms layer collapse.

Many surfaces of the hood insulation did penetrate the inner part of the front exterior (seen in Figure 4.8) but instead of removing them the hood insulation was displaced with 2 mm in the negative z-direction which significantly decreased the number of penetrating surfaces. Some of the penetrating surfaces could however be left as they stuck through less than 1 mm and thereby not risked to cause prisms layer collapse problems due the thickness of the surfaces was more than 6 mm for these region.

At the front spoiler the thickness of the surfaces became smaller than the surface mesh size but was solved by applying a different PID which in Harpoon was given a finer surface mesh in order to ensure that the surface mesh size was kept lower than the volume thickness. Suitably this occurred around the front spoiler which had curved surfaces which resulted in better geometry representation of the mesh. The interior surface was also PID divided in the same manner and set with the same surface mesh size as the risk of wrong snapping otherwise could occur.
The same method as for the front exterior thickened surfaces was used for the surface thickening creation of the lower part of the rear bumper. However, a different scaling center was used as the scaled rear bumper surface otherwise intersected to the original surface which can be seen in Figure 4.9. The scaling center was set equal to zero for all coordinates as it improved the positioning of the interior surfaces. Same scaling factor as the front was used.

Some geometrical simplifications were needed in order to make the geometry more suitable for the prisms layers. The gap between the head lights and the exterior needed to be closed off, in order to not cause prisms layer steps at the headlights edges which worsened the mesh quality.

The intersection between the rear spoiler and rear window caused a sharp corner which caused the prisms layers to collapse and even sometimes negative volumes to exist. This was solved by adding a small chamfer (seen highlighted in Figure 4.10) at the intersection in order to reduce the sharpness of the corner which mesh generators better can manage. To ensure that no negative volumes existed a commando was implemented into the Ansys Fluent script which removed the negative volumes. See Appendix A.8 for more details.
Figure 4.10. Highlighted surface was an added chamfer created to prevent the prisms layer to collapse at the otherwise sharp angle occurring between the rear spoiler and window. This was only necessary on the V60 (sportswagen) as no sharp corners existed on the S60 (sedan).

4.3.2 Prisms layer settings
For the prisms layer generation the surface mesh expansion originally proposed by [6] was used due the smoother refinement behavior.

As the simulation was performed with the realizable \( k-e \) model with standard wall function the \( y^+ \) should be around 30 which correspond to the log-law region of the boundary layer. From [6] it could be seen that a first layer thickness of 1 mm corresponded to a \( y^+ \) of around 30 for the considered Reynolds number.

Settings for the prisms layers were also tested on the bonnet test model. The target was to achieve as many prisms layers as possible but still obtain an acceptable transition (growth rate < 1.5) between the prisms layers and bulk mesh. A better transition could be obtained but resulted in less cells inside the boundary layer which was deemed more important. Different surface mesh sizes were tested for the bonnet in order to find the best possible settings for all the surface mesh sizes used at the models. This was needed as the mesh cell sizes corresponds to the surface mesh sizes and thereby different prisms layer settings was needed for the specific surface mesh size. In Table 4.5 is the obtained prisms layer settings presented.

Table 4.5. Obtained prisms layer settings from the test on only the bonnet which correspond to maximum achieved layers with good transition to the bulk mesh.

<table>
<thead>
<tr>
<th>Surface mesh size [mm]</th>
<th>First layer thickness</th>
<th>Number of layers</th>
<th>Growth rate</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>1 mm</td>
<td>6</td>
<td>1.2</td>
</tr>
<tr>
<td>2.5</td>
<td>1 mm</td>
<td>4</td>
<td>1.2</td>
</tr>
<tr>
<td>1.25</td>
<td>0.7 mm</td>
<td>3</td>
<td>1.1</td>
</tr>
</tbody>
</table>

Seen in Table 4.5 is that a smaller first layer thickness was used for the finest surface mesh size. This was due 3 layers could be achieved instead of 2 layers as it was more important to cover the boundary layer with more cells than focusing on the \( y^+ \) value [5].

Due to Harpoon used the octree approach the boundary layer was created after the initial mesh. Harpoon removed the cells which was close to the surface and then created the prisms layers. By using this method the number of prisms layers and its distribution was limited due to the limited space and thereby could maximum six prisms layers be achieved without poor transition to the bulk mesh even though several more layers was wanted.
Slight different approach for the PID dividing than for the AEDCAE02 mesh method was needed to ensure good quality of the mesh and reduce the problems for the prisms layer generation. Even though the intendance was to generate prisms layers for the full exterior non-suitable geometry for prisms layers existed for some parts of the exterior. As mentioned before the grill, front spoiler and number plate was not suited for prisms layers. As problems with too skewed cells occurred when prisms layers were placed on the A-pillar and rails it was deemed better to use a finer surface mesh size for these parts to keep the y+ and number of cells covering the boundary layer inside a reasonable level. At the rear of the car a larger region of fine surface mesh size was used as problems for the prisms layers could occur at the transition between different surface mesh sizes. In order to ensure that good resolution and quality of the mesh was obtained at the expected region for separation (D-pillar for example), it was better to ensure this problem occurred earlier where attached flow was expected. Problems for the prisms layers at the transition of surface mesh size was discovered to affect the flow severely for tests of a newer version of Harpoon which is more described in Appendix A.3. The new PID diving for the V60 can be seen in Figure 4.11 and was similar for the S60 model. The number after the PID name represent the surface mesh size level, where 6 was the finest (1.25 mm) used surface mesh size.

Figure 4.11. PID dividing used in this thesis in order to create volumes and have better control over the prisms layer distribution.

A finer surface mesh size was used for the lower part of the front as newer VCC models was expected to have a more rounded front and thereby separation may occur at the sides of the lower front. The front windscreen was also divided into two different PID (colored in purple and turquoise) as the front windscreen vipers would cause problems for the prisms layers due to its complex geometry. Building prisms layers on such complex geometries would only make the method more complex and un-robust.

In Table 4.6 are the used surface mesh size and prisms layers settings for the V60 presented. The S60 had same settings and PID dividing except at the rear where a slight change was made due to different geometry.
Table 4.6. Used surface mesh size and prisms layers settings for the V60 model.

<table>
<thead>
<tr>
<th>Part/PID</th>
<th>surface mesh size [mm]</th>
<th>First layer thickness [mm]</th>
<th>Number of obtained layers</th>
<th>Growth rate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Exterior front lower - 5</td>
<td>2.5</td>
<td>1</td>
<td>4</td>
<td>1.2</td>
</tr>
<tr>
<td>Exterior front upper - 4</td>
<td>5</td>
<td>1</td>
<td>6</td>
<td>1.2</td>
</tr>
<tr>
<td>Exterior main body - 4</td>
<td>5</td>
<td>1</td>
<td>6</td>
<td>1.2</td>
</tr>
<tr>
<td>Exterior mouldings - 6</td>
<td>1.25</td>
<td>1</td>
<td>1</td>
<td>1.0</td>
</tr>
<tr>
<td>Exterior rear - 6</td>
<td>1.25</td>
<td>0.7</td>
<td>3</td>
<td>1.1</td>
</tr>
<tr>
<td>Exterior rear - 4</td>
<td>5</td>
<td>1</td>
<td>6</td>
<td>1.2</td>
</tr>
<tr>
<td>Exterior rear spoiler - 5</td>
<td>2.5</td>
<td>1</td>
<td>4</td>
<td>1.2</td>
</tr>
<tr>
<td>Exterior shark fin - 4</td>
<td>5</td>
<td>1</td>
<td>6</td>
<td>1.2</td>
</tr>
<tr>
<td>Exterior mirror - 5</td>
<td>2.5</td>
<td>0.7</td>
<td>4</td>
<td>1.2</td>
</tr>
<tr>
<td>Exterior A-pillar - 5</td>
<td>2.5</td>
<td>0.7</td>
<td>4</td>
<td>1.2</td>
</tr>
</tbody>
</table>

As can be seen in Table 4.6 smaller first layer thickness were used for the mirrors and the part of the A-pillar as better mesh quality was achieved with the used values.

In order to ensure that the boundary layer was covered from the correct region in the boundary layer the $y^+$ distribution was important to inspect. Typically similar $y^+$ distribution was achieved for the V60 and S60 model and therefore is only the V60 models $y^+$ distribution presented.

In Figure 4.12 and 4.13 the $y^+$ distribution can be seen for the AEDCAE02 mesh strategy and the mesh with prisms layers. Large differences in the $y^+$ can be seen as the first layer thickness varied between 5 and 1.25 mm for the mesh without prisms layers as the node height depended on the surface mesh size. The $y^+$ values for the mesh without prisms layers was between 5 and 300. Patterns also existed in the $y^+$ distribution for the mesh without prisms layers which caused a large varying $y^+$ which may cause unphysical behavior of the boundary layer and thereby poor skin friction prediction.

Looking at the $y^+$ distribution for the mesh with prisms layers the values for the major parts of the exterior varied between 20 and 70 which was a more suitable value for wall functions. Also a smoother behavior of the $y^+$ distribution could be seen as the node height not depends on the surface mesh. Ring pattern for the $y^+$ could be seen for the prisms mesh and was an effect of Harpoon surface mesh build up which is more described in 2.3.2.

Poorer $y^+$ could be seen for the wheels, door handles and lower part of the windscreen for the mesh with prisms layers. This was due to no prisms layers were built on these surfaces due to the complex geometry which might cause mesh quality problems. Prisms layers on the tyres were however considered in Section 4.5.
When looking at the mesh with prisms layers higher \( y^+ \) values could be seen at the front corner, top windscreen, and mirror of the car as these regions were exposed to accelerated flow which increased the local Reynolds number and thereby the \( y^+ \) value. This could be prevented if the first layer thickness was decreased for these regions but would require some iterative processing before wanted distribution was achieved. It would also increase the manual work in the meshing method as different first layer thickness would be needed for parts of the cars. However, this effect was partly considered for the mirrors as accelerated flow often occurs in these regions due the curvature of the mirror and also due to the increased area blockage at the mirrors which causes accelerated flow.

In Figure 4.13 the \( y^+ \) distribution for the rear of the car can be seen. Lower \( y^+ \) values could generally be seen for both mesh methods at the rear of the car as result of the finer surface mesh size which lowered the first node height for the AEDCAE02 mesh method and the decreased first layer thickness for the prisms layer mesh method. This lowered \( y^+ \) generally corresponded better to the wall functions requirements. For the mesh without prisms layers the effect of the surface mesh could be clearly seen to affect the \( y^+ \) value. For the mesh with prisms layers also clear difference could be seen at the D-pillar of the car as a smaller first layer thickness was used in that region. However, the \( y^+ \) value was around 20 in that region which still well meets the wall functions requirements.

The \( y^+ \) for the underbody can be seen in Figure 4.14. High \( y^+ \) values could be seen for the panels and exhaust system while the rest of the underbody had a lower \( y^+ \) value due to high velocity flow only affected the panels and exhaust system. As the aim was to generate prisms layers mainly on the
exterior no effort was put to achieve better $y^+$ distribution for the underbody in this part of the thesis. Here can the effects of the hexahedral fitting technique used in Harpoon be seen for the front undershield as a step like behavior occurred for the $y^+$ distribution which not was favorable.

Figure 4.14. $y^+$ distribution at the underbody for the car models. Note the stair step distribution at the front undershield due to the fitting of the hexahedral cells of the hex dominant mesh which caused varying first node height from the surface.

Low $y^+$ occurred at some regions of the car models, especially at the wakes. However, instead of increasing the first layer thickness until a higher $y^+$ values were achieved it was better to maintain the same first layer thickness. In these wakes no boundary layer was created and therefore no need to model its behavior properly existed.

In order to verify that the mesh resolution in the boundary layer affected the behavior of it, velocity data for the five lines was exported and compared for the mesh with and without prisms layers for the V60 model. The lines were placed 1000 mm from each other on the top of the car in order to see the if differences occurred along the car which can be seen in Figure 4.15.

Figure 4.15. Velocity profiles comparison for the mesh without prisms layers and the mesh with prisms layers for five lines placed equally distanced over the car. Note the large difference in behavior at the front of the car and the similar behavior at the rear or the car for the two mesh methods.

In Figure 4.15 large differences for the boundary layer behavior at the front of the car can be seen as the prisms layers increased the mesh resolution. The boundary layer behavior for the prisms layer mesh was more realistic as larger velocity gradients were expected at the beginning of the boundary layer. It can also be seen that the behavior was smoother but also that the thickness was generally smaller than the mesh without prisms layers. Smoother behavior for the boundary layer could also be seen for the prisms layer mesh as a better mesh resolution was achieved for the wall bounded flow. However, at the rear of the car less difference could be seen on the boundary layer behavior for the
two meshes. This was due to two reasons, a finer surface mesh was used at the rear of the car which generated a better $y^*$ distribution and as the shark fin before the rear spoiler created a separation wake which caused similar behavior of the boundary layer.

The difference of mesh resolution can be seen in Figure 4.16 where the velocity contour was plotted at the rear spoiler of the V60 model. As a more constant first node height from the surface was obtained with the prisms layers the behavior of the boundary layer was not wavy as without the prisms layers. Different behavior of the separation for the V60 could also be seen as the flow seemed to stay more attached to rear spoiler than without prisms layers. This is often an effect of poorly resolved boundary layer.

![Figure 4.16](image)

Figure 4.16. Comparison of the velocity contours at the rear spoiler for the V60 for a mesh without prisms layers (left) and with prisms layers (right). Note the different behavior of the boundary layer as it behaves wavy for the mesh without prisms layers due to the varying first node distance from the surface.

In Figure 4.17 velocity profile comparison between the two meshes can be seen at the rear spoiler and wake for the V60 model. As could be seen in Figure 4.15 the boundary layer behaved similar at the rear spoiler which also can be seen in Figure 4.17. However, a noticeable difference could be seen for the rear wake as a flatter separation occurred for the prisms layer mesh compared to the mesh without prisms layers where the flow stays attached longer which resulted in different velocity profiles for the rear wake of the car models.

![Figure 4.17](image)

Figure 4.17. Velocity profile comparison between the mesh with and without prisms layers. No differences was caused between the different meshing methods at the rear spoiler but differences for the separation caused differences in the wake which can clearly be seen for the velocity profiles.

### 4.3.3 General mesh method and geometrical dependency

In order to ensure that the results from the prisms layer mesh method mainly was generated by the prisms layer and not from the thickened surfaces an independency study of the geometrical and surface mesh expansion influence was made. Four cases were considered which are presented in Table 4.7 were also the $\Delta C_D$, $\Delta C_{L_{f}}$ and $\Delta C_{L_{r}}$ against the AEDCAE02 method are presented.
Table 4.7. Differences for the aerodynamic force coefficients depending on the mesh method and created volumes for the front exterior and rear bumper.

<table>
<thead>
<tr>
<th>Mesh method</th>
<th>(\Delta C_D)</th>
<th>(\Delta C_Lf)</th>
<th>(\Delta C_{Lr})</th>
</tr>
</thead>
<tbody>
<tr>
<td>AEDCAE02</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>AEDCAE02 + Surface mesh expansion and exterior surface thickness</td>
<td>0.003</td>
<td>-0.007</td>
<td>0.023</td>
</tr>
<tr>
<td>AEDCAE02 + Prisms layers and exterior surface thickness</td>
<td>-0.004</td>
<td>-0.022</td>
<td>-0.010</td>
</tr>
<tr>
<td>AEDCAE02 + Surface mesh expansion + Prisms layers and exterior</td>
<td>-0.007</td>
<td>-0.027</td>
<td>-0.006</td>
</tr>
</tbody>
</table>

Seen in Table 4.7 significant differences occurred for the lift force distribution when the surface mesh expansion and surface thickness was added to the AEDCAE02 mesh method. Increased rear lift force and reduced front lift force was not favorable as it results in poorer correlation to the wind tunnel measurements. When only adding the surface thicknesses and prisms layers to the mesh method a large decrease for the front lift force was achieved but also a smaller decrease of the rear lift and drag force could be seen. However, these effects seems to add to each other when both the surface mesh expansion, surface thickness and prisms layers were added to the meshing method resulting in lower drag and lift forces.

Due to the limited space inside the engine bay it was important to investigate these geometrical and mesh dependence effects on the mass flow through the cooling package as the surface thickness decreased the engine bay space. In Table 4.8 the outflow of each component in the cooling package can be seen. In order to ensure that the obtained values were not dependent on the particular simulation the standard deviation were checked for some configurations where the geometrical and mesh settings were equal for the front of the car. A maximum standard deviation of 0.001 kg/s could be seen for all the components in the cooling package.

Table 4.8. Comparison on the flow through the cooling package for the different meshing methods. A noticeable difference can be seen for the flow through the condenser and radiator when the front and rear exterior surface thicknesses were added.

<table>
<thead>
<tr>
<th>Mesh method</th>
<th>Charged air cooler outflow [kg/s]</th>
<th>Condenser outflow [kg/s]</th>
<th>Radiator outflow [kg/s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>AEDCAE02</td>
<td>0.230</td>
<td>0.956</td>
<td>1.118</td>
</tr>
<tr>
<td>AEDCAE02 + Surface mesh expansion and exterior surface thickness</td>
<td>0.231</td>
<td>0.941</td>
<td>1.103</td>
</tr>
<tr>
<td>AEDCAE02 + Prisms layers and exterior surface thickness</td>
<td>0.230</td>
<td>0.946</td>
<td>1.107</td>
</tr>
<tr>
<td>AEDCAE02 + Surface mesh expansion + Prisms layers and exterior</td>
<td>0.233</td>
<td>0.949</td>
<td>1.112</td>
</tr>
</tbody>
</table>

Seen in Table 4.8 significant differences occurred for the outflow of the condenser and radiator when adding the surface mesh expansion and exterior surface thicknesses to the meshing method. However, when adding surface mesh expansion, prisms layer and exterior surface thicknesses the values approach the AEDCAE02 mesh methods values again.

The simulation results of the prisms layer mesh method was compared with results from CFD and wind tunnel measurements performed in [4] for both the considered car models and several configuration affecting both the underbody and body work.
4.4 Effects of detailed tyres

In the AEDCAE02 method slick tyre geometries which were morphed into the shape corresponding to the load of the car and rotational force was used. Effects on the lift force distribution with more detailed geometry of the tyres have been seen in [8]. The lift forces were especially affected by the detail level of the tyre grooves. In this thesis the 17 inches Continental Sport Contact 3 tyre was considered which were the tyres used for the models in the wind tunnel measurements. The simulation results were compared to CFD simulations with slick tyres and wind tunnel measurements in [4]. To only see the effects of detailed tyres the AEDCAE02 meshing method was used but with increased mesh refinement for the tyres to capture the complex geometry better.

The CAD model of the tyre was obtained from the tyre manufacturer (seen in Figure 4.18) and corresponded to the geometry without loads. Extra surfaces corresponding to manufacturing tools was also included in the CAD model but not used in the thesis. However, no side walls were included in the CAD model.

Figure 4. 18. The obtained CAD geometry from the tyre manufacturer. Note the round surfaces which corresponds to manufacturing tools.

As a tyre on a real car not is fully circular due to the loads of the vehicle and its rotation the tyre needed to be morphed by use of procedures created in [7] and [8]. The tyre morphing procedure needed the tyre to be fully geometrical cleaned and include side walls. The given CAD geometry from the tyre manufacturer included several geometry problems as surfaces was not fully connected and even some of them faulty. Only the outer surfaces which were in contact with the ambient air were left in the model after the cleaning. The used slick tyres were based on the geometry for the detailed tyre and therefore the side walls on the undeformed slick tyres fitted on the undeformed detailed tyres. The sidewalls were exported from a fully circular tyre and merged with the detailed tyre in order to achieve the full geometry of the tyres.

Before the morphing of the tyres the tyres needed to be positioned in the correct z-coordinate to correspond to the used curb height. In this thesis curb + 4 was used which corresponds to a loading of 4 people and a fully filled fuel tank.

To achieve the shape of the grooves that corresponded to the forces acting on the tyre the tyre morphing procedure created in [8] was used. However, as the tyres geometry differed at the grooves to
the morphed tyres in [8] some modifications of the tyre procedure was needed in order to achieve correct shape of the tyres.

Different PID dividing of the grooves was needed as grooves included a plateau which can be seen in Figure 4.19 and 4.24 colored in white. When the morphing procedure proposed in [8] was used these plateaus got removed during the morphing and was therefore needed to be divided into a separately PID so they could be morphed independent to the other parts of the tyre. The complete PID dividing can be seen in Figure 4.19.

![Figure 4.19. PID dividing for the tyre and its grooves to achieve correct shape after the morphing.](image)

The morphing procedure was done in ANSA and consisted of several morphing boxes which controlled the movement of the tyre. The tyre morphing procedure was created so the geometry of the tyre was preserved while the surface mesh was morphed into the wanted shape. All the morphing boxes can be seen in Figure 4.20.

The edge of the side wall which was in contact to the tyre was manually fitted to a separate morphing box which prevented it to move during the morphing procedure as the tyre rim contact not was influenced by the deformation of the tyre. The widest part of the tyre was also manually fitted to a morphing box in order to achieve the correct sidewall bulge of the tyre, seen in Figure 4.20. When morphing the tyre the surface mesh was loaded and connected to the morphing boxes which moved the surface mesh into the wanted position.

![Figure 4.20. The morphing boxes controlling the deformation of the tyres. In the lower right corner can the three morphing boxes used for morphing the contact patch be seen.](image)
Four morphing parameters were available to modify when using the morphing procedure, the radial expansion, vertical displacement, axial contraction and the sidewall bulge. The vertical displacement was used for moving the tyre into the correct position for the z-coordinate and was not used in this thesis as the tyres were correctly positioned before morphing. The used values for the morphing parameters are presented in Table 4.9 and were based on experimental values found in [7] for a V70 station wagon fitted with 16 inch wheel measured in static condition.

Table 4.9. Used parameter values for the tyre morphing in ANSA.

<table>
<thead>
<tr>
<th>Morphing parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Radial expansion</td>
<td>3</td>
</tr>
<tr>
<td>Vertical displacement</td>
<td>0</td>
</tr>
<tr>
<td>Axial contraction</td>
<td>3</td>
</tr>
<tr>
<td>Sidewall bulge</td>
<td>3.5</td>
</tr>
</tbody>
</table>

The same values for the parameters was used for the front and rear tyres. The radial expansion increased the radius of the tyre which corresponds to the centrifugal force due to the rotation. Due to the increased radius the width of the tyre decreases as more material was pressed out due to the centrifugal force which creates an axial contraction of the tyre. This can be seen in Figure 4.21 as the morphed tyre radius was slightly larger than the un-morphed tyre. The grooves for the morphed tyre were also more noticeable due to the axial contraction.

Figure 4.21. Comparison between an un-morphed tyre (left) and a morphed tyre (right). Note the slightly larger radius and more noticeable grooves for the morphed tyre due to the radial expansion and axial contraction due to the centrifugal force.

Due to the load of the vehicle a sidewall bulge occurred which also affected the tyre contact patch which can be seen in Figure 4.22.
Some assumptions were needed to be done during the morphing of the tyre, especially around the contact patch. The compression of the tyre pattern was assumed to be only 1 mm which corresponds to a 14.3% compression of the tyre pattern and was one of the assumptions made in the earlier study [8]. The plateau of the tyre pattern was assumed to have the same distance to the bottom of the tyre pattern before and after the morphing. The tyres rain grooves which had a radius at the bottom of the groove became rectangular at the contact patch as in [8].

The morphing was only done for the tyres on the left side of the car and was then copied and rotated to the right side of the car as the Continental Sport Contact 3 tyre only exist in no specific rolling direction configuration. Consideration for different shape for each tyre (due to different weight distribution on each tyre) was not considered as limited measurement data existed for each tyre.

The final mesh created in Harpoon can be seen in Figure 4.23 and consisted of a surface mesh size of 1.25 mm. This mesh size was just enough to capture the geometry of the detailed tyres as many details of the tyres were in the order of a few millimeters.
Simulations were also done with the prisms layer mesh method fitted with the detailed tyre for both the models in order to see if this caused any significant differences.

4.4.1 Special consideration for the groove plateau
A surface mesh size of 0.1 mm was needed in ANSA at the grooves with the plateau around the contact patch to prevent rough spikey sides of the grooves during the morphing of the surface mesh. This was due to the need of a different PID approach for the groove plateaus. However, the finer mesh size was just needed around these grooves to ensure better geometry representation of the surface mesh exported to Harpoon. The needed mesh refinement can be seen in Figure 4.24.

![Figure 4.24. Needed mesh size in ANSA before morphing in order to prevent spikey sides of the grooves after morphing the grooves at the contact patch.](image)

The different PID dividing was needed in order to capture the groove plateaus and ensure that no surfaces blocking the rain grooves occurred which could have happened with the PID dividing proposed in [8].

4.5 Mesh refinement for the engine bay and underbody
The importance of the underbody has been seen in many studies [10], [11] and [12]. In the AEDCAE02 mesh method 5 mm surface mesh size and 10 mm volume mesh size was used for the engine bay (except for the cooling package) and underbody. In order to see if significant difference on the lift distribution could be achieved with mesh refinements a mesh consisting of very refined underbody and engine bay was simulated. The mesh consisted of around 350 million cells and was only simulated for the baseline case of V60. Prisms layers were added for the exterior while surface mesh sizes of 1.25 and 2.5 mm was used for the underbody and certain parts of the engine bay. A significant difference could be seen for the drag and front lift force which corresponded better to the wind tunnel measurements. However, this mesh was only made to see if differences would occur and was not feasible to implement into the VCC CFD procedure due to greatly increased computational time (≈ 4 times longer than an AEDCAE02 simulation). If similar effects could be achieved with a mesh size around 200 million cells it would be more feasible to implement in the VCC CFD procedure if similar improved correlation could be obtained.

Therefore were tests performed with mesh refinements at the front grill and spoiler intake to see its effect on the total aerodynamic forces. Due to that high velocity flow occurred at this part of the vehicle it was important to investigate mesh refinement effects for this area. A mesh refinement box was added in front of the cooling package in the engine bay. The refinement box size were set to the
same size as the cooling package refinement box (1.25 mm) and thereby behaved as an extension of the cooling package refinement. Finer surface mesh (2.5 mm) was also set to the engine as high velocity flow passed around it and thereby was important to be resolve better. The use of this refinement box resulted in a slight increased drag and front lift force which improved the correlation to the wind tunnel.

In order to validate the results for prisms layers on tyres in [6] but also to ensure better y+ distribution for the tyres tests were performed with prisms layer on the tyres. A first layer thickness of 1 mm was used with a growth rate of 1.2 which resulted in six layers. This had a significant effect on the front lift force as it increase by almost 100 % (compared to the findings for the V60 baseline configuration for the prisms layer mesh method) and thereby corresponded better to the wind tunnel measurements. Also a slight increased drag and rear lift forces were obtained.

As both the refinement in front of the cooling package and prisms layers on the tyres resulted in better correlation to the wind tunnel measurements they were considered for this study. 2.5 mm surface mesh were added to the exhaust system, suspension, wheelhouses, parts of the drivetrain and all the underbody panels which could be seen to affect the flow in the previous performed simulation. Also a refinement box consisting of 5 mm cells were placed at the underbody to ensure good mesh resolution for the high speed flow at the underbody. A complete list of all the refined components can be found in Appendix A.

The increased mesh resolution for the engine bay and underbody can be seen in Figure 4.25. The mesh used for the refined engine bay and underbody. Note the mesh refinement between the cooling package and the grill and spoiler intake. Refinements for the underbody panels can especially be seen at the front of the car.

As the underbody was identical for the V60 and S60 only configurations for the V60 were considered as poorer correlation between simulations and wind tunnel measurement occurred for the V60 model. The prisms layer mesh method was used for the top exterior of the car as improved behavior of the wall bounded flow had been seen.

With the refined engine bay and underbody finer mesh resolution for the wall bounded flow at the underbody could be obtained which in turn generated a more suitable y+ distribution for the underbody which can be seen in Figure 4.26. The prisms layers on the tyres also helped to reduce the y+ to a more suitable level for the wall treatment as it was around five times lower than before use of prisms layers which can be seen in Figure 4.14.
Figure 4.26. $y^+$ distribution for the underbody for the refined engine bay and underbody mesh. Note the stair step behavior for the $y^+$ at the underbody panels as no prisms layers were used at the underbody.
5. Results and Discussion

In this section the achieved results from the developed methods are presented, discussed and compared to the findings in [4] for both the CFD simulations and wind tunnel measurements. The results are presented in the same order as in the Method section (Chapter 4).

5.1 Prisms layer generation

Due to two car models were considered and different flow behavior for the prisms layer mesh method could be seen for them the results and discussion are divided for each model.

5.1.1 V60

Due to that no prisms layers were considered for the underbody similar flow behavior for the underbody was expected for the AEDCAE02 and prisms layer mesh methods. However, seen in Figure 5.1 slight differences did occur for the front undershield as the AEDCAE02 mesh method seemed to cause spots of higher pressure at the start of the front undershield. This occurred due to the flow stayed more attached over the front undershield for the AEDCAE02 mesh method which caused higher pressure on some protruding details for the front undershield.

Slight difference also occurred at the engine undershield as an effect of the different flow over the front undershield. Higher pressure occurred on the engine undershield for the AEDCAE02 mesh method which was one of the reasons which resulted in higher front lift force than for the prisms layer mesh method.

Generally slight higher flow speed at the underbody occurred for the prisms layer mesh method as more flow seemed to pass under the car and thereby increased the flow velocity at the underbody. This explained the slight lowered pressure on the engine shield for the prisms layer mesh method which in turn caused the slightly higher mass flow out of the cool air charger seen in Table 4.8.

Large difference can be seen for the high pressure zone at the front side of the front tyres in Figure 5.1. This was due to a different flow separation occurred in front of the front wheels which created larger wakes around the tyres for the mesh method including prisms layers than for the AEDCAE02 mesh method. These wakes also generates a different flow to the rear of the car, as different size of the high pressure region occurred at the top of the front side of the rear tyres.

Figure 5.1. Surface pressure comparison for the underbody of the V60 baseline model for the AEDCAE02 mesh method (left) and the mesh method with prisms layers (right). Note the difference of the high pressure zone on the front wheels as
slight larger high pressure zones occurred for the prisms layer mesh method. Differences on the engine undershield could also be seen for the AEDCAE02 mesh method as higher pressure occurred which resulted in an increased front lift force.

The different flow around the front wheel could also be seen for the different surface pressure on the rims in Figure 5.2. Due to a more separated flow around the front wheels larger pressure wakes occurred around the front wheels which generated a lower pressure on the rims for the prisms layer mesh method. As the flow became more attached over the front wheels for the AEDCAE02 mesh method high pressure zones occurred at the rear of the front wheelhouses. Also a higher pressure at the middle of the rotating front rims occurred which seemed unreasonable.

Difference between the mesh methods could also be seen for the A-pillar vortices as stronger effect on the surface pressure could be seen for the prisms layer mesh method. This were the result of more separated flow around the A-pillars for the prisms layer mesh method due to the increased mesh resolution for the wall bounded flow. The A-pillar vortices also tended to follow the A-pillars more for the prisms layer mesh method than for the AEDCAE02 mesh method due to the stronger vortices and more interaction with the low pressure on top of the roof.

Figure 5.2. Surface pressure comparison seen at the top exterior for the V60 baseline model between the AEDCAE02 mesh method (left) and the prisms layer mesh method (right). Slight differences could be seen around the A-pillars as a more distinct effect of the A-pillar vortices occurred for the prisms layer mesh. Also a lower pressure at the rear part of the front wheelhouses could be seen as an effect of the larger wakes around the front wheels for the prisms layer mesh.

Significant differences occurred for the two considered mesh methods around the rear of the V60 model. Seen in Figure 5.3 lowered pressure occurred around the rear lamps for the AEDCAE02 mesh method as a result of attached flow around parts of the rear lamps. More distinct separation for the prisms layer mesh method could be seen as higher pressure was obtained for the rear part of the V60 model which reduced the drag force. The lower pressure on the rear lights also caused different lift force distribution for the rear of the car for the mesh methods. Due to higher pressure occurred at the rear of the car for the prisms layer mesh method a lowered rear lift force was achieved, reducing the over prediction of the rear lift force.

Similar pressure distribution at the rear of the rear tyres occurred for the mesh methods which were expected as no prisms layers were built on the tyres or the underbody.

Asymmetrical flow behavior could also be seen for the AEDCAE02 mesh method as different pressure distribution occurred between the left and right rear lamp. This phenomenon did not occur for the prisms layer mesh method and seems more reasonable as the major parts of the cars exterior was symmetrical. The flow behavior at the underbody was also reasonably symmetrical for both mesh methods which could be seen for the surface pressure distribution at the underbody in Figure 5.1.
For the study [12] which also used a prisms layer mesh method similar behavior for the rear pressure distribution to the prisms layer mesh method in this thesis could be seen for simulations performed for a SAAB 9-3 station wagon.

Figure 5.3. Surface pressure comparison for the rear of the V60 baseline for the AEDCAE02 mesh method (left) and the prisms layer mesh method (right). Clear differences for the pressure distribution for the rear lamps of the car can be seen between the mesh methods. Lower pressure occurred for the AEDCAE02 mesh method around the rear lights as the flow stayed more attached compared to the flow for the prisms layers mesh method where a distinct separation occurred for the rear.

As mentioned earlier different flow behavior occurred around the front wheels which clearly can be seen for the total pressure around the car for the two different mesh methods in Figure 5.4. The difference came from the flow stayed attached over the front wheels for the AEDCAE02 mesh method while the prisms layer mesh method caused separation around the front wheels.

Due to the separated flow before the front wheels occurred, front wheels wakes stretching all the way to the rear wake was generated for the prisms layer mesh method which not was obtained for the AEDCAE02 mesh method. However, wakes occurred for the AEDCAE02 mesh method but started after the front wheels and then grew into the rear wake. The prisms layer mesh method wakes seemed to correspond better to real world flow as fully attached flow over the front rotating wheels seemed unreasonable and hard to obtain in reality.

Wider wakes occurred for the AEDCAE02 mesh method than for the prisms layer mesh method as the rear rotating wheels greatly increased the growth of the wakes. This did not happen for the prisms layer mesh methods wakes as they already had become stable before they reached the rear wheels, thereby causing a smaller growth of the wakes.

Similar pressure distribution for the flow at the underbody could be seen for the mesh methods which were expected due to identical mesh settings were used for these parts and the engine bay. Differences for the left and right side of the cars could be seen for both mesh methods and may have been caused by the asymmetrical flow behavior inside the engine bay.
Figure 5.4. Total pressure coefficient comparison for the V60 baseline between AEDCAE02 mesh method (top) and the prisms layer mesh method (bottom) seen from above for a plane positioned at the wheel center. Note the different wake behavior between the mesh methods as the prisms layer mesh method caused flow separation at the front tyres.

Due to different separation occurred at the rear of the car for the two mesh methods different wake behavior can be seen in Figure 5.5. This happened due to different separation at the rear spoiler which can be seen in Figure 4.16 as the flow stayed more attached for the AEDCAE02 mesh method as a result of the poorer mesh resolution for the boundary layer. The earlier separation for the prisms layer mesh method generated a larger rear wake compared to the wake for the AEDCAE02 mesh method.

Slight thicker low pressure distribution occurred at the top of the spoiler as a result of better modeled boundary layer for the prisms layer mesh method which can be seen in Figure 5.5.
Figure 5.5. y = 0. Total pressure coefficient comparison between AEDCAE02 mesh method (top) and prisms layer mesh method (bottom). Note the different angle of the separation from the rear spoiler as the flow stayed more attached for the AEDCAE02 mesh method and thereby created a smaller but stronger low pressure wake compared to the prisms layer mesh method.

Even though the skin friction contribution to the total drag and lift forces may be small the skin friction could help to enlighten the different flow behavior for the mesh methods. For the front of the car significant differences for the skin friction can be seen for the different mesh methods in Figure 5.6. This was due to the prisms layer improved the behavior of the boundary layer with more suitable $y^*$ values and improved mesh resolution for the wall bounded flow. The prisms layer mesh method therefore captured the large velocity gradients in the beginning of the boundary layer which resulted in a better skin friction prediction.

Also in Figure 5.6 the clear different flow behavior over the front wheel can be seen as the low pressure wakes caused slower flow rear of the front wheels which in order generated lower skin friction for the prisms layer mesh method. This could be compared for the AEDCAE02 mesh where the flow stayed attached over the front wheels causing higher flow velocities near the wall and thereby higher skin friction after the front wheels. Big differences for the skin friction at the front wheels could also be seen as the wake created for the prisms mesh caused lowered skin friction.

Stronger effect from the A-pillar vortices were more clearly seen for the prisms layer mesh method as it increased the skin friction over the driver and passenger windows.

Ring patterns for the skin friction coefficient could be seen at the top of the roof and at the front for the AEDCAE02 mesh method as a result of Harpoon’s surface defining which caused varying first node height from the surface. This effect could also be seen for the prisms layer mesh method but only for the $y^*$ distribution (Figure 4.12 and 4.13) due to the smaller variation of the first node height.
Figure 5.6. Skin friction coefficient comparison between AEDCAE02 mesh method (left) and prisms layer mesh method (right) for the V60 baseline configuration. Great differences could be seen as the skin friction was better predicted for the prisms layer mesh method. Note the large difference between the meshes for the A-pillar vortices which caused higher skin friction for the prisms layer mesh method.

In order to see the effect of the skin friction contribution to the total forces the viscous forces were extracted from the solution and can be seen in Table 5.1. No significant difference could be seen for the drag force contribution but for the lift force contribution a larger percentage difference occurred between the meshing methods. However, as different flow field occurred around the car model different skin friction contribution was expected, especially as the flow tended to stay more attached which increased the skin friction contribution for the AEDCAE02 mesh method. Due to this larger differences was to be expected if more similar flow behavior would occur for the methods.

Table 5.1. Skin friction contribution to the drag force for the V60 baseline configuration.

<table>
<thead>
<tr>
<th>Mesh method</th>
<th>(C_D)</th>
<th>(C_L)</th>
</tr>
</thead>
<tbody>
<tr>
<td>AEDCAE02</td>
<td>11.05 %</td>
<td>1.12 %</td>
</tr>
<tr>
<td>Prisms layers</td>
<td>11.72 %</td>
<td>2.01 %</td>
</tr>
</tbody>
</table>

In order to see the different behavior of the pressure drag the total pressure equal to zero was illustrated with use of isosurfaces in Figure 5.7. Clear difference for the front wheels could be seen as the separation over the front wheel occurred before the front wheel for the prisms layer mesh method while it occurred at the front wheel for the AEDCAE02 mesh method.

Large difference could be seen at the rear low pressure wake as the attached flow at the rear lamps for the AEDCAE02 mesh method caused a smaller wake (also seen in Figure 5.5) compared to the prisms layer mesh method which created a larger wake due to more separated flow. Similar flow behavior could be seen for the rear wheels for the meshes.

The effects of the finer mesh resolution adjacent to surfaces could be seen for the total pressures behavior at the window mouldings and door handles as more separated flow was captured from the geometrical details compared to the AEDCAE02 mesh method. Different behavior of the total pressure at the roof of the car could be seen as the finer resolution of the wall bounded flow caused total pressure equal to zero at the roof and rails.
To locate where the largest difference for the total drag force occurred between the mesh methods the drag force coefficient was plotted against the x-coordinate in Figure 5.8. Similar drag force behavior could be seen until the front wheels as the different separation increased the drag force for the prisms layer mesh method. However, the AEDCAE02 mesh method obtained higher drag force after the front wheels due to the more attached flow but also due to a larger higher pressure zone of the front windscreen which can be seen in Figure 5.2.

Due to the more distinct separation at the rear of the car for the prisms layer mesh method obtained higher pressure at the rear surface of the car (seen in Figure 5.3) which resulted in lowered drag force.
the prisms layer mesh method as it caused a higher pressure distribution for the rear lamps which resulted in lowered rear lift force.

Figure 5.9. Accumulated lift force coefficient for the two considered mesh methods for the V60 baseline configuration. Note how the different flow behavior around the front wheels affected the lift force distribution.

5.1.2 S60
Large similarities to the flow behavior for the V60 model can be seen for the S60 model when the pressure distribution for the surfaces in Figure 5.10 were investigated. However, slight differences around the front wheels could be seen as a clear distinct separation not occurred in the same manner as for the V60 model for the prisms layer mesh method, despite identical geometry until the A-pillars. This difference occurred for all the simulated configurations for the S60 model which tends on a constant difference occurred for the mesh and thereby the flow behavior. As Harpoon used the octree approach the difference in the geometries could have triggered a different mesh generation behavior around the front wheels and thereby a different flow behavior. It does however indicate some robustness and mesh sensitivity problems for the prisms layer mesh method.

However, similar pressure distribution difference for the front side of the rear tyres can be seen when comparing the AEDCAE02 mesh method to the prisms layer method mesh in Figure 5.10 which also can be seen in Figure 5.2 for the V60 model. Due to the S60 model is a sedan model stronger A-pillar vortices occurred than for the V60 model and an increased effect on the surface pressure distribution could be seen for the prisms layer mesh method compared to the AEDCAE02 mesh method.
For the pressure distribution at the rear of the car seen in Figure 5.10 an opposite behavior to the V60 model can be seen as the flow stayed more attached with the prisms layer mesh method. Attached flow with the prisms layer mesh method seemed unreasonable due to a more distinct separation was expected and may therefore been a result of too poor mesh resolution in the boundary layer. The geometry was also rounder at the rear than for the V60 model which made it harder for the RANS model to predict the correct separation point. The V60 model also used a rear spoiler which triggered flow separation at the rear and thereby also triggered earlier flow separation around the D-pillars and rear lamps. The rear wakes of bluff bodies are also known to be highly unsteady [2] and [3] which can cause problems to capture correctly with use of steady-state simulations. The use of the isotropic eddy viscosity assumptions also made it hard for the realizable $k$-$\epsilon$ model to capture the full effects of the separation and wake as it consisted shear flow and flow recirculation.

Too attached flow is also a known problem for the $k$-$\epsilon$ models and can occur inconsistently if too coarse mesh resolution occurs in the boundary layer [5]. This may be the main reason why separated flow occurred for the V60 model but not for the S60 model for the prisms layer mesh method.

As 1.25 mm surface mesh was used at the rear only three prisms layers could be achieved for the rear part of the S60 model which made around five cells to cover the boundary layer. This did not meet the recommendations of at least ten cells to cover the boundary layer proposed in [5]. Tests were performed with a decreased first node height which slightly increased the number of cells inside the boundary layer but resulted in no improvements.

From the wind tunnel measurements performed in [4] more unstable flow behavior at the rear of the car could be seen for the S60 model compared to the V60 model which could make the steady state solution harder to be achieved. More stable wake behavior for the V60 model could also be expected as a more controlled separation occurred due to the use of the rear spoiler while the rounded rear of the S60 not triggers a clear separation point and which thereby increased the unstable flow behavior and trigger attached flow behavior. Separated flow was achieved without prisms layer which is similar results as in [19] compared to the prisms layers. This may be due to a poorer mesh resolution for the wall bounded flow can generate earlier separation compared to fine mesh [5].

Similar behavior was also seen for the prisms meshes created in newer version of Harpoon (5.4beta17) for the V60 model due to poor cell quality at the surface mesh size transition. This problem is more described in Appendix A.3.
Opposite to the prisms layer mesh methods behavior for the rear of the V60 model asymmetrical behavior for the prisms layer mesh method occurred for the S60 model. The reason for the asymmetrical behavior could be due to asymmetrical flow behavior for the underbody and rear tyres which may cause differences for the rear wake of the car as a more sensitive rear wake was discovered for the S60 model. Asymmetrical behavior for the S60 models rear window has also been seen in wind tunnel measurements [22] but was not obtained in the simulations.

Lower pressure on the top of the boot occurred for the AEDCAE02 mesh method compare to the prisms layer mesh method. This was one of the reasons why the rear lift force was less over predicted for the prisms layer mesh method.

![Figure 5.11. Pressure coefficient comparison for the S60 baseline configuration for the AEDCAE02 mesh method (left) and the prisms layer mesh method (right). Note that attached flow occurred around the rear lamps hence the lowered pressure for the prisms layer mesh method.](image)

Even though the flow stayed more attached around the rear lamps of the S60 model for the prisms layer mesh method less attached flow could be seen at the top edge for the boot compared to the AEDCAE02 mesh method. This resulted in a different separation angle for the two mesh methods which can be seen in Figure 5.12. The flow stayed more attached for the AEDCAE02 mesh method which resulted in a smaller but stronger rear wake.

The clearer separation seen for the prisms layer mesh method resulted in a higher pressure at the end edge of the boot which can be seen when comparing the pressure distribution at the rear edge of the boot for the mesh methods in Figure 5.11. The decreased pressure at the top of the boot was the main reason for the larger rear lift for the AEDCAE02 mesh method.
Figure 5.12. Total pressure coefficient for the S60 baseline configuration for the AEDCAE02 mesh method (top) and prisms layer method (bottom). Note the more attached flow at the rear of the car for the AEDCAE02 mesh method which caused a smaller wake than the prisms layer mesh method.

The different flow behavior for the mesh methods can clearly be seen in Figure 5.13 where the total pressure equal to zero was illustrated with an ISO-surface. As mentioned before strangely not the same behavior at the front wheels as seen for the V60 model (see Figure 5.7) could be seen for the S60 model as the flow stayed attached to the lower part of the front doors.

Similar to Figure 5.7 can also a more separated flow be seen around the window mouldings in Figure 5.13 for the prisms layer mesh as result of better prediction of the wall bounded flow behavior. Stronger behavior of the A-pillar vortices could also be seen for the prisms layer mesh method as it interacted more with the low pressure flow on the roof.

Seen in Figure 5.12 a more distinct separation occurred for the prisms layer mesh at the rear edge of the boot which also can be seen in Figure 5.13, as more flat behavior of the total pressure could be seen for the prisms mesh method compared to the AEDCAE02 mesh method. Less effect of the shark fin antenna could also be seen for the prisms mesh method as it caused more attached flow at the rear edge of the boot for the AEDCAE02 mesh method, as it in some manners worked as a vortex generator and thereby increased the energy in the flow, causing a more attached flow behavior.

Due to the different flow behavior at the front wheel the flow at the rear wheel got heavily affected for the S60 model which affected the rear wake behavior. Due to the more separated flow at the front wheel for the prisms mesh method the flow around the rear wheel was more separated which caused a wider behavior of the rear wake compared to the AEDCAE02 mesh method where the flow stayed more attached around the rear wheels.
Figure 5.13. Total pressure equal to zero for the S60 baseline configuration seen for the AEDCAE02 mesh method (left) and the prisms layer method (right). Similar flow behavior around the front tyre can be seen which not correlated to the behavior seen for the V60 model. Note the different separation for the rear of the car for the different mesh methods.

For the skin friction coefficient seen in Figure 5.14 more attached flow behavior around the front wheels could be seen for the prisms layer mesh method which not can be seen for the V60 model in Figure 5.6. The obtained behavior for the S60 model with the prisms layer method behaved more similar to the AEDCAE02 mesh method which not was expected.

Figure 5.14. Skin friction coefficient for the S60 baseline configuration for the AEDCAE02 mesh method (left) and the prisms layer mesh method (right). Note the more similar behavior of the skin friction around the front wheel which not occurred for the V60 model.

Almost identical values for the skin friction contribution to the drag force can be seen for the S60 model in Table 5.2 as seen for the V60 model in Table 5.1. Larger difference was expected between the two mesh methods as lower pressure drag was obtained for the S60 model due to the smaller rear wake. Also more similarities in the flow behavior could be seen for the two mesh methods which showed the lower importance of predicting the skin friction more correctly.

Table 5.2. Skin friction contribution to the total drag and lift forces for the S60 baseline configuration for the AEDCAE02 and prisms layer mesh methods.

<table>
<thead>
<tr>
<th>Mesh method</th>
<th>$C_D$ %</th>
<th>$C_L$ %</th>
</tr>
</thead>
<tbody>
<tr>
<td>AEDCAE02</td>
<td>11.09 %</td>
<td>0.82 %</td>
</tr>
<tr>
<td>Prisms layers</td>
<td>11.76 %</td>
<td>1.11 %</td>
</tr>
</tbody>
</table>

Compared to the V60 model differences between the mesh methods for the accumulated drag occurred earlier as difference occurred in front of the front wheels which can be seen in Figure 5.15. Differences around the front wheel occurred due to the separation for the prisms layer mesh method compared to the AEDCAE02 mesh method. Different to the behavior for the V60 model no differences occurred between the mesh methods at the rear wake. This was due to the flow stayed
more attached for the rear of the car for the prisms layer mesh method which lowered the pressure at the rear of the car and thereby increased the drag force.

Figure 5.15. Accumulated drag force coefficient for the two considered mesh methods for the S60 baseline configuration. Note the large difference between the two mesh methods at the front wheels as a larger wake for the prisms layer mesh method occurred.

Also for the accumulated lift force coefficient similar behavior to the V60 model can be seen for the S60 model in Figure 5.16. Difference between the mesh methods could be seen to occur at the front wheels and around the front windscreen. A larger difference than seen for the V60 model could be seen at the rear wake for the S60 model as a result of the higher pressure on the top of the boot (seen in Figure 5.11) for the prisms layer mesh method.

Figure 5.16. Accumulated lift force coefficient for the two considered mesh methods for the S60 baseline configuration. Note the difference between the two mesh methods around the front wheels and at the rear wake.

5.1.3 Force comparison between the mesh methods and wind tunnel measurements
Generally poorer correlation to the wind tunnel measurements were obtained for the drag force with the prisms layer mesh method. The reason for the V60 model was due to the more distinct separation at the rear of the car which caused a higher pressure at the rear of the car which lowered the drag force.
compared to the more attached flow for the AEDCAE02 mesh method. As more attached flow occurred when the prisms layer mesh method were used on the S60 model somehow similar correlation as for AEDCAE02 occurred. Better correlation could be seen for the prisms layer mesh method on the V60 and S60 model for the underbody configuration concerning rear of the car. This was due to more disturbed flow occurred at the underbody and hence stronger rear wake which increased the attached flow around the rear lamps for both the models and thereby increased the drag force. Perfect correlation could be seen for the prisms layer mesh method for the engine undershield (EUS) configuration for the S60 model but was more a coincidence than correct obtained flow field.

Figure 5.17. Drag force coefficient delta values to the wind tunnel measurements for the considered mesh methods and configurations of the car models. Note that positive values corresponds to an under prediction of the drag force compared to the wind tunnel measurements.

In [1] increased drag could be seen for a S60 model when it was simulated inside the VCC wind tunnel due to the larger front area blocking ratio at the test section. Different boundary conditions also existed between the simulated CFD domain and the wind tunnel as road like conditions were the main objective with the CFD simulation, not to replicate the behavior in the wind tunnel. Also the geometrical differences between the CAD model and the wind tunnel test model could cause reasons for these differences. For example the front undershield positioning could have caused different flow behavior of the underbody which could lead to significant differences for the aerodynamic forces.

Generally more fluctuations and instabilities could be seen for the front and rear lift forces compared to the drag force during the simulations as they were more sensitive than the drag force for minor flow field changes during the solution. The lift forces are generally hard to capture properly for both CFD simulations and wind tunnel measurements as the lift force is affected by flow over the large areas on the top and bottom surfaces of the cars. The drag force is less sensitive as it mainly is affected by the front and rear surfaces of the cars.

Due to the increased mass flow underneath the cars, lowered the static pressure for the prisms layer mesh method lowered front lift force was generally obtained which decreased the correlation to the wind tunnel as can be seen in Figure 5.18. The increased separation around the front wheels for the prisms layer mesh method also reduced the front lift force and thereby also the correlation to the wind tunnel measurements. Interestingly similar correlation between the mesh methods was obtained for the configuration without front undershield (FUS) which increased the pressure on the engine undershield (EUS) and thereby more similar pressure distribution was obtained at the underbody for the mesh
methods. This showed the importance of the front undershield as it greatly affected the flow behavior for the underbody.

Due to the larger front area block ratio in the test section of the wind tunnel more increased velocity occurred at the front of the car than in the CFD domain. This increased flow velocity increased the mass flow through the engine bay which was believed to increase the pressure inside the engine bay causing increased front lift prediction in the wind tunnel measurements. This could be one of the main reasons why poor correlation generally could be seen for the front lift force.

Better correlation for the rear lift was achieved for all the geometrical configurations with the prisms layer mesh method. This was due to the increased pressure at the rear lamps and rear spoiler due to the more distinct separation for the V60 model as can be seen in Figure 5.3. For the S60 model generally higher pressure was obtained at the top of the rear boot which decreased the rear lift and increased the correlation to the wind tunnel measurements. Less difference between the mesh methods were expected for the configuration with separation edge on the D-pillar (SED) as a more distinct separation for the AEDCAE02 mesh method occurred.

Figure 5.18. Front lift force coefficient delta values to the wind tunnel measurements for the considered mesh methods and geometrical configurations of the car models. Note the general high under prediction of the front lift for the vehicle.
Figure 5.19. Rear lift force coefficient delta values to the wind tunnel measurements for the considered mesh methods and configurations of the car models. Note the achieved better correlation for all the configurations to the wind tunnel measurements with the prisms layer mesh method.

In order to see if similar increase or decrease of the aerodynamic forces occurred for the different configurations the aerodynamic force trends are compared for the both considered mesh methods and the wind tunnel measurements. The force trends are defined by the configuration coefficient value subtracted the baseline configuration coefficient value.

In Figure 5.20 the drag force coefficient trends can be seen for all the considered geometries. Poorer correlation could in general be seen for the prisms layer mesh method when compared to the AEDCAE02 mesh method. Interestingly the best trend correlation occurred when the engine undershield was removed which tends to show that the front underbody panels affects the aerodynamic forces greatly.

Poor drag force correlation could especially be seen for the underbody configurations affecting the rear wake greatly. This was due to more disturbed flow occurred for the underbody which affected the wakes behavior greatly. The wake often became stronger which increased the attached flow around the rear lamps which in turn increased the drag force greatly. If the distinct separations still had occurred for the V60 model improved correlation to the wind tunnel measurements was expected for the prisms layer mesh method. This theory was supported by the improved correlation for the S60 model as attached flow already occurred for the baseline configuration with the prisms layer mesh method.

Therefore the increased drag force was not in the same magnitude as without prisms layers and thereby corresponded better to the wind tunnel measurement.

The differences seen for the rear exterior configurations were so small that the difference most likely came from the numerical error. Better correlation for AEDCAE02 mesh method was expected for the separation edge at the D-pillars at it decreased the drag due to a more distinct separation similar to the prisms layer mesh method.
Figure 5.20. Drag force coefficient trends for the different mesh methods and wind tunnel measurements for the considered car models and geometrical configurations.

Seen in [1] the boundary conditions around the tyres in the wind tunnel also affected the total lift force value for the cars. As previous explained the purpose of the CFD simulations is not to reproduce the flow behavior in the wind tunnel hence the different boundary conditions could have caused these significant differences for the lift force values but also for the lift force balance of the car.

Generally better trend correlation can be seen in Figure 5.21 for the front lift force for the geometrical configurations affecting the front of the underbody. This was not expected as poorer correlation for the delta values against the wind tunnel measurements was seen in Figure 5.18. Better trends for the front lift force could especially be seen for the front underbody configurations. As different flow behavior occurred over the front undershield its effect for the total front lift force was seen to be great. This could also be seen for the configuration where the front undershield was removed where a completely different trend was achieved for the two mesh methods for the V60 model. The different trend for the prisms layer mesh corresponded much better to the trend seen for the wind tunnel measurements, for both car models. As the effect of the different flow over the front undershield also affected the flow over the engine undershield better correlation against the wind tunnel measurements was achieved for both the car models with the prisms layer mesh method.

The differences seen for the configurations affecting the rear part of the cars was so small that the numerical errors risked causing wrong trends.
Figure 5.21. Front lift force coefficient trends for the different mesh methods and wind tunnel measurements for the considered car models and geometrical configurations. Note the improved correlation for the engine undershield.

A generally better rear lift force trend prediction for the S60 model with the prisms layer mesh method can be seen in Figure 5.22. For the V60 model mainly the underbody configurations affecting the rear of the underbody seemed to correlate better with the wind tunnel measurements for the prisms layer mesh method. This was due to the rear lift increased due to the more attached flow around the rear lamps which decreases the pressure on the lamps and thereby increases the rear lift force. Similar behavior could be seen for the S60 model.

The front underbody configuration seemed to affect the rear lift force more in the simulations than for the wind tunnel experiments. This was due to the flow stayed more attached for the lower part of the rear bumper which reduced the pressure on it and thereby the rear lift force. This behavior was believed to not have occurred for the wind tunnel measurements as over attachment tends to happen for the turbulence model used in the simulations. It does however tend to show that the rear wake behavior for both the car models are sensitive for the flow behavior at the underbody as the underbody flow got more disturbed with the underbody configurations.

Figure 5.22. Rear lift force coefficient trends for the different mesh methods and wind tunnel measurements for the considered car models and geometrical configurations. Note the generally better correlation for the configurations affecting the rear underbody.
5.1.4 General discussion about the prisms layer mesh method

Due to Harpoon only could build prisms layers on both sides of surfaces extra work was required in ANSA in order to prepare the model for prisms mesh generation. This extra CAD work to create the thickened surfaces could for some geometries be challenging to achieve correctly and could sometimes require several iterations before acceptable geometrical quality was achieved. The thickness of the surfaces was important to keep larger than the used surface mesh size in order to enable Harpoon to successfully keep the thickness after meshing. Due to surfaces penetrating the thickened surface could cause problems increased CAD geometry cleaning was needed which in general not is favorable among engineers.

The thickened surfaces had a surprisingly large impact on the overall aerodynamic forces and especially for the rear lift force which increased with over 20 drag counts (0.020) for the V60 model which can be seen in Table 4.7. This unfortunately included some uncertainties into the results for the prisms layer mesh methods. However, the use of the prisms layer mesh method clearly removed the increased rear lift force but poses the question if the rear lift force would been decreased even more and thereby improved the correlation more if the need of thickened surfaces did not exist. Tests were performed for a newer version of Harpoon (5.4beta17) where prisms layer could be built on a single side of surfaces. However, problems with poor mesh quality at the prisms layer showed immaturity problems in the new version. See Appendix A.3 for further details.

Minor geometrical changes were needed to make the geometries suitable for prisms generation. These modifications were so small that no significant differences to the flow behavior was expected and could not be seen in the results.

Due to the method was developed on CAD models for production cars a lot of more effort was needed in order to make the geometry suitable for prisms layer generation. It did however help to increase the robustness of the thicken surface method.

Even though prisms layers only were built on the exterior problems with mesh quality did occur. High cell skewness caused a lowered average mesh quality for the prisms layer meshes. Often few cells were so skewed that negative volumes occurred. The negative volumes were however removed from the mesh inside Ansys Fluent but the lowered mesh quality should be considered as it could influence the results. Several more mesh smoothing iterations were needed to be used in Harpoon and Ansys Fluent in order to improve the mesh to similar mesh quality level as for the AEDCAE02 mesh method.

The prisms layers did however achieve a better modeled boundary layer which seems to behave more realistic compared to the boundary layer obtained with the current mesh method AEDCAE02. Seen in Figure 4.15 the boundary layer thickness growth with the distance of the car for the prisms layer mesh method seemed to better correspond to boundary layer theory described in Section 3.1. The boundary layer thickness for the AEDCAE02 mesh method behaved almost constant over the whole car. This occurred due to around three times more cells covered the boundary layer for the prisms layer mesh method compared to the AEDCAE02 mesh method.

More controlled first node height from the surface can also be seen for the prisms layer mesh method in Figure 4.16 which resulted in a less wavy behavior for the boundary layer which improved the correlation to physical flow behavior.

Better capturing of the large velocity gradients can be seen with the prisms mesh method in Figure 4.15 and was the reason for the better predicted skin friction for the cars. This was achieved as the prisms layer mesh method increased the number of cells inside the boundary from 2-3 cells to around...
6 cells for major parts of the cars. It can also be seen in Figure 4.16 that the prisms layers mesh more controlled first node height from the surface generated more realistic behavior of the boundary layer, as it otherwise could behave unphysical wavy as the first node height varied. However, more cells in the boundary were deemed to be needed due to the inconsistent behavior for the rear wake of the cars. Even though correct flow field may not be achieved with more cells inside the boundary layer consistency of the flow behavior would be expected. With consistent behavior it may then be possible to tune in the turbulence model to achieve better correlation with wind tunnel measurements.

Due to Harpoon builds the prisms layer cells after the initial mesh and then removes the cells at and adjacent to the surfaces the space for the prisms layer cells gets limited. Due to this, it was not possible to generate more prisms layer cells to increase the mesh resolution in the boundary layer with the currently used surface mesh sizes. It is no need to only cover the boundary layer with the prisms layer cells as the bulk mesh cells also can cover it. However, the idea with prisms layers is to use the high aspect ratio cells to be able to use larger surface mesh size and thereby not increase the mesh size greatly. But due to the limited number of prisms layers a finer surface mesh size would be needed in order to increase the number of cells inside the boundary layer in Harpoon.

As the first layer thickness will be needed to be decreased to meet this requirement the y⁺ distribution will be changed. y⁺ values below 30 may not be suitable for the Standard Wall Function in Ansys Fluent as it could cause accuracy problems. By instead using Enhanced Wall Treatment the y⁺ value would be less sensitive for the mesh resolution as it uses a two-layer zonal mode to model the boundary layer behavior.

Minor difference could be seen when the surface mesh expansion was used as it caused smoother growth of the mesh refinement in the domain. It did however increase the robustness of the meshing method as the risk for the mesh to direct the flow decreased. However, in Figure 5.4 it can be seen that the rear refinement boxes limited the growth of the rear wake at the sides. A simulation was performed where a wider refinement box were used at the rear of the cars but did not show any differences in the results.

No differences occurred for the main shape between the detailed and slick tyres as only the grooves made the differences of the geometries. No consideration of a wider side bulge on the inside of the tyres was considered during the morphing procedure which caused a slight difference to the real geometrical shape of the tyres. This effect of the shape difference was however assumed to be small on the total aerodynamic forces.

5.2 Effects of detailed tyres
Detailed tyres where simulated for both the S60 (sedan) and the V60 (sportswagen) for a number of different configurations. Similar flow behaviors were seen for the V60 and S60 and are therefore presented together.

Similar pressure distribution at the front side of the front tyres for the detailed and slick tyres occurred even though pressure leakages through the rain grooves were expected to influence the detailed tyres. However, smaller differences can be seen for the front side of the rear tyre as a slightly higher pressure occurred at the top outer corner for the slick tyres compared to the detailed tyres seen in Figure 5.23. This was due to the flow separated at the front wheels for the detailed tyres which can be seen in Figure 5.24 compared to the slick tyres which separation occurred after the front wheels.

Slight lower pressure also occurred at the lower part of rear detailed tyres due to the flow could pass through the rain grooves, causing pressure leakage for the front side of the tyres.
For the underbody and wheelhouses no significant differences for the surface pressure could be seen between the different tyre models.

![Image](image1.png)

Figure 5.23. Similar behavior can be seen for the pressure distribution at the underbody and front side of the wheels for the slick and detailed tyres.

Detached flow behavior could be seen for the detailed tyres fitted to both the car models as the grooves of the tyres caused more disturbances to the flow. A finer surface mesh size (1.25 mm) for the detailed tyres may also have triggered the separation compared to the slick tyres which had a coarser mesh size (5 mm). Similar behavior was seen for simulations with detailed tyres in [8] but as the same surface mesh sizes were used in that particular study no conclusions about what triggered the separation can be done with absolute certainties.

![Image](image2.png)

Figure 5.24. Total pressure coefficient for the V60 baseline configuration with detailed tyres fitted to it. Slight differences to the slick tyres can be seen when comparing to Figure 5.4 as the flow separates more similar to the prisms layer mesh method. Similar behavior occurred for the S60 model.

Minor differences can be seen for the total pressure distribution in Figure 5.25 where a plane was placed directly after the front wheels. The differences occurred on the side of the car as the detailed tyres and finer mesh caused more disturbances to the flow which caused the earlier separation and larger lower pressure zones on the side of the cars. This behavior could also be seen in [8] for simulations of 16 inch wheels fitted with detailed tyres for the same S60 model. The pressure distribution also corresponded better to pressure measurements performed in the VCC wind tunnel for
the S60 model [8] as stronger connection between the upper and lower separation wakes for the front wheels existed.

No major differences were expected between the slick and detailed tyres as it already have been seen in studies [7] and [8] that the shape of the tyre had greater influence on the flow field and the total aerodynamic forces than the detail level of the tyre model.

![Image](image1.png)

Figure 5.25. Only minor differences can be seen between the slick (left) and detailed (right) tyres for the S60 baseline configuration seen at plane located at the end of the front tyre (wheel center + 320 mm). Slight larger pressure wakes occurred around the detailed tyres compared to the slick tyres. Similar behavior for both considered car models.

The larger front wheel wakes for the detailed tyres compared to the slick tyres can also be seen for the total pressure equal to zero for the S60 model in Figure 5.26. Discussed in [8] the detailed tyres caused more disturbances to the flow which in turned caused a larger rear wake of the car which also was the case in this study and can be seen when the lower part of the rear wakes were compared in Figure 5.26.

![Image](image2.png)

Figure 5.26. Isosurface showing total pressure equal to zero for the S60 model fitted with slick tyres (left) and detailed tyres (right). Note the larger wakes from the wheels for the detailed tyre model compared to the slick tyres.

The different tyre models caused different surface pressure distribution for the rear of the car models. Stronger effects could be seen for the S60 model as the rear wake seemed to be more sensitive for disturbed flow at the underbody. In Figure 5.27 can lower pressure be seen for the lower bumper for the S60 fitted with detailed tyres compared to the model fitted with slick tyres. This was due to the more disturbed flow caused by the tyre grooves which caused more attached flow for the rear bumper. This also caused more attached flow around the top part of the rear lamps as the rear wake became
stronger with the more attached flow with the detailed tyre model. Similar but not as strong behavior could be seen for the V60 model fitted with the detailed tyres.

Figure 5.27. Pressure coefficient seen for the S60 baseline with slick tyres (left) and with detailed tyres (right). Similar behavior can be seen for the rear of the different tyre models but due to the more disturbed flow caused by the detailed tyres a stronger wake occurred which causes more attached flow rear bumper and lamps as lowered pressure occurred.

No significant difference could be seen for the accumulated drag coefficient comparison between the slick and detailed tyre models (even though different flow around the front wheel occurred) fitted on the V60 and S60 models and were therefore not presented in this thesis. However, noticeable differences occurred for the accumulated lift coefficient for the different tyre models fitted on the V60 model which can be seen in Figure 5.28. The differences occurred at the rear of the car due to lower pressure on the rear bumper due to the more attached flow with the detailed tyres. Also the different pressure distribution on the front side of the rear tyres caused a lowered lift force for the rear of the car models. Slightly lowered lift after the front wheels also occurred for the detailed tyre as an effect of the earlier separation at the side of the cars, seen in Figure 5.24.

Figure 5.28. Accumulated lift force coefficient comparison between the slick and detailed tyres for the S60 baseline configuration. Due to the more disturbed flow caused by the detailed tyre model lowered lift occurred, especially for the rear of the car.

No direct trend of improved or worsened correlation can be seen in Figure 5.29 for the total drag forces for the car models fitted with the detailed tyre model. For the baseline configuration slight
decreased correlation could be seen for the detailed tyre compared to the slick tyre. However, the differences were so small that the worsen correlation could been from the numerical error. Slight improved correlation could be seen for the closed cooling and rim cover configurations. As more air got directed around the car due to the closed intake to the engine bay more of the flow passed by the tyres. As can be seen in Figure 5.25 and 5.26 larger wheel wakes occurred for the detailed tyres which increased the drag force, hence improving the correlation to the wind tunnel measurements. Seen in [8] the level of details for the tyres affected the performance of the rim design. This effect could also be seen in this study as better correlation was achieved for rim covers configuration fitted with the detailed tyre.

![Figure 5.29](image)

Figure 5.29. Drag force coefficient delta to the wind tunnel measurements for the considered tyre models and configurations of the car models. Note that positive values corresponds to an under prediction of the drag force compared to the wind tunnel measurements.

Generally poorer correlation between simulation and wind tunnel measurements for the front lift force can be seen in Figure 5.30 when the detailed tyre model was used. The decreased front lift was due to the earlier created wake around the front wheels occurred for the detailed tyre, seen in Figure 5.24. Less difference could be seen for the S60 model as the detailed tyre model had less effect on the flow behavior compared to the V60 model, also was seen in [8].
Figure 5.30. Front lift force coefficient delta to the wind tunnel measurements for the considered tyre models and configurations of the car models. Generally poorer correlation for the detailed tyres can be seen. Note that positive values corresponds to an under prediction of the front lift force compared to the wind tunnel measurements.

Reduced rear lift can generally be seen as an effect of the detailed tyres in Figure 5.31. This was due to the earlier discussed lowered pressure on the front side of the rear tyres and the more attached flow around the rear bumper. Less effect on the rear lift force could be seen for the S60 model compared to the V60 model when the detailed tyres were fitted. This was due detailed tyres effect was enlarged by the rear wake of the cars and due to the larger wake for the V60 model the effects became stronger [8].

Figure 5.31. Rear lift force coefficient delta to the wind tunnel measurements for the considered tyre models and configurations of the car models. Generally improved correlation for the detailed tyres can be seen.

Improved correlation for the drag force trend for different configurations can be seen in Figure 5.32. The importance of including details of the tyres for studies of rims design can clearly be seen as better trend correlation to the wind tunnel measurements could be seen with the detailed tyres fitted on both car models. This also corresponded well to the conclusions in [8]. Slight worsened correlation could be seen for the underbody configuration of the S60 model due to the more sensitive rear wake. As the detailed tyres disturbed the flow even more the effects of the underbody configurations were increased which caused more attached flow around the rear lamps which resulted in some over prediction for the
drag force trend. However, this behavior improved the trend correlation for the V60 model when both
the tank panels (TP LH & TP RH) were removed.

![Figure 5.32] Drag force trends for different configuration and car models fitted with the detailed and slick
tyre models. Note the generally improved trend correlation with the detailed tyres.

Even though the car models are identical until the A-pillars differences between the trend correlations
between the models can be seen for the front lift force in Figure 5.33. A generally better trend
correlation could be seen for the V60 model while the effects of the detailed tyres not affected the S60
model in the same manner, also seen for the other forces.

![Figure 5.33] Front lift force trends for different configuration and car models fitted with the detailed and slick tyre models. Note the generally worsened trend correlation for the S60 model but not for the V60 model.

Worsened rear lift force trend correlation for the detailed tyre can in general be seen for the V60
model in Figure 5.34. However, slight better trend correlation could be seen for the S60 model which
not was expected as less effect of the detailed tyres for the rear lift force was seen in Figure 5.31.

The worsened correlation for the closed cooling configuration was obtained from more air passing
under and above the car and as more disturbed flow was caused by the detailed tyre more attached
flow around the rear of the cars occurred which in turned increased the rear lift greatly. As this over
attached flow not was expected to occur in reality it was one of the major reasons for the over predicated rear lift force.

Figure 5.34. Rear lift force trends for different configuration and car models fitted with the detailed and slick tyre models.

5.2.1 General discussion about the detailed tyres

The developed morphing procedure in [8] needed to be modified to handle the different tyre groove geometry for the tyre considered in this thesis. With the updated morphing procedure it did handle a wider sort of tyre grooves which made the procedure more robust for different geometries. The modification added a few steps which not increased the procedures time to be perform severely.

Generally the detailed tyre disturbed the flow more which resulted in the earlier separation around the front wheels but also in the different rear wake behavior for both models. It could be seen that the rear wake was more sensitive on the S60 model as the detailed tyres generally increased the drag force more for the underbody configurations.

Bear in mind that the differences in the results may not only come from the detail level of the tyres but also from the finer surface mesh as it affects the volume mesh around the wheels. The finer volume mesh resulted in improved prediction of the wall bounded flow which may cause differences to the results. To be fully certain the results originates from the tyre detail level simulation may need to be performed with a refined surface mesh for the slick tyres. Test were however performed with prisms layers on the tyres which improved the mesh resolution for the wall bounded flow and show significant difference which not correlated to the results for the detailed tyres. This may in some manners indicate that the results most likely come from the detail level of the tyres.

The grooves of the tyre made the flow around the tyre dependent on the rotational position of the tyre which could not be captured by use of steady-state simulations. To capture the effect transient simulation technique and use of sliding mesh would be needed which would have increased the simulation time greatly. However, in contrast to the rims (which geometry also was rotational position dependent) the details were so small that only minor differences would be expected with consideration of the rotational position.

The different effect of the detailed tyre for the car models enlightened the importance of testing new simulation methods on several different geometries before conclusions about the new methods could be drawn.
5.3 Detailed tyres and prisms layer mesh

Detailed tyres combined with prisms layer mesh method were simulated for the baseline case for the V60 and S60 models. Only the baseline configurations were simulated as no significant differences could be seen compared to simulations performed with the prisms layer mesh method and slick tyres.

Compared to the V60 model fitted with slick tyres and mesh with prisms layer mesh method only minor differences can be seen for the front wheel wake and the wake at the rear in Figure 5.35. The larger front wake was an effect of the earlier separation caused by the improved mesh resolution in the boundary layer and the more disturbed flow caused by the grooves of the detailed tyres.

![Figure 5.35. Larger front wheel wakes occurred compared to only the prisms layer mesh method. Also more effect on the rear wake could be seen as the lower part of the rear wake was larger for the detailed tyre mesh compared to prisms layer mesh method.](image)

Slight different behavior around the rear lamps occurred when the detailed tyres were fitted to the prisms layer mesh method. More disturbed flow at the underbody caused by the grooves of the tyres generated more attached flow around the rear lamps which caused a lowered pressure at the rear of the car, seen in Figure 5.36. As attached flow already occurred for the S60 models rear lamps only slight differences could be seen for the pressure distribution at the rear of the car.

![Figure 5.36. The V60 baseline configuration fitted with detailed tyres and meshed with the prisms layer mesh method. Note that the stronger wake behavior caused more attached flow around the rear lamps which lowered the pressure.](image)

Similar drag force build up can be seen for the V60 model mesh with the two mesh methods, the model meshed with the prisms layer method and fitted with detailed tyres, in Figure 5.37. However,
slight difference at the rear wake could be seen for the prisms layer mesh method fitted with slick and detailed tyres due to the lowered pressure at the lamps.

![Figure 5.37. Drag force comparison between the different meshing methods and added detailed tyres. Similar behavior occurred for the S60 model.](image)

Almost identical behavior for the lift force build up can be seen for the prisms layer mesh method fitted with slick and detailed tyres seen in Figure 5.38. Slight differences after the front wheel occurred due to the larger wakes around the front wheels. These wakes also affected the pressure distribution at the rear tyres which caused a slight difference for the pressure distribution hence the rear lift force of the car. Also the lowered pressure around the rear lamps contributed to slight higher lift force at the rear of the cars.

![Figure 5.38. Comparison between the mesh methods and added detailed tyres. Identical behavior can be seen for the S60 except that lowered lift occurred at the rear of the car with the detailed tyres.](image)
5.4 Mesh refinement for engine bay and underbody

The simulations performed with refinements added in front of the cooling package improved the correlation to the wind tunnel measurements for all aerodynamic forces as the $C_D$ increased with 0.004, the $C_{Lf}$ with 0.005 and the $C_{Lr}$ decreased with 0.004. This was due the increased mesh resolution better resolved the high velocity flow through the grill and spoiler intake and through the cooling package. However, no significant differences could be seen for the mass flow outflow through the cooling package components when compared to the results from the prisms layer mesh method.

The simulations with added prisms layers on the tyres showed a dramatically increase of $C_{Lf}$ as it increased with 0.012. Also the $C_D$ increased with 0.003 while the $C_{Lr}$ increased with 0.010. This improved the front lift force correlation greatly while the rear lift force correlation was slightly worsened.

However, for the final method slight differences can be seen in Figure 5.39 for the total pressure equal to zero when the engine bay and underbody was refined. Due to the added prisms layers on the tyres a larger separation wake occurred around the front wheels which was more similar to the behavior seen for the detailed tyre combined with the prisms layer mesh method. Also a larger rear wake occurred as the lower part of the rear wake was enlarged. This happened due to different separation occurred around the rear wheels due to the prisms layers which increased the mesh resolution for the wall bounded flow.

![Figure 5.39. The refined engine bay and underbody mesh method for the baseline V60 configuration. Note the larger separation around the front tyre compared to only the prisms layer mesh method. More similar behavior for the rear wheel wake to the prisms layer mesh method with detailed tyres occurred.](image)

Different behavior for the total pressure around the front wheels can be seen for the refined engine bay and underbody mesh method in Figure 5.40. The two pressure wakes seen in Figure 5.25 became connected but not as wide as for the detailed tyre simulations. This behavior consisted well with the experimental measurements performed in [8]. Different behavior for the total pressure distribution could also be seen for the underbody as the finer mesh resolution induced a different flow behavior.
Figure 5.40. Total pressure distribution at a plane placed directly after the front wheel (wheel center + 320 mm) for the S60 baseline configuration for the refined engine bay and underbody mesh method. The two earlier separation wakes were connected for this mesh method which can be compared to the prisms layer mesh method where they behaved separately.

Only minor differences could be seen for the surface pressure distribution between the baseline configurations for the prisms layer mesh method and the refined engine bay and underbody mesh method. However, significant differences occurred for the rear of the car for the underbody configurations as can be seen in Figure 5.43 where the underbody configuration without front wheel deflectors was presented for the prisms layer mesh method and refined engine bay and underbody method.

Attached flow could already be seen around the left rear lamp for the V60 model meshed with the prisms layer mesh method but due to the increased mesh resolution at the engine bay and underbody higher energy was obtained at the rear wake which increased the attached flow around the rear lamps and the D-pillars. Also a lowered pressure on the rear of the tyres could be seen as a result of the prisms layers on the tyres, which resulted in increased drag force. However, a clearer separation could be seen for the lower part of the rear bumper for the refined engine bay and underbody mesh method as more distinct separation of the pressure zones could be seen. Lowered pressure could also be seen for the rear windscreens for the refined engine bay and underbody mesh method which resulted in increased drag force and increased rear lift force.
Minor difference for the surface pressure distribution could be seen for the underbody for the baseline configuration between the prisms layer mesh method and the refined engine bay and underbody mesh method. However, for the underbody configurations significant differences occurred which can be seen in Figure 5.42 where the configuration without the front undershield can be seen for the prisms layer and refined engine bay and underbody mesh methods. Surprisingly a more attached flow could be seen over the end of the lower front exterior which directed more flow onto the engine undershield which increased the pressure on the front side of it hence the drag and front lift forces. The refined engine bay and underbody mesh also generated a lowered pressure on the underbody panels and higher pressure on the front side of the rear wheels which had effects on the lift force balance of the car. More asymmetrical flow behavior for the underbody also occurred with the refined engine bay and underbody mesh method.

The different pressure distribution was believed to be generated from the increased mesh resolution for the wall bounded flow which increased the possibilities to capture it more correctly. A finer volume mesh for the underbody also seemed to generally increase the flow effects of the underbody.
Figure 5.42. Pressure distribution for the surfaces of the underbody for the prisms layer mesh method (top) and the mesh method with refined engine bay and underbody (bottom) for the V60 without front undershield. Note the general lower pressure on several underbody panels and the increased pressure on the engine undershield.

Slight better correlation to the wind tunnel measurements can be seen for the refined engine bay and underbody mesh method compared to the prisms layer mesh method for the drag force in Figure 5.43. However, the drag force correlations were still not always better than the AEDCAE02 mesh method. Interestingly significant improvements could be seen for the configuration affecting the very front of the underbody. However, these configurations caused considerably more disturbances to the flow under the car which increased the attached flow at the back of the V60 model (seen in Figure 5.41), which increased the drag greatly.
Figure 5.43. Drag force coefficient delta to the wind tunnel measurements for the three considered mesh methods and for some underbody configurations for the V60 model. Note the better correlation to the wind tunnel measurements for the refined engine bay and underbody mesh method for the configuration affecting the very front of the car.

Improved correlation for the front lift force for the refined engine bay and underbody mesh method can also be seen in Figure 5.44, for the configurations which affected the very front of the underbody. This was believed to come from the prisms layers on the tyres which improved the behavior of the wall bounded flow on the tyres. As both the removed front wheel deflector (fwDEFL) and front undershield (FUS) greatly affect the amount of flow around the front wheels the effect of the prisms layers on the tyres became more significant.

Figure 5.44. Front lift force coefficient for the three considered mesh methods for baseline and three underbody configurations for the V60 model. Note that positive value represented an under predicted value compared to the wind tunnel measurements.

Generally worsened correlation for the rear lift force was obtained with the refined engine bay and underbody mesh method compared to the prisms layer mesh method which can be seen in Figure 5.45. This was due to the lowered pressure at the rear of the car which induced higher rear lift force which decreased the correlation to the wind tunnel measurements. However, the mesh method still improved the correlation to the experimental data compared to the AEDCAE02 mesh method.
Worsened drag force trend can generally be seen in Figure 5.46 for the refined engine bay and underbody mesh method. However, for the configuration without engine undershield (EUS) similar drag force trend could be seen between the mesh methods due to the flow separated similarly to the prisms layer mesh method for the refined engine bay and underbody mesh method. The over predicted drag trend for the configurations without front wheel deflectors (fwDEFL) and front undershield (FUS) was due to the increased energy in the rear wake which caused more attached flow around the rear lamps and D-pillars which increased drag greatly.

For the front lift force trend an improved correlation to the wind tunnel trends can be seen in Figure 5.47 for the refined engine bay and underbody mesh method. This was believed to come from the improved prediction of the wall bounded flow which improved the prediction of the pressure distribution for the underbody of the car. Also in this case greatly improved correlation could be seen for the removed front undershield (FUS) configuration which shows the importance to model the flow correctly at the beginning of the underbody.
Figure 5.47. Front lift force trend for the mesh methods and underbody configurations. Note the greatly improved correlation for the refined engine bay and underbody mesh method.

Also for the rear lift force trend the correlation to the trends for the wind tunnel measurements were improved with the refined engine bay and underbody mesh method, seen in Figure 5.48. For the removed front wheel deflector (fwDEFL) configuration the trend direction was correctly predicted for the rear lift force which not had be seen with the earlier tested mesh methods. However, for the removed front undershield and engine undershield configurations improved correlation could be seen even though not correct trend direction still was fully obtained with the refined engine bay and underbody mesh method.

Figure 5.48. Rear lift force trends for the considered mesh methods and the underbody configurations affecting the front of the car. Notice the improved trend prediction for the refined engine bay and underbody mesh method compared to the other two considered mesh methods.

5.4.1 General discussion about the refined engine bay and underbody mesh method

More effective procedure for the refined engine bay and underbody mesh method could clearly be developed but due to the limited time and possibilities in Harpoon it was hard to obtain. The wanted refinements could possibly be done with help of prisms layer which would reduce the mesh size and thereby also the lead time for the method. It was also unclear if the need of refining the underbody existed for models with higher ride height as the ground effect would affect less. The effects were also
small for the baseline which indicates that the mesh refinements may not be needed for geometries with flat or close to flat floors. However, if underbody configurations are to be considered the effects could be seen to be large.

### 5.5 Effects on processing and computational cost

All the mesh methods and the detailed tyre model increased the computational cost compared to the AEDCAE02 method which can be seen in Figure 5.49 as the mesh size increased for all the considered mesh methods and the detailed tyre model.

![Mesh size comparison for considered mesh methods compared to the AEDCAE02 method for the V60 model.](image)

The prisms layer mesh method slightly increased the simulation time as the mesh size increased with almost 50 million cells for the V60 model. However, due to poorer mesh quality was obtained with the prisms layer mesh method more smoothing iterations were needed in both Harpoon and Ansys Fluent which also slightly increased the total lead time.

Only adding detailed tyres increased the mesh size with around 20 million cells which not had a significant effect on the simulation time. Some more steps were however needed in the morphing procedure which increased the CAD effort but only needed to be considered once for each model and curb height.

For the refined engine bay and underbody mesh method the total lead time was more than doubled due to the greatly increased mesh size which increased the mesh generation time and needed simulation time.
6. Conclusions

A Study of some numerical and geometrical parameters affecting the lift force of passenger cars have been made. Significant impact on the aerodynamic forces and flow field could be seen for the different mesh methods which indicated that the solutions obtained with the AEDCAE02 method not were fully mesh independent. However, improved trend correlation to wind tunnel measurements was achieved with use of different meshing techniques and methods.

Absolute correlation for the absolute values of the aerodynamic forces should not be obtained with currently used simulation method as the simulated domain and boundary conditions differed between the CFD and wind tunnel. These differences had shown significant differences for the flow field and thereby the aerodynamic forces in [1].

The increased mesh resolution for the wall bounded flow generally caused more detached flow which was favorable as the $k$-$\varepsilon$ turbulence models generally over predict attached flow.

Increased CAD work was needed in order to create the volumes so prisms layer generation would be possible on a single side of a surface in Harpoon.

Improved behavior of the boundary layer could clearly be seen with the implemented prisms layers which corresponded better to theoretical behavior of boundary layers. It also had an effect on the flow separation which was more distinct with use of prisms layers and thereby decreased the overall predictions of the attached flow. This also led to improved skin friction behavior which may be more important to consider for future car models as the pressure drag force tends to be decreased with the development.

Improved correlation for the absolute value of the rear lift force and the trends for the lift forces could be obtained with the prisms layer mesh method while the drag force correlation generally was worsened for both car models.

Detailed tyres caused similar behavior for the flow around the wheels as for the prisms layer mesh method. Slight improved behavior to experimental measurements in study [8] was seen for the detailed tyre but could be seen to be captured with use of less computational costly methods as prisms layers on tyres resulted in similar behavior. The detailed tyres did however cause more disturbed flow for the underbody which affected the rear wake of the cars. Especially, the more sensitive wake of the S60 model was affected and resulted in over attached flow around the rear lamps.

Large effects of the detailed tyres where expected for the drag force but only minor differences compared to the slick tyres could be seen for the drag force in the results. However, the effect on the lift force was significant which also corresponded well to earlier study [8] for the same car models. Also the importance of including the detailed tyre when considering rim configurations could be seen to be important in this study. However, for configuration simulations not considering different rim designs the need for including the tyre grooves seemed unnecessary as the general shape of the tyre has a larger impact on the flow field hence the aerodynamic forces as also seen in [7] and [8].
The effects of the refinements for the engine bay and underbody were significant and improved the trend correlation to the wind tunnel measurements greatly. Only slight differences for the flow field at the underbody occurred between the AEDCAE02 mesh method and the refined engine bay and underbody mesh method for the baseline configuration. However, for the underbody configurations large differences for the flow field and thereby the pressure field occurred which showed the importance of refining the mesh for the engine bay and underbody. Most importantly, improved lift force trend correlation to the wind tunnel measurements could be seen for the refined engine bay and underbody mesh method. This was deemed more important than the absolute values as the flow for the wind tunnel have shown to not be the same as in the CFD domains [1].

The general conclusion for the study was that improved correlation for the aerodynamic forces and especially the lift force and distribution could be improved for steady-state simulations by use of an updated mesh strategy.
7. **Future work**

An improved numerical method is still needed in order to improve the correlation between CFD simulations and wind tunnel measurements. The mesh has been seen to have great importance and can increase the correlation.

Increased number of cells covering the wall bounded flow should be considered as it may improve the correlation but also make the flow behavior more consistent between the models and geometrical configurations. With consistent flow behavior the boundary conditions and turbulence model may be trimmed in to match the behavior of the wind tunnel measurements.

To increase the number of cells inside the boundary layer less focus may be needed on the $y^+$ and thereby use of the Enhanced wall treatment should be considered to minimize the need of obtaining specific $y^+$ values. Variable first layer thickness depending on the flow field and geometry would also be needed to be considered in order to meet these requirements better.

Due to Harpoon limits the number of prisms layer possible to achieve a refined surface mesh would be needed in order to increase the mesh resolution for the boundary layer. Use of other mesh generator which would make it possible to achieve more prisms layer should thereby be considered if increased mesh resolution for the boundary layer is wanted for a smaller computational cost.

The refined engine bay and underbody method showed great potentials for improvement of the correlation between CFD simulations and wind tunnel measurements. However, the computational time increased but can possibly be reduced by use of prisms layer at the underbody and wheels.

Improved understanding of the flow behavior inside the wind tunnel is needed as significant differences have been seen in [1] which worsens correlation to the CFD simulations and may cause miss leading correlation behavior.

The need of transient and more advanced turbulence models which not models the whole turbulence flow field should be considered. This would decrease the turbulence modelling effect on the results as the more resolved structures would increase the physical behavior of the flow.

Most importantly better understanding between the differences of the CFD simulation and wind tunnel measurements can be obtained if flow visualization data exist for the considered models. This would help to see where the significant difference occurs and where major improvements may be needed in the CFD procedure.
References


Appendix A

A.1 Geometry problems
Due to the complexity of the CAD models problems occurred during meshing which resulted in mesh leaks into regions where no flow was expected or wanted. This especially occurred in the beginning of this thesis where holes in the CAD model caused Harpoon to build the mesh under the A-pillars, seen in Figure A.1. During the solving in Ansys Fluent high velocities then occurred in these region causing divergence of the solution.

![Figure A.1. Mesh leakage in under the a-pillars which caused non-physical high velocity flows in these regions.](image)

The CAD geometry of the cooling package also caused problems in Ansys Fluent as a few poor quality cells were created in Harpoon. This happened due to surfaces were penetrating the cooling package in the CAD geometry and as Harpoon then snapped the volume mesh to the wrong nodes this resulted in poor quality cells. This sometimes led to poor convergence and divergence.

A.2 Additional figures for the configurations
Here can the configurations affecting the rear exterior of the V60 model be seen in Figure A.2 and A.3.

![Figure A.2. Large triangle placed between the rear spoiler and rear lights. Called LTR in configuration Table (2.2.2) and Results (Chapter 5). The small triangle had the same shape but was around half the size of the LTR.](image)
A.3 Harpoon version 5.4beta17

During this thesis a new version of Harpoon was released which made it possible to build prisms layers on one side of surfaces. The side was the prisms layers were to be built was the set by setting the general normal for the surface/PID. This corresponded to the direction of which most area could be seen for the surface but could only be set in the coordinate system directions. So for example the rear bumper of the car was set in the positive x-direction.

However, in this version problems with the prisms layers occurred where the surface mesh size changes size. As also the prisms layers are affected by this in Harpoon quality problems occurred in these regions which can be seen in Figure A.4. The problems consisted of poor cell quality and of fewer layers.

This resulted in poorer resolution of the boundary layer which caused a different behavior of the boundary layer. This different behavior of the boundary layer caused over attached flow around the rear lamps of the baseline V60 configuration which not occurred for the older version of Harpoon and was deemed to be unphysical.
A.4 Additional simulations

Additional investigations were done during this thesis and are presented in this section.

Additional smoothing iterations were added in Harpoon and Ansys Fluent in order to improve the mesh quality but generally resulted in longer mesh generation and computational time. The total aerodynamic forces were not significant affected by the increased number of smoothing iterations as only marginally better mesh resolution occurred with many (around 10 in Harpoon) more smoothing iterations.

Enhanced wall treatment was tested on the same mesh as the baseline V60 configuration mesh with the prisms layer mesh method. The differences for the aerodynamic force coefficients are presented in Table A.1.

Table A.1. Differences between Standard wall function and Enhanced wall treatment for the same prisms layer mesh for the V60 baseline configuration.

<table>
<thead>
<tr>
<th>$\Delta C_D$</th>
<th>$\Delta C_{lf}$</th>
<th>$\Delta C_{lr}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>-0.001</td>
<td>-0.003</td>
<td>-0.009</td>
</tr>
</tbody>
</table>

This resulted in slightly worse correlation for the absolute values for the $C_D$ and $C_{lf}$ while improved correlation occurred for $C_{lr}$. More separated flow could also be seen when the total pressure equal to zero was investigated.

No significant differences could be seen when prisms layer were added on the muffler for the S60 baseline configuration and was therefore not more investigated.

Curvature correction was tested in order to see if the over attached flow around the rear lamps of the S60 model could result in more reasonable flow around the lamps. No significant differences occurred and the flow still stayed attached over the rear lamps of the S60 model.

In Figure 5.4 can the separations occurring on the side of the car be seen to be slight limited by the end of the refinement box. To see if the refinement box at the rear of the car not was wide enough which could cause the flow to be directed by the mesh and thereby affect the result. One simulations was therefore run with an increased width of the rear refinements box but no significant difference could be seen.

A test with increased length of the rear refinement box was also done to see if it influenced the results. No differences could be seen for the flow field or the aerodynamics forces and therefore can the used refinements boxes used in this thesis still be used.

As the wind tunnel measurements was performed in 140 km/h while the CFD simulations was performed in 100 km/h a noticeable difference between the Reynolds numbers occurred. To see if that could be one of the reasons why to the differences between CFD and wind tunnel measurements it was investigated. Same mesh as for the 100 km/h simulations were kept which resulted in a slightly poorer $y^+$ distribution but still was much better than a case without prisms layers. The cooling pack fans speed were assumed increase linearly with the free stream velocity to consider the increased engine speed/power needed. No significant differences for the flow field and the aerodynamic forces could be seen for the increased velocity and was therefore not more investigated.
A.5  Surface thickness creation procedure

Original model

- Volumes only need to be created for the front exterior and the rear bumper of the car.
- Different PID dividing is needed.

Front PID dividing

Show only the surfaces where you want to have prisms layers and change the PID to something like this configuration.

The PID naming will later be changed. The used here is just for clarification.

Front scaling

- Scale the front.
- If base level 40 mm is going to be used use a scale factor of 0.992 to provide enough thickness of the volume.
- Use the front wheel center as scaling point and set y = 0.
- Use auto-offset for PID in Transformation options.

Different scale factors may be needed for different models! Use the Measure tool to ensure that you have a thickness of the volume larger than the surface mesh size!
Connect the surfaces

Surfaces so they do not have the same name as the outer surfaces. E.g. wall-exterior-interior-front. Change the name of the interior.

Connect the outer and inner surfaces edges with FACES—NEW—Coons.
Do this for all the surrounding edges so a volume is created.

Penetration

When the front volume is create a number of parts penetrates the interior side of the volume. The surfaces who penetrates needs to be removed as they can cause prisms layer collapses. The intersection tool works well for it. The hood insulation can be moved some mm in the negative z-direction so not all the surfaces needs to be removed.

Cleaned

No penetrating surfaces to ensure no prisms layer collapse.
Rear volume

The same thing needs to be done for the rear bumper. A good thing is to set an own PID for the lower part of the rear bumper as only that parts needs to be generated into a volume. Sometimes it can be easier to scale parts of the rear bumper and then sew everything together to a volume. It can also be good to cut away parts which not is needed, e.g. the tow system.

A higher scaling factor can also be used here. For the XC60 a scaling factor of 0.995 worked well. The interior parts where then also moved into a acceptable position was found. Instead of using the rear wheel center as scaling point the origin was used as it generates a better scaled copy around the exhaust covers.

Harpoon

Do not forget to command Harpoon to find the created volumes and build prisms layers on the correct surfaces.

Add a couple of more smoothing iterations can be a clever idea to reduce the number of highly skewed cells.
A.6 Updated tyre morphing procedure

Revised Procedure for Morphing Tyres

Information

- This tyre morphing procedure is based on the work provided by Peter Milinaric and Teddy Hobeka.
- ANSA version 14.1.2 was used for creation of this procedure.
- Continental Sport Contact 3 was used for this procedure (CSC3_X82127_STEP2.ansa).

Creating the tyre model

- The tyre is often achieved from the manufacturing in this condition where the outer surfaces represent the manufacturing tool for the tyre.
Creating the tyre model

- Remove all the unneeded surfaces so only the outer surfaces exist.

Creating the tyre model

- The model may need to be cleaned and fixed in order to work properly.

Creating the tyre model

- It is now time to merge the tyre thread with the CSC3_tyre_slick.ansa in order to achieve the tyre side wall in the correct position.
Creating the tyre model

- After merging it should look like this.
- As can be seen the slick tyre patch sticks through the grooves.
- Removed the slick patch.
- Use the TOPO command in ANSA to connect the grooves to the remaining side wall.

![Tyre Model](image1)

Creating the tyre model

- Now it is time to mesh the tyre. Go to PERIMETERS-SPACING-AUTO-STL and set Chordal Derivation = 0.1 and Max Length = 1. This fineness is only needed if the grooves has a plateau otherwise a slight coarser mesh size can be used.
- Mesh with MESH GEN-STL-VISIBLE. This should generate a mesh around 3.5 million cells.
- This is needed to capture all the geometrical features properly.
- Ensure that your mesh do not have any leakages.

![Mesh Example](image2)

Creating the tyre model

- Now should the tyre model be finished.
- It is now time to move the tyre to the correct position and morph it to the wanted shape.
- Before moving it ensure that you know the correct rolling direction!
Moving the tyre

- It can be a good idea to change the name of the PID’s before moving it to correct position as the side walls PID’s get the same name.
- Set different PID for the grooves which will make it possible to morph them later.
- It is however no need for changing PID for the whole grooves. It is just enough to do it around the expected contact patch.
- Just change the bottom of the grooves PID.

Moving the tyre

- Generally the bottom of the grooves should just have another PID compared to the rest of the tyre.
- Special PID’ing can be needed for some grooves.
- For example in the with this kind of groove it is a good idea to PID it in this way as problems may otherwise occur when morphing.
- Different PID for the grooves plateau is only needed at the contact patch!

Moving the tyre

- It is now time to move the tyre into correct position.
- Insert the morphing boxes by merging Tyres_ready_to_morph.ansa.
- This part present the correct position of the tyres and the current tyre needs to be moved to their positions.
Moving the tyre

- Select Transform-Copy and select all the entities for the tyre (do not forget to turn on curves!).
- Select the Transform menu and select three points on the tyre and the tyre you want to move it to.
- Do it for both tyres.
- Control the tyres rolling direction!

Fitting the Morphing Boxes

- Change all the PID's so it is either front or rear.
- Remove the old slick tyres (included in Tyres_ready_to_morph.ansa) and import (merge) Morphin_boxes_tyre_with_pattern.ansa.
- Sometimes the morphing boxes needs to be moved to better fit the positioned tyre.
- Often ANSA 14.1.2 crashes when doing this in windows 7 and therefore it can be better do to this and the box fitting in LINUX by use of Thinlinc.

Fitting the Morphing Boxes

- Fit the morphing boxes to the tyre rim contact and tyre widest point by use of BOX MORPHING - FIT tool.
- According to Hobelka it is better to fit the boxes to the curves but the morph box to fit to the tyre rim contact can be fitted to the tyre contact edge.
- It is better to fit half the tyre side separately.
- Due to this it can sometimes be needed create (use Project) hot points at the top of the curves.
Fitting the Morphing Boxes

- In the pictures below Ellipses 1, 2, and 3 have been numbered and selected in a red box as to avoid confusion.
- Ellipse 1 is an ellipse below ground level. The control points that form this ellipse will be used when flattening out the part of the tyre that extrudes below ground level for a better visualization of ground contact patch. This will be the last step of the morphing procedure.
- Ellipse 2 is at ground level. When the tyre is to be translated vertically to match the measured contact patch, the whole morphing box should be translated in a way that ground is at the level of Ellipse 2.
- Ellipse 3 is the ellipse at the level where the grooves should stop. Rain grooves on a tyre stay open when the tyre contacts the ground as their main job is to evacuate water. We will use this third ellipse to move the rain grooves to that level. The location in $z$ of this ellipse can be approximated as follows: if the total depth of the grooves is 8mm, then assuming a 1mm compression of the tyre pattern then ellipse 3 is to be located 7 mm above ground level.

Tyre Morphing

- Translate the three Ellipses vertically in $z$ (by use of BOX MORPHING - MOVE) so that Ellipse 3 becomes ground level. As shown in the pictures below:
Tyre Morphing

- Show all PIDs and then load all morphing boxes for the tyre by using BOXES – LOAD - VISIBLE and then select all and press the middle button.

Tyre Morphing

- Open CONTROLS - PARAMS. Ensure that Morphing is active!
- Select parameter and then Morph and then specify values according to the table.
- Correct order is important for the parameters.
- The lift parameter is not needed if the tyre are correctly positioned before morphing!

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>radiusss@_1</td>
<td>3</td>
</tr>
<tr>
<td>Lift_1</td>
<td>3</td>
</tr>
<tr>
<td>axialcontract @_1</td>
<td>3</td>
</tr>
<tr>
<td>bulge@_1</td>
<td>3.5</td>
</tr>
</tbody>
</table>

Tyre Morphing

- After this is done all Ellipses need to be transferred back with use of BOX MORPHING - MOVE - Translate to their previous position with Ellipse 2 being at ground.
- Morphing should be inactive!
Tyre Morphing

- Only show the groove PID (not the plateau).
- Check that the distance between Ellipse 2 and 3 so it correspond to the wanted groove depth.

Tyre Morphing

- Move Ellipse 2 (ground) so it goes below the groove PID as seen in figure below.

Tyre Morphing

- Load morphing boxes by use of BOXES – LOAD - VISIBLE and then select all.
- Active Morphing.
- Then use BOX MORPHING - MOVE - Translate and move Ellipse 2 to Ellipse 3.
Tyre Morphing

- Move back Ellipse 2 and then only show the groove plateaus.
- Now load the morphing boxes again and move Ellipse 2 up to Ellipse 3 and subtract the plateau depth to capture the plateaus geometry.

Tyre Morphing

- Deactivate morphing.
- Now move Ellipse 2 back down to ground level and show all tyre PiDs and load the morphing boxes.

Tyre Morphing

- Activate morphing.
- Now finally move Ellipse 3 up to the ground level.
Tyre Morphing

- You should now have a morphed detailed tyre with correct contact patch.
A.7 Harpoon script for prisms layer generation

import tgrid ../HARPOON/surf.msh
**VERSION v5.2a**
**PREFERENCES USED**
**Max Skew 0.999500**
**Target Skew 0.980000**
**Max Face Warpage 40.000000**
**noreset**
**intersect**
**Separation Angle 40.0**
**Setting No. of Cells Between walls to 3**
**Setting BDF Exports to Short Format**
**Setting BDF Pyramid Treatment to use degenerate PENTA elements**
**Setting Max No. Separate Volumes to 100**
**Setting No. Cells for Auto Volume Delete to 5**
**Setting Part Description to use STL name**
**Setting Fluent Thin Wall Treatment to Single Sided**
baselev 40.000000
farfield global
farfield xmin -14232
farfield ymin -4750
farfield zmin 173
farfield xmax 35768
farfield ymax 4750
farfield zmax 10173
wlevel xmax -3
wlevel xmin -3
wlevel ymax -3
wlevel ymin -3
wlevel zmax -3
wlevel zmin -3

**************************
************
** REFINEMENT BOXES **
**************************
** Surrounding box **
refine
0 0
-159 -1540 173
10034 1540 2061
** Farfield box **
refine
0 1
4482 -1110 173
7718 1110 1822
** Underbody box **
refine
0 2
618 -1110 173
5601 1110 867
** Nearfield box **
refine
0 2
4482 -1110 173
5601 1110 1822
** Mirror box **
refine
0 2
2193 -1177 960
4482 1177 1360
** COND and RAD box **
refine 2 5
1024 345 530
1024 -345 530
1100 -345 530
1100 345 530
1024 345 885
1024 -345 885
1100 -345 885
1100 345 885
** CAC box **
refine 2 5
950 345 392
950 -340 392
1100 -340 392
1100 345 392
950 345 545
950 -340 545
1100 -340 545
1100 345 545
**********************************************
** MESH METHODS**
**********************************************
type hex
expand slow
mesh both
remove volume -3
** SINGLE LEVEL**
level 1
gminlev 1
gmaxlev 5
plevel * -6 6 6 0
plevel * -5 5 5 0
plevel * -4 4 4 0
**********************************************
** SURFACE MESH EXPANSION **
**********************************************
pexp wall-exterior-front-a-pillar-* 4
pexp wall-exterior-front-mirror-5 8
pexp wall-exterior-body-4 15
pexp wall-exterior-front-4 15
pexp wall-exterior-front-radius-6 3
pexp wall-exterior-front-wheelhouse-radius-6 3
pexp wall-exterior-rear-4 15
pexp wall-exterior-rear-6 3
pexp wall-exterior-rear-radius-6 3
pexp *suspension* 15
pexp *underbody* 15
pexp wall-wheel* 8
**********************************************
** PRISMS GENERATION **
**********************************************
** First layer thickness | # of layers | 0 | Growth rate | 0 **
** Test showed that it was more robust to keep # of layers equal over the whole car **
layer wall-exterior-body-4 1.0 6 0 1.2 0
layer wall-exterior-front-4 1.0 6 0 1.2 0
layer wall-exterior-front-5 1.0 6 0 1.2 0
layer wall-exterior-front-6 1.0 6 0 1.2 0
layer wall-exterior-front-lamp-5 1.0 6 0 1.2 0
layer wall-exterior-front-mirror-5 0.7 6 0 1.2 0
layer wall-exterior-front-rails-5 1.0 6 0 1.2 0
layer wall-exterior-front-sharkfin-5 1.0 6 0 1.2 0
layer wall-exterior-rear-4 1.0 6 0 1.2 0
layer wall-exterior-rear-6 0.7 6 0 1.2 0
layer wall-exterior-rear-radius-6 0.7 6 0 1.2 0
layer wall-exterior-rear-wheelhouse-radius-6 1.0 6 0 1.2 0
layer wall-exterior-front-wheelhouse-radius-6 1.0 6 0 1.2 0
layer wall-exterior-front-mouldings-6 1.0 6 0 1.2 0
layer wall-exterior-front-a-pillar-5 0.7 6 0 1.2 0
layer wall-exterior-rear-spoiler-5 1.0 6 0 1.2 0

**************************************************************
**FIND VOLUMES**
**************************************************************

vfind start
baffle-underbody*
wall-engine-bay-*
wall-coolpack-fan-*
wall-coolpack-shroud-*
wall-powertrain-*
wall-suspension-*
wall-underbody-*
wall-exterior-front-*
wall-exterior-front-lamp-5
wall-interior-front-*
wall-exterior-front-radius-6
wall-coolpack-cond-tank-6
wall-exterior-rear-6
wall-underbody-rear-upper-structure-4
vfind end

**************************************************************
**SORT OUT VOLUMES TO KEEP**
**************************************************************

**MRF WHEEL FRONT LS**
vnamekeep begin fluid-wheel-front-ls-mrf
fan-wheel-front-ls-mrf-5
wall-wheel-front-ls-rim-stationary-5
vnamekeep end

**MRF WHEEL REAR LS**
vnamekeep begin fluid-wheel-rear-ls-mrf
fan-wheel-rear-ls-mrf-5
wall-wheel-rear-ls-rim-stationary-5
vnamekeep end

**MRF WHEEL FRONT RS**
vnamekeep begin fluid-wheel-front-rs-mrf
fan-wheel-front-rs-mrf-5
wall-wheel-front-rs-rim-stationary-5
vnamekeep end

**MRF WHEEL REAR RS**
vnamekeep begin fluid-wheel-rear-rs-mrf
fan-wheel-rear-rs-mrf-5
wall-wheel-rear-rs-rim-stationary-5
vnamekeep end

** CAC **
vnamekeep begin fluid-cac
fan-coolpack-cac-in-6
fan-coolpack-cac-out-6
wall-coolpack-cac-6
vnamekeep end
** COND **

  vnamekeep begin fluid-cond
  fan-coolpack-cond-in-6
  fan-coolpack-cond-out-6
  wall-coolpack-cond-6
  vnamekeep end

** RAD **

  vnamekeep begin fluid-rad
  fan-coolpack-rad-in-6
  fan-coolpack-rad-out-6
  wall-coolpack-rad-6
  vnamekeep end

** FAN LARGE**

  vnamekeep begin fluid-fan-large-mrf
  fan-coolpack-fan-large-in-6
  fan-coolpack-fan-large-out-6
  wall-coolpack-fan-large-blade-6
  wall-coolpack-fan-large-shroud-stationary-6
  vnamekeep end

** FAN SMALL**

  vnamekeep begin fluid-fan-small-mrf
  fan-coolpack-fan-small-in-6
  fan-coolpack-fan-small-out-6
  wall-coolpack-fan-small-blade-6
  wall-coolpack-fan-small-shroud-stationary-6
  vnamekeep end

***********************************************************************
**SET BC ON FAN SURFACES (RADIATOR)**
***********************************************************************

setbc fan-* radiator

***********************************************************************
**SMOOTH**
***********************************************************************

smooth 4 0.98
smooth 4 all
smooth 4 0.98

***********************************************************************
**EXPORT TO FLUENT**
***********************************************************************

vischeck
export fluent vol ./HARPOON/harpoon_volmesh.msh
**save harpoon ./HARPOON/harp_vol
A.8 Removal of negative volumes in Ansys Fluent
Here is the script used in Ansys Fluent to remove the negative volumes. It was most efficient used directly after the constant velocity initialization (/solve/initialize/initialize-flow) and before the FMG initialization (/solve/initialize/fmg-initialization/ yes).

```
adapt/mark-inout-iso-range
yes
cell-volume
-1
0
mesh/modify-zones/sep-cell-zone-mark
volume_1
0
yes
mesh/modify-zones/delete-cell-zone
volume_1:*()

:: MESH QUALITY CONTROL
/mesh/size-info
/mesh/quality
/mesh/smooth-mesh
"quality based"
8
0.005

/mesh/quality
```
A.9 PID set up for the refined engine bay and underbody method

baffle-underbody-fwd-4
baffle-underbody-tank-panel-ls-5
baffle-underbody-tank-panel-rs-5
baffle-underbody-ubp-ls-5
baffle-underbody-ubp-rs-5
fan-coolpack-cac-in-6
fan-coolpack-cac-out-6
fan-coolpack-cond-in-6
fan-coolpack-cond-out-6
fan-coolpack-fan-large-in-6
fan-coolpack-fan-large-out-6
fan-coolpack-fan-small-in-6
fan-coolpack-fan-small-out-6
fan-coolpack-rad-in-6
fan-coolpack-rad-out-6
fan-exterior-front-grille-intake-5
fan-exterior-front-spoiler-intake-5
fan-underbody-engine-undershield-hole-5
fan-wheel-front-ls-mrf-5
fan-wheel-front-rs-mrf-5
fan-wheel-rear-ls-mrf-5
fan-wheel-rear-rs-mrf-5
wall-coolpack-4
wall-coolpack-5
wall-coolpack-cac-6
wall-coolpack-cac-support-6
wall-coolpack-cond-6
wall-coolpack-cond-tank-6
wall-coolpack-fan-flaps-6
wall-coolpack-fan-large-blade-6
wall-coolpack-fan-large-omega-6
wall-coolpack-fan-large-shroud-stationary-6
wall-coolpack-fan-small-blade-6
wall-coolpack-fan-small-omega-6
wall-coolpack-fan-small-shroud-stationary-6
wall-coolpack-rad-6
wall-coolpack-rad-support-6
wall-coolpack-shroud-5
wall-engine-bay-air-distribution-4
wall-engine-bay-air-guides-4
wall-engine-bay-air-guides-4
wall-engine-bay-battery-4
wall-engine-bay-brake-system-4
wall-engine-bay-bumper-hood-systems-4
wall-engine-bay-catalyzer-5
wall-engine-bay-cleaning-4
wall-engine-bay-climate-4
wall-engine-bay-electrical-architecture-4
wall-engine-bay-electronic-4
wall-engine-bay-electronics-4
wall-engine-bay-engine-4
wall-engine-bay-engine-mounts-4
wall-engine-bay-functional-black-trim-4
wall-engine-bay-hood-structure-4
wall-engine-bay-plenum-cover-4
wall-engine-bay-steering-4
wall-engine-bay-structure-4
wall-engine-bay-transmission-4
wall-engine-bay-wires-4
wall-exterior-body-4
wall-exterior-cleaning-4
wall-exterior-front-4
wall-exterior-front-5
wall-exterior-front-6
wall-exterior-front-a-pillar-5