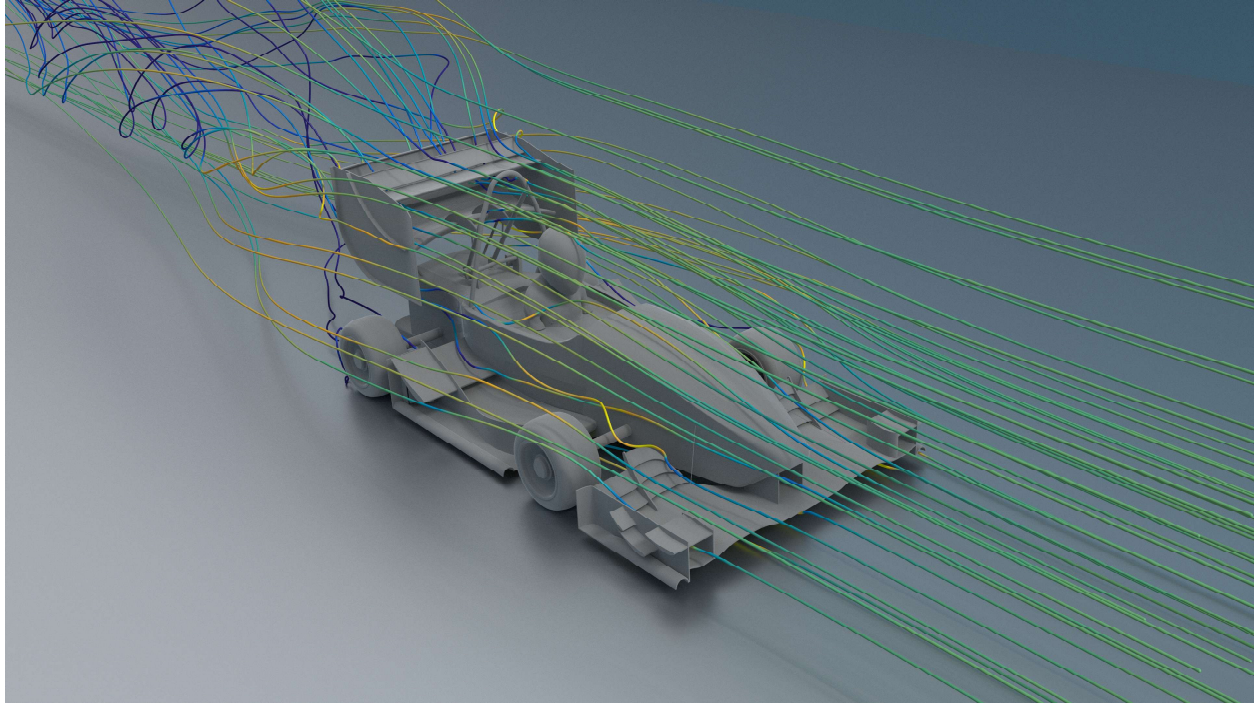




CHALMERS
UNIVERSITY OF TECHNOLOGY



Construction and Evaluation of a Scale Model of The Chalmers Formula Student Car

Degree project report in Mechanics and Maritime Sciences

Blend Ahmed Omar
Darin Dara Hassan
David Stjernström Eriksson
Harun Trnjanin

MECHANICS AND MARITIME SCIENCES

CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden 2026
www.chalmers.se

DEGREE PROJECT REPORT 2026

Construction and Evaluation of a Scale Model of The Chalmers Formula Student Car

Blend Ahmed Omar
Darin Dara Hassan
David Stjernström Eriksson
Harun Trnjanin



Mechanics and Maritime Sciences
CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden 2026

Construction and Evaluation of a Scale Model of The Chalmers Formula Student Car

Blend Ahmed Omar

Darin Dara Hassan

David Stjernström Eriksson

Harun Trnjanin

©

Blend Ahmed Omar

Darin Dara Hassan

David Stjernström Eriksson

Harun Trnjanin , 2026.

Supervisor: Vincent Fröberg, Chalmers Formula Student

Supervisor: Panos Papakostas, Chalmers Formula Student

Supervisor: Alexey Vdovin, Department of Mechanics and Maritime Sciences

Examiner: Simone Sebben, Department of Mechanics and Maritime Sciences

Degree project report 2026

Department of Mechanics and Maritime Sciences

Chalmers University of Technology

SE-412 96 Gothenburg

Sweden

Telephone +46 31 772 1000

Cover: Wind visualization constructed in STAR-CCM+ showing a simulation of the airflow around the simplified car.

Typeset in L^AT_EX

Gothenburg, Sweden 2026

Construction and Evaluation of a Scale Model of The Chalmers Formula Student Car

Blend Ahmed Omar

Darin Dara Hassan

Harun Trnjanin

David Stjernström Eriksson

Department of Mechanics and Maritime Sciences

Chalmers University of Technology

Abstract

In Formula Student competitions, small improvements in aerodynamic performance can have a decisive impact on overall vehicle competitiveness. The aim of this project was to 3D-print a modular scale model of the Chalmers Formula Student car, enabling aerodynamic evaluation of a few design configurations in both computational and experimental environments.

The original vehicle geometry was simplified in CAD to simplify manufacturing and ensure feasibility of scale-model testing. Non-critical features were removed while preserving the main aerodynamic surfaces responsible for downforce and drag generation. The resulting modular model allowed systematic variation of key aerodynamic components across different configurations.

Computational fluid dynamics (CFD) simulations were conducted using STAR-CCM+, where aerodynamic forces and properties were evaluated for each configuration. The same configurations were tested experimentally in the Chalmers wind tunnel to enable comparison and validation of the numerical results.

The results showed consistent discrepancies between CFD and wind tunnel measurements, with CFD generally predicting higher values of both drag and downforce. Despite these differences in absolute values, the relative trends between configurations were largely consistent, indicating that CFD could still be used effectively for comparative design studies.

The observed discrepancies are primarily attributed to differences in setup between the numerical and experimental environments, including boundary condition simplifications, surface quality of the printed model, and experimental mounting constraints. Overall, the study demonstrates that although CFD requires careful calibration against experimental data, it remains a valuable tool in the aerodynamic development of a Formula Student car. Furthermore, the results highlight that small variations in surface quality and manufacturing tolerances can have a significant influence on the resulting aerodynamic performance, underscoring the sensitivity of scale-model testing.

Keywords: Formula Student, Aerodynamics, Scale Model, Wind Tunnel Testing, CFD, CAD, 3D-printing, Modular Design, Drag Coefficient, Lift Coefficient.

Acknowledgements

We would like to thank Simone Sebben and Alexey Vdovin for all the help and guidance we have received throughout the project.

Our thanks also go to Panos Papakostas and Vincent Fröberg, for being available to help in every way they could, whether it was running simulations in CFD or answering our general questions about the project.

We would also like to thank both CASE lab and eXPerimentverkstaden at Chalmers for providing the 3D-printers used in the thesis, it could not have been done without them.

Blend Ahmed Omar, Darin Dara Hassan, Harun Trnjanin, David Stjernström Eriksson
Gothenburg, May 2026

List of Acronyms

Below is the list of acronyms that have been used throughout this thesis listed in alphabetical order:

CAD	Computer Aided Design
CASE	Chalmers Autonomous Systems and Electronics
CFD	Computational Fluid Dynamics
CFS	Chalmers Formula Student
FDM	Fused Deposition Modeling
FS	Formula Student
FUSE	FUSE Lab
FVM	Finite Volume Method
NX	Siemens NX
PETG	Polyethylene Terephthalate Glycol
PLA	Polylactic Acid
RANS	Reynolds Averaged Navier-Stokes
SST	Shear Stress Transport
STL	Stereolithography
WT	Wind Tunnel

Nomenclature

Below is the nomenclature used in throughout this thesis:

A	Area
C_D	Drag coefficient
C_L	Lift coefficient
C_p	Pressure coefficient
F_D	Drag force
F_g	Gravitational force
F_L	Lift force
k	Turbulent kinetic energy
L_c	Characteristic length (wheelbase)
m	Mass
$\dot{m}$	Mass flow rate
p	Pressure
Re	Reynolds number
S	Cross-sectional area of the wind tunnel test section
u, v, w	Components of velocity vector
V	Volume
δ	Boundary layer thickness
λ	Scale factor
Λ	Blockage ratio
μ	Dynamic viscosity
ν	Kinematic viscosity
ρ	Density

Contents

List of Acronyms	ix
Nomenclature	xi
List of Figures	xvii
List of Tables	xix
1 Introduction	1
1.1 Background	1
1.1.1 Formula Student	1
1.1.2 Chalmers Formula Student	2
1.1.3 Integration of CFD and Wind Tunnel Testing	2
1.2 Purpose	3
1.3 Goals	3
1.4 Limitations	3
2 Theory	5
2.1 Fluid Dynamics Theory	5
2.1.1 Governing Laws and Equations	5
2.1.1.1 Continuity Equation	5
2.1.1.2 Conservation of Momentum and Navier Stokes Equations	5
2.1.2 Dimensionless Numbers	6
2.1.2.1 Reynolds Number	6
2.1.2.2 Lift Coefficient	6
2.1.2.3 Drag Coefficient	6
2.1.2.4 Pressure Coefficient	7
2.1.3 Boundary Layers	7
2.1.3.1 No Slip Condition	7
2.1.3.2 Boundary Layers	7
2.2 Wind Tunnel Testing	8
2.2.1 Similarity Principles of Scale Testing	8
2.2.1.1 Geometric Similarity	8
2.2.1.2 Kinematic Similarity	8
2.2.1.3 Dynamic Similarity	8
2.2.2 Wall Interference	9

2.2.2.1	Blockage	9
2.2.3	Measurement Techniques	10
2.2.3.1	Balance Measurements	10
2.3	CFD-Theory	10
2.3.1	Finite Volume Method (FVM)	10
2.3.2	Reynolds Averaged Navier-Stokes (RANS)	11
2.3.3	Turbulence Models	11
2.3.3.1	The $k-\epsilon$ Model	11
2.3.3.2	The $k-\omega$ Model	12
2.3.3.3	The SST $k-\omega$ Model	12
2.3.4	Discretization Error and Convergence	12
2.3.4.1	Mesh Independence	13
2.3.5	Boundary Conditions	14
2.3.5.1	Velocity Inlet	14
2.3.5.2	Pressure Outlet	14
2.3.5.3	No Slip Wall	14
2.3.5.4	Slip Wall	14
2.3.5.5	Symmetry Plane	14
2.4	Additive Manufacturing	14
2.4.1	Basic Principle of 3D-printing	15
2.4.2	3D-printer Settings	15
2.4.2.1	Layer Height	15
2.4.2.2	Nozzle Diameter	15
2.4.2.3	Infill	15
2.4.2.4	Supports	16
2.4.3	Print Orientation	16
3	Methods	17
3.1	CAD	17
3.1.1	Simplifications	17
3.1.2	Modular Components	19
3.1.2.1	Front Wing	19
3.1.2.2	Rear Wing	21
3.1.2.3	The Driver	23
3.1.3	Preparations for 3D-printing	23
3.1.3.1	Scale	24
3.1.3.2	Components Below The Rear Wing	25
3.1.3.3	Sidepods	26
3.1.3.4	Rear Wing Assembly	26
3.1.3.5	Front Wing	27
3.1.3.6	Wheels	27
3.1.3.7	The Monocoque	28
3.2	CFD	29
3.2.1	The File Used in The CFD Simulations	30
3.2.1.1	Mesh Convergence	33
3.2.1.2	Mesh	33

3.2.2	Simulations	34
3.3	3D-Printing and Assembly of The Model	35
3.3.1	Workflow	35
3.3.2	3D-Printing Setup	36
3.3.3	Assembly	36
3.4	Wind tunnel	40
3.4.1	Pre-Testing Setup	40
3.4.2	Wind Tunnel Testing and Measurements	41
4	Results	43
4.1	CAD	43
4.1.1	The Complete Simplified Car	43
4.2	CFD	43
4.2.1	Mesh Study	44
4.2.2	Forces and Coefficients From CFD	45
4.2.3	CFD-Scenes	46
4.2.4	Accumulated C_d and C_l	50
4.3	Wind Tunnel Test	50
4.4	Comparison Between Wind Tunnel and CFD	51
5	Discussion	53
5.1	A Comparison of the Simplified Car and the Reference Car in CFD	53
5.2	Comparison between Wind Tunnel and CFD	54
5.2.1	Deformations of The Model	54
5.2.2	Surfaces	55
5.2.3	Differences in Setup	55
5.2.4	Inherent Errors in CFD	57
5.3	The Effect of a Driver	57
5.3.1	The Driver in CFD	58
5.3.2	The Driver in The Wind Tunnel	58
5.4	The Effect of a Tilted Front Wing	59
5.4.1	The Tilted Front Wing in CFD	59
5.4.2	The Tilted Front Wing in the Wind Tunnel	60
6	Conclusion	61
6.1	The Model	61
6.2	Suggestions of Improvement	61
6.2.1	Improvements to the Sidepods	62
6.2.2	Tilted Front Wing	62
	Bibliography	63
A	Appendix 1	I
A.1	Python Code Used to Process The Wind Tunnel Data	I
A.2	Wind Tunnel Balance Output	II

List of Figures

1.1	The 2025 CFS car [3].	2
2.1	Boundary layer development over a flat plate [17] (CC0 1.0).	7
2.2	Illustration of geometric similarity [18] (CC BY-NC-ND 4.0).	8
2.3	Visualization of blockage effects in wind tunnels [18] (CC BY-NC-ND 4.0).	9
2.4	Visualization of a control volume and its cells.	11
2.5	Visualization of how the discretization is an approximation of the actual geometry. The same principle applies to the meshing of complex geometries in STAR-CCM+.	13
3.1	CAD model of the front suspension before and after simplifications. .	18
3.2	CAD model of the rear suspension before and after simplifications. . .	18
3.3	CAD model of the section below the rear wing before and after simplifications.	19
3.4	CAD model of the front wing before and after it being mounted to the monocoque.	20
3.5	CAD model of the two different front wings used in the tests.	20
3.6	A view of the front wing modularity.	21
3.7	CAD model of the reinforced structure separated and attached to the front wing.	21
3.8	Front view of the rear wing before and after being mounted to the rest of the car.	22
3.9	Rear view of the rear wing before and after being mounted to the rest of the car.	22
3.10	CAD model of the rear wing and main hoop assembly before and after it being made modular.	23
3.11	CAD model of the hole in the monocoque for the driver and also the driver model.	23
3.12	The components below the rear wing.	25
3.13	The components below the rear wing as they will be 3D-printed. . . .	26
3.14	CAD model of the sidepods showing the extrusion for a later connection to monocoque.	26
3.15	CAD model of the hole for the rods and also showing the placement of the two rods when added to the car.	27
3.16	CAD model of the two front wing halves that later will be connected.	27
3.17	A complete view of the assembly of the wheels to the axles.	28

3.18	A view of the monocoque with all the holes and features.	28
3.19	Model of the wind tunnel used in the CFD-simulations showing the dimensions of the tunnel and the car.	31
3.20	Model of the wind tunnel used in the CFD-simulations showing the types of boundaries on the wind tunnel surfaces. The unnamed surfaces are symmetry planes.	32
3.21	A visual representation of the various volumetric refinements used in the simulations. The car is displayed in blue.	34
3.22	Views of the cars used in the reference simulation and the simulation of the simplified car.	35
3.23	Views of the cars used in the the reference simulation and the simulation of the simplified car.	35
3.24	Visualisation of the workflow regarding the 3D-printing.	36
3.25	A view of the half-completed 3D-printed car. In this state, the car is missing the sidepods, both wings and the optional driver.	37
3.26	A view of the completed 3D-printed car. In this figure, the car is displayed with the original front wing and without the driver.	38
3.27	A picture of the 3D-printed car model showing the front wing supports in red.	39
3.28	Custom mounting platform and jig used to secure the scale model in the wind tunnel.	40
3.29	Different configurations of the scale model tested in the wind tunnel.	41
4.1	A final assembly of the complete simplified car.	43
4.2	Mesh study showing variation of lift and drag coefficients with mesh resolution.	44
4.3	The mesh used in the STAR-CCM+ simulations at different zoom-levels.	45
4.4	Front views of the pressure coefficient distribution of the simplified car and the car with a driver.	46
4.5	Top views of the pressure coefficient distribution of the simplified car and the car with the driver.	47
4.6	Bottom and rear view of the pressure coefficient distribution of the simplified car. Note that the scale is different compared to figure 4.5 to highlight the differences.	47
4.7	Velocity scenes of the four configurations of the car.	48
4.8	A streamline scene of the simplified car showing the path the air takes around the car.	49
4.9	An isometric view of the car with the tilted wing showing areas with high-energy losses.	49
4.10	A figure showing the accumulated Cd for the different cars.	50
4.11	A figure showing the accumulated Cl for the different cars.	50
5.1	A velocity scene showing the acceleration of the flow when the clearance gets smaller.	56

List of Tables

3.1	Physics Models Used in STAR-CCM+ Simulation.	29
3.2	Boundary Conditions Used in STAR-CCM+ Simulation.	32
3.3	Non 3D-printed parts of the model.	37
4.1	Mesh study results.	44
4.2	Forces and coefficients for the different cars in CFD.	46
4.3	Forces and coefficients of the 3D-printed model (corrected values). . .	51
4.4	Comparison of aerodynamic coefficients between wind tunnel (WT) and CFD.	51
4.5	Comparison of aerodynamic forces between wind tunnel (WT) and CFD. All units are in Newtons [N].	51

1

Introduction

Aerodynamics plays a central role in the competitiveness of a Formula Student car, with properties such as downforce and drag playing a major role in car performance. As vehicle designs become more complex, the need for reliable aerodynamic evaluation methods increases. A primary tool for aerodynamic development has become Computational Fluid Dynamics (CFD), but experimental validation is important to ensure credibility and to identify limitations. For teams operating on limited time and resources, establishing an efficient methodology that includes numerical and physical tests is of significant value to ensure the best result in competition. Considering the need for reliable and efficient aerodynamic evaluation, this chapter details the background, purpose, goals, and limitations of the project regarding Chalmers Formula Student (CFS).

1.1 Background

This section will provide some background to the thesis. This consists of some general information on Formula Student, more specific information on Chalmers Formula Student team, as well as some background regarding the integration of CFD and wind tunnel testing.

1.1.1 Formula Student

Formula Student (FS) is a competition in engineering where students from universities all over the world develop and build formula racing cars. The competition has been going on since 1979, when it was first arranged by *Society of Automotive Engineers*. Since then, FS has grown to include over 600 universities [1].

There are two main categories of competition in FS, *Dynamic* and *Static*. The dynamic category includes the parts of the competition related to the driving of the car. This includes "Acceleration", "Skid Pad ", "Autocross", "Efficiency" and "Endurance". These sub-competitions are meant to test the cars driving abilities. The static part of the competition is comprised of "Business Presentation", "Cost Report" and "Design" which are meant to test the engineering and business side of FS. Aside from this, there is also a driverless cup where the cars compete in autonomous driving [1].

1.1.2 Chalmers Formula Student

Chalmers University of Technology has an FS team, Chalmers Formula Student, whose car (figure 1.1) has achieved notable success in the past years. The team designs and manufactures the majority of the car in-house with a focus on carbon fiber frames, suspension, mechanical drivetrains, and aerodynamics. The current way of testing the aerodynamics of their car is a combination of Computational Fluid Dynamics (CFD) with occasional wind tunnel visits at Volvo Cars. When trying to figure out how to make the aerodynamic analysis more robust and accurate, CFS realized that a scale-model car in combination with their access to the Chalmers wind tunnel could be a great asset. This insight is the primary motivation for this study and its purpose [2].



Figure 1.1: The 2025 CFS car [3].

1.1.3 Integration of CFD and Wind Tunnel Testing

CFD has many advantages over traditional wind tunnel testing. One of these is the ability to take measurements of the flow anywhere in the domain without affecting the flow itself. In real-world tests, measurements are always done at discrete points, and the act of measuring often affects the flow (intrusive measurements), rendering the data less accurate. Furthermore, CFD enables simulation of the vehicle in true on-road conditions, including a moving reference frame, whereas wind-tunnel experiments require the model to be mounted using support structures that inevitably introduce flow interference [4]. In addition, CFD allows for cheap and rapid prototyping and evaluation of multiple design iterations, whereas wind-tunnel testing requires the fabrication of a physical model beforehand, substantially increasing the time required for each modification [5].

Although the advantages of CFD are plentiful, there are disadvantages. One of these is the computational cost. Borg [6] report that a complete aerodynamic CFD simulation of a vehicle may take as long as 72 hours, whereas a wind-tunnel test using an existing model can be conducted in substantially less time. The accuracy of a CFD simulation is highly dependent on user input and modeling choices. Errors in geometry preparation, inappropriate boundary conditions, inappropriate physical models, or incorrect assumptions can all introduce significant inaccuracies in the results [7]. Consequently, reliable CFD analysis requires a high level of expertise from the user, particularly in fluid dynamics and numerical modeling. In addition, CFD methods still struggle to precisely resolve certain complex flow phenomena. One notable limitation is the prediction of boundary-layer transition, which remains challenging to model accurately due to its sensitivity to disturbances and modeling assumptions [8].

Since both wind tunnel testing and CFD have advantages and disadvantages, a combination of the two could be a way to provide a better understanding of the aerodynamics of an FS car. Acquiring this understanding would then aid the team in developing a competitive car.

1.2 Purpose

The purpose of this project is to develop a simplified yet aerodynamically representative scale model of the Chalmers Formula Student car using a modular design approach. This model will be 3D-printed and tested in the Chalmers wind tunnel. The model is intended to support ongoing vehicle development by enabling efficient evaluation of aerodynamic performance while maintaining sufficient accuracy in representing key characteristics such as drag and lift.

1.3 Goals

The goals of this project are to:

- Develop an accurate, modular scale model capable of representing the full-scale Chalmers Formula Student car.
- Compute the aerodynamic coefficients, such as the drag coefficient (C_d) and lift coefficient (C_l), for the scale model using CFD simulations as well as wind tunnel testing.
- Compare the results obtained from CFD simulations with those from wind tunnel experiments, doing this for different configurations.
- To propose design improvements for the CFS car based on a combined analysis of CFD simulations and wind tunnel measurements.

1.4 Limitations

Due to the complex nature of the work in this thesis, several limitations were identified. These are presented in this section.

The scale model will be 3D-printed in plastic. Although carbon fiber (used in the full-scale car) is stiffer, the aerodynamic loads on the scaled model in the wind tunnel are significantly smaller than on the full-scale car. This means that the loads and deformations will be different. The intention is therefore not to match the stiffness of the carbon fiber structure relative to full-scale loads but to try to achieve small aeroelastic deformation during testing. Using carbon fiber or another material would increase the project timeline, would be outside the expertise of the group, and would make it more difficult to modify components in preparation for wind tunnel tests.

Another limitation of this study is the use of Reynolds-Averaged Navier–Stokes (RANS) equations in the CFD simulations. This provides a time-averaged representation of the flow field, meaning that unsteady turbulent structures are not fully resolved. While this approach makes the simulations feasible within the project’s time constraints, it also introduces limitations in accurately capturing complex flow phenomena, such as flow separation and vortex shedding. As a result, the simulations represent a steady-state approximation of the flow rather than its true behavior, which may lead to discrepancies when compared to real-world data.

The use of a stationary ground and non-rotating wheels in the CFD simulations is also a limitation of this study. This simplification was made to reduce model complexity, but it affects the accuracy of the results, as the interaction between rotating wheels and a moving ground influences the airflow around the car, particularly in the boundary layer of the ground and wake regions. By not accounting for these effects, the simulations may not fully capture the true aerodynamic behavior, which can lead to discrepancies compared to real-world conditions.

Finally, the model used in the CFD analysis and the physically printed model will differ in surface quality. This is because the digital model has an idealized, perfectly smooth surface, whereas the physical model exhibits layer lines and surface roughness resulting from the 3D printing process. However, due to the meshing, the surface used in CFD will not be perfect. It will however be different from the 3D-printed one. These differences may lead to discrepancies between the two methods; however, the group has chosen not to account for them, as doing so would expand the scope of the project beyond the available time and resources.

2

Theory

The following section presents all the necessary theory needed to understand this thesis.

2.1 Fluid Dynamics Theory

The following chapter will cover all the necessary fluid dynamics theory needed to understand the thesis.

2.1.1 Governing Laws and Equations

In this section, the governing laws and equations of fluid dynamics are covered.

2.1.1.1 Continuity Equation

The continuity equation represents the principle of mass conservation. It states that any change in mass within a control volume is equal to the net mass flux entering or leaving the system, under the assumption that no mass is generated or consumed within the flow. In differential form, the continuity equation is written as

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho u) + \frac{\partial}{\partial y}(\rho v) + \frac{\partial}{\partial z}(\rho w) = 0, \quad (2.1)$$

where ρ is the fluid density, and u , v , and w are the velocity components in the x , y , and z directions, respectively.

For a control volume, the same law can be expressed in integral form as

$$\dot{m}_{in} - \dot{m}_{out} = \frac{\partial}{\partial t} m_{CV}, \quad (2.2)$$

meaning that the difference between incoming and outgoing mass flow rates equals the time rate of change of mass within the control volume. This condition is fundamental and is used in nearly all analyses in fluid dynamics [9].

2.1.1.2 Conservation of Momentum and Navier Stokes Equations

Conservation of momentum is a concept in fluid dynamics which means that the sum of body forces acting on a fluid control volume equals the rate of change of momentum within that control volume [10]. By applying the conservation of momentum to a differential fluid element and incorporating the effects of viscosity, the

Navier-Stokes equations (2.3) can be derived. These equations explain the motion of all particles in a flow of a Newtonian fluid.

$$\rho \left(\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} \right) = -\nabla p + \mu \nabla^2 \mathbf{u} + \mathbf{f} \quad (2.3)$$

The term $\rho \partial \mathbf{u} / \partial t$ represents the *local acceleration*, describing the rate of change of momentum with time at a fixed point. The nonlinear term $\rho(\mathbf{u} \cdot \nabla) \mathbf{u}$ is the *convective acceleration*, which accounts for momentum transport due to the fluid motion itself. On the right-hand side, $-\nabla p$ denotes the *pressure gradient force*, $\mu \nabla^2 \mathbf{u}$ represents the *viscous diffusion of momentum* arising from the fluid's viscosity, and $\mathbf{f}$ includes any *body forces* such as gravity.

Because the Navier-Stokes equation is non-linear, analytical solutions are rare, and numerical solutions using CFD are used for practical engineering applications [11].

2.1.2 Dimensionless Numbers

This section presents the various dimensionless numbers used throughout the thesis.

2.1.2.1 Reynolds Number

Reynolds number, equation (2.4), describes the ratio of inertial and viscous forces in a fluid. This allows the Reynolds number to be used when determining if a flow is laminar, turbulent, or in the transition region [12]. When calculating the Reynolds number of flow over a car, the wheelbase is commonly used as the characteristic length.

$$Re = \frac{\rho V L_c}{\mu} \quad (2.4)$$

In this equation μ is the dynamic viscosity of the fluid which is a measure of the fluid's internal resistance to deformation under shear stress.

2.1.2.2 Lift Coefficient

The lift coefficient, equation (2.5), is a number that is used as a way to measure the lift of a body. C_L is a dimensionless number that includes the effects of "shape, inclination, and some flow conditions on lift" [13].

$$C_L = \frac{F_L}{\frac{1}{2} \rho U_\infty^2 A} \quad (2.5)$$

Where A is the frontal projected area of the object.

2.1.2.3 Drag Coefficient

The drag coefficient, equation (2.6), is a dimensionless number that represents the effect that factors such as geometry and position relative to the flow have on drag

[14].

$$C_D = \frac{F_D}{\frac{1}{2} \rho U_\infty^2 A} \quad (2.6)$$

2.1.2.4 Pressure Coefficient

The pressure coefficient (C_p) is a dimensionless parameter that relates the local static pressure to the freestream dynamic pressure, allowing pressure variations across the flow field to be compared [15]. The equation below (2.7) shows how the pressure coefficient is calculated.

$$C_p = \frac{p - p_\infty}{\frac{1}{2} \rho V_\infty^2} \quad (2.7)$$

2.1.3 Boundary Layers

This section introduces the necessary boundary layer theory needed for this thesis.

2.1.3.1 No Slip Condition

As Weyburne [16] states, the no slip condition is a type of boundary condition which says that the fluid velocity always has the same magnitude and direction as the velocity of the solid. The consequence of this is that the fluid velocity has to slow down as it gets closer to the wall and forms a gradient. This gradient of velocity near the wall is called the boundary layer. This happens on all surfaces in real life.

2.1.3.2 Boundary Layers

As stated in section 2.1.3.1, the boundary layer is the gradient of velocity that forms near a wall due to the no slip condition. The boundary layer thickness δ is defined as the height above the wall at which the speed in the boundary layer u reaches 99% of the freestream velocity u_∞ (or u_0) [16]. See figure 2.1 for a visualization of a boundary layer.

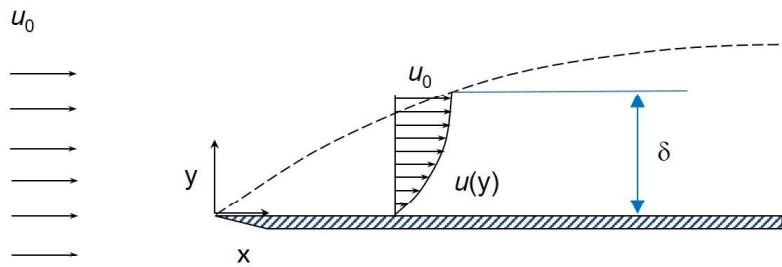


Figure 2.1: Boundary layer development over a flat plate [17] (CC0 1.0).

2.2 Wind Tunnel Testing

This section of the theory covers wind tunnel theory.

2.2.1 Similarity Principles of Scale Testing

To ensure that measurement data obtained from a scale model test in a wind tunnel are representative of the full-scale configuration, three essential similarity principles must be satisfied: geometric, kinematic, and dynamic similarity. These principles are introduced and discussed in the following subsections.

2.2.1.1 Geometric Similarity

When testing models at different scales, it is important that all dimensions are scaled by the same factor. This is what is known as geometric similarity. Geometric similarity ensures that two models have the same shape, even if the size differs, and is essential for accurate scale testing. See figure 2.2 for a visualization of geometric similarity. When geometric similarity is satisfied, all corresponding lengths in the model and the full-scale prototype have a constant ratio [18]. Geometric similarity is a prerequisite for both kinematic and dynamic similarity, covered in sections 2.2.1.2 and 2.2.1.3.

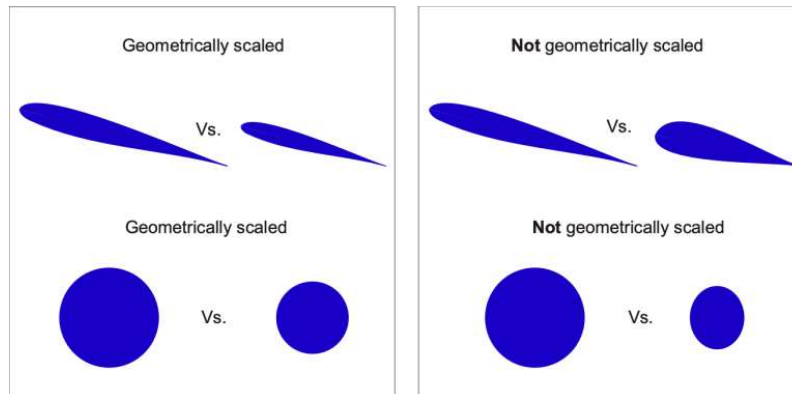


Figure 2.2: Illustration of geometric similarity [18] (CC BY-NC-ND 4.0).

2.2.1.2 Kinematic Similarity

Kinematic similarity refers to the similarity of the motion of the fluid between a scale model and a full-scale prototype. This condition is satisfied when the velocities at corresponding points in the two flows are proportional in magnitude and have the same direction. As a result, the flow patterns, including streamlines and particle paths, are geometrically similar between the model and the prototype [18].

2.2.1.3 Dynamic Similarity

Dynamic similarity is the concept of matching key dimensionless numbers in different conditions to ensure that the flow is equal for the prototype model and full scale

model. If dynamic similarity is ensured between two trials, all the physics governing the flow will be scaled correctly between the two cases, which is crucial for obtaining useful data. In wind tunnel testing, Reynolds number, see section 2.1.2.1, and Mach number are the most important dimensionless numbers to match [18]. In this thesis, the velocities are sufficiently low that compressibility effects can be neglected. Consequently, Mach number effects are not considered and dynamic similarity could instead be achieved by matching the Reynolds number between cases.

2.2.2 Wall Interference

In real-world environments, walls close to and around the vehicle are rare. In wind tunnel testing however, walls are always present and frequently interfere with the aerodynamics. Those effects are covered in this section.

2.2.2.1 Blockage

Blockage effects occur due to a body in a wind tunnel occupying a significant portion of the test section's cross-sectional area. This causes streamlines to curve around the body which changes the flow, see figure 2.3. When the streamlines change, so does the pressure field and thus the measurements taken on the body which makes blockage effects important to consider [18]. The primary mechanism behind blockage is the confinement of the flow within the wind tunnel, which, due to continuity, forces the air to accelerate around the model. According to Bernoulli's principle, this increase in velocity leads to a reduction in static pressure, thereby affecting the measured aerodynamic characteristics.

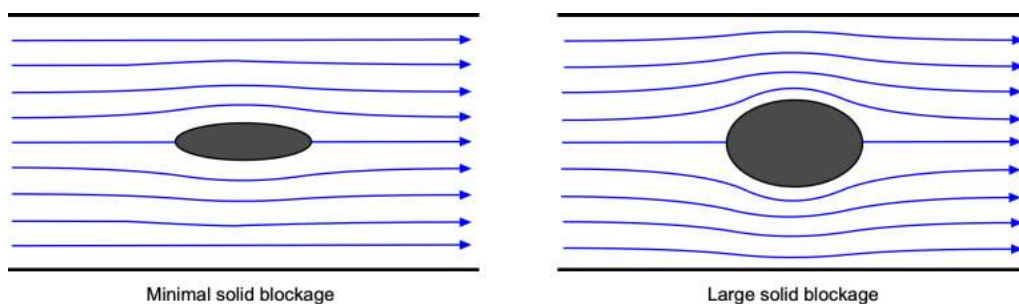


Figure 2.3: Visualization of blockage effects in wind tunnels [18] (CC BY-NC-ND 4.0).

Effects of blockage start to occur if the blockage ratio Λ exceeds 0.05. The ratio is calculated as $\Lambda = \frac{A_{ref}}{A_t}$ where A_{ref} is the projected frontal area of the body and A_t is the test section area of the wind tunnel [18].

To reduce the effects of blockage on the measurements, a blockage correction formula is often used. This formula provides correction factors to convert the measured coefficients such as C_d and C_l into values that better represent real-world conditions

without walls. At the Chalmers wind tunnel, equation 2.8 below is used, [19].

$$Coeff_{true} = Coeff_{measured} \cdot \left(1 - \frac{A}{S}\right)^{1.288} \quad (2.8)$$

Where:

- A is the projected frontal area of the car
- S is the cross-sectional area of the wind tunnel test section

2.2.3 Measurement Techniques

This section covers the theory regarding the measurement techniques used to measure forces on a model in a wind tunnel.

2.2.3.1 Balance Measurements

A force balance is a type of instrument that allows for measurements of forces in three directions as well as corresponding moments. This gives a complete picture of the forces and moments acting on the balance [18]. This is the type of measurement apparatus present in the Chalmers wind tunnel.

2.3 CFD-Theory

Computational fluid dynamics (CFD) uses numerical analysis and computers to solve and analyze problems that involve flows; these flows can be fluids and gases. CFD simulations involve using the commonly applied equations of fluid mechanics and fundamental laws of mechanics to formulate a problem mathematically [20].

2.3.1 Finite Volume Method (FVM)

The finite volume method (FVM) is a common approach in CFD and used in STAR-CCM+. The theory behind FVM being used in CFD is that the governing partial differential equations (Navier-Stokes, mass and energy conservation) are being integrated and solved over a control volume, where these control volumes are the result of the domain being divided into small cells/elements where each cell is a control volume, see 2.4, [21]. Solving the equations in a defined control volume is comparatively easy. Thus, solving complex problems using many control volumes to represent the geometry is an efficient way to solve these equations.

2.3.3.2 The k- ω Model

This model is similar to the previous model in that, just like before, it is the modelled k equation that is solved together with a dissipation equation. However, this time the specific dissipation ω is used, which gets its name from its definition: ω is proportional to ε/k [22]. This model works better at near wall-boundaries and is also a common model used for CFD simulations.

2.3.3.3 The SST k- ω Model

The SST k- ω model is the one primarily used during this study and is a combination of the two models previously presented in chapters 2.3.3.1 and 2.3.3.2. The model uses the function, F_1 , shown in equation 2.10 to ensure that the k- ω model is used near walls and the k- ε model is used in free streams because those regions are where the respective models have their advantages [23].

$$F_1 = \tanh\left\{\left\{\min\left[\max\left(\frac{\sqrt{k}}{\beta^*\omega y}, \frac{500\nu}{y^2\omega}\right), \frac{4\sigma_{\omega}k}{CD_{k\omega}y^2}\right]\right\}^4\right\} \quad (2.10)$$

2.3.4 Discretization Error and Convergence

As described in section 2.3.1, the domain on which you want to solve your complex equations is discretized into many small cells according to the FVM. This is called a discretization (meshing in this thesis) of the domain and naturally, the discretization error is the error that is caused by this discretization. The reason that an error occurs is that the discretization is an approximation of the actual geometry. A visualization of this is presented in figure 2.5. In order to minimize the discretization error, you increase the resolution of the mesh so that the discrepancy between the meshed model and the actual geometry decreases. This is why fine meshes produce more accurate results [24].

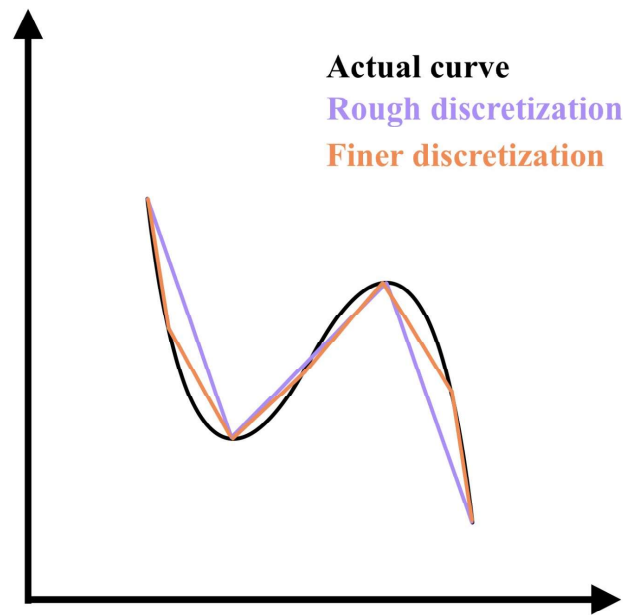


Figure 2.5: Visualization of how the discretization is an approximation of the actual geometry. The same principle applies to the meshing of complex geometries in STAR-CCM+.

Another error present in simulations using FVM is iterative errors. These errors are the result of a non-converged solution. When running a steady-state simulation as in this thesis, the software iterates through time steps until it has reached a steady flow field which does not evolve further. This means that the solution has converged and the result can be used. Using the values of a non converged solution means that the flow field is not yet stable and not representative of the actual flow that would occur in real life. To ensure that convergence has been reached, residuals as well as properties such as drag are examined to see whether they are stationary regarding the iterations which would imply convergence [25]. The residuals of the simulation are the error in each cell regarding how well they satisfy the governing equations such as continuity and are thus critical in determining convergence as low residuals imply that the solution is adhering to the governing physical laws [26].

2.3.4.1 Mesh Independence

A simulation can converge even if the mesh is inadequate in correctly approximating the geometry. This means that the solution will be incorrect for the actual geometry. When the mesh is detailed enough that refining it further will not make the results change, the solution has become mesh independent. This means that the mesh is not the limiting factor in ensuring accurate results. Achieving mesh independence can be done by conducting a mesh study where different mesh sizes are tested to see whether they produce the same results. If they do not, you refine the mesh until refining it further does not change the result [27].

2.3.5 Boundary Conditions

In CFD, there is always a finite domain in which the calculations occur. Because of this, the boundaries of the domain have to be defined according to what should happen there, because in real life, the domain never has a boundary. This definition of what is supposed to happen in these boundaries is called boundary conditions. You choose these boundary conditions to represent a real world domain without borders as accurately as possible [28]. The subsections below will go through some common boundary conditions used in CFD.

2.3.5.1 Velocity Inlet

A velocity inlet boundary condition is used when the flow rate into the domain is known. The velocity can then be assigned and other parameters such as pressure will be calculated from this [29].

2.3.5.2 Pressure Outlet

As da Silva Andrade Sarah [29] writes, a pressure outlet is used when the pressure is specified or known. Velocity will then be calculated using the pressure.

2.3.5.3 No Slip Wall

A no-slip wall boundary imposes the no-slip condition (2.1.3.1) on the surface [29]. As a result, boundary layers develop along the surface.

2.3.5.4 Slip Wall

A slip wall boundary implies that the fluid particles slide with zero friction along the surface. They do not slow down due to the presence of the wall which leads to no buildup of boundary layers. The velocity profile will not change as the distance to the wall decreases [30].

2.3.5.5 Symmetry Plane

An symmetry boundary condition is used when the flow field is assumed to be identical on either side of a symmetry plane. This allows the computational domain to be reduced by half, with the solution mirrored across the plane to reconstruct the full domain [31].

2.4 Additive Manufacturing

Additive manufacturing is a manufacturing technique in which complex geometries are created by building them layer by layer. There are many different types of techniques for doing additive manufacturing, but one of the most common and the one used in this thesis is 3D-printing using polymers.

2.4.1 Basic Principle of 3D-printing

3D-printing is a subtype of additive manufacturing called fused deposition modeling, or FDM for short. It works by extruding a heated polymer through a small nozzle. The nozzle is programmed to move to certain positions which creates the layers. To create the next layer, the nozzle moves up a step (layer height) and deposits material on top of the last layer. This goes on until the desired geometry is created [32]. Common materials used are PLA (polylactic acid) and PETG (polyethylene terephthalate glycol).

In order to go from a CAD-file to a physical 3D-printed object, you follow the workflow described in Zhou et al. [32]. The first step after the CAD-modeling is done is to convert the CAD-model to a stereolithography (STL) file. This converts the model into a tessellated one, which slicing software can interpret. The slicing software converts the STL file into many thin connected layers. These are the layers that the 3D-printer later will extrude. The last step is to generate G-code which is the "language" of the 3D-printer. When given G-code, the 3D-printer will construct the geometry associated with it.

2.4.2 3D-printer Settings

In the slicer software, there are many parameters to tune to achieve the desired result of the 3D-print. This section covers those parameters.

2.4.2.1 Layer Height

The layer height is the thickness of each layer in the 3D-printed model. This is directly linked to the quality and resolution of the finished 3D-print. This value is often 10% – 75% of the nozzle diameter (section 2.4.2.2). Lower values equal better quality prints with an increase in printing time while higher values accelerate the printing process but lower the quality [33].

2.4.2.2 Nozzle Diameter

The nozzle diameter is the size of the hole which the polymer gets extruded through. This determines the layer height, layer width and thus also the quality and strength of the print. The most used nozzles have a diameter of $0.2\text{mm} - 0.6\text{mm}$, where smaller nozzles result in longer print time but better quality. [33].

2.4.2.3 Infill

To reduce weight and material usage while still ensuring good structural stability, 3D-printers do not usually produce solid objects. The inside of the geometries are instead partially filled where the percentage of the filled volume to the unfilled volume is called infill. This ranges from 0% (hollow) to 100% (solid). The infill is distributed according to a geometric pattern, where grids, triangles and gyroids are common ones [33].

2.4.2.4 Supports

To be able to 3D-print complex geometries with overhangs, support material is needed. The support material is like a framework on which 3D-printing of the geometry can occur when there is overhang. There are many different types of supports like grid structures and organic structures [33].

2.4.3 Print Orientation

Print orientation refers to the direction in which a geometry is printed. Since 3D printing is based on a layer-by-layer extrusion process, the orientation of these layers significantly affects the final surface quality of the structure. It also has a major influence on mechanical strength, as printed parts are generally stronger when loads are applied parallel to the layer lines rather than perpendicular to them, where inter-layer bonding is weaker [33].

3

Methods

3.1 CAD

This section presents what CAD-work was done in this thesis. The CAD-software used was Siemens NX. In this part of the thesis, parts that are red have been removed, while green parts have been added.

3.1.1 Simplifications

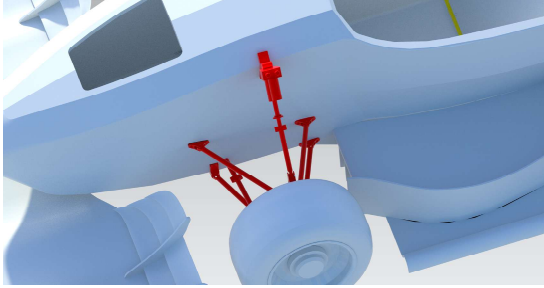
A complete CAD assembly of the CFS race car was provided by the CFS-team, which served as the starting point for the work. To meet the requirement of making the front and rear wings modular and at the same time produce a 3D-printed scale model that could withstand the loads of wind tunnel testing within the available time frame, several modifications to the original geometry were necessary.

The geometrically complex components, such as the suspension and the steering parts below the rear wing, were simplified. These components would otherwise have been difficult to print and would also have increased the complexity of the wind tunnel testing due to their delicate nature. In addition, certain cavities and small internal features in the original CAD model were filled or removed to make the 3D printing process more reliable and to improve the strength of the printed parts. All simplifications were made with consideration to preserve the main flow features and make the model suitable for testing.

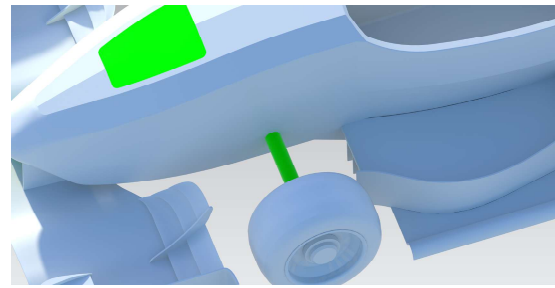
The CFS team's digital CAD-model included a complex suspension setup, which would have made it difficult to 3D print and test in the wind tunnel without risking structural failure. Thin and detailed components could potentially break during testing and, in the worst case, cause damage to the wind tunnel. To ensure structural stability and printability, a simplified suspension system was developed.

This process began by modifying the wheel hubs of the front and rear wheels. The cylindrical section inside the hub was expanded to create a more robust structure. It was further simplified by introducing a structural beam running from one wheel through the monocoque and connected to the opposite wheel, see figures 3.1 and 3.2 for front and rear views respectively. This solution ensured that the wheels remained stable during wind tunnel testing and also allowed the model to be securely mounted to a turntable using threaded rods.

The remaining suspension members, such as the wishbones and pushrods, were also simplified. The original complex geometry was removed entirely, see figures 3.1 and 3.2 for front and rear views respectively. This is a major simplification that was justified by CFD simulations of its effects as well as by how much it simplifies printing and assembly.

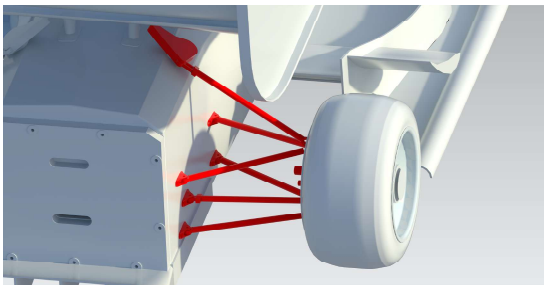


(a) Front suspension before simplifications. The red parts will be removed in the simplification.

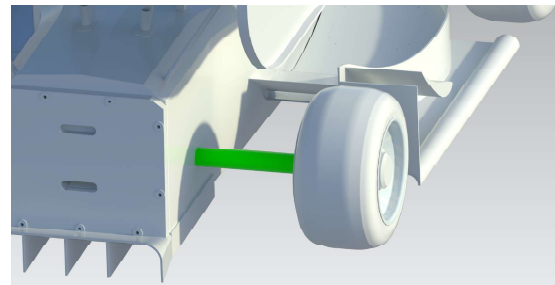


(b) Front suspension after simplifications. Suspension rods have been removed, and a beam through the wheels has been added, as indicated by the green. Also, the monocoque hole has been filled as seen by green.

Figure 3.1: CAD model of the front suspension before and after simplifications.



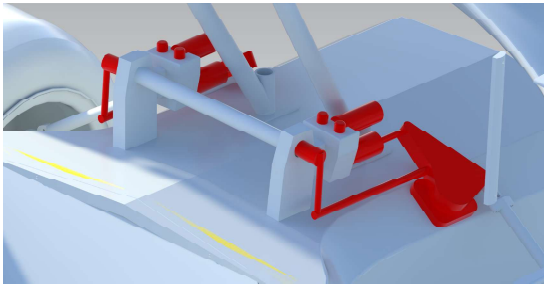
(a) Rear suspension before simplifications. The red parts will be removed in the simplification.



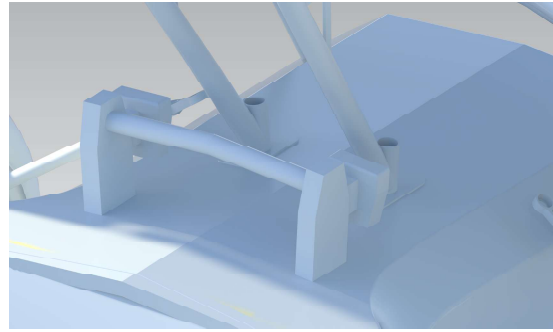
(b) Rear suspension after simplifications. Suspension rods have been removed, and a beam through the wheels has been added as indicated by the green part.

Figure 3.2: CAD model of the rear suspension before and after simplifications.

The steering components located on top of the car behind the main hoop also featured complex geometry in the original digital model. These components were simplified by removing certain detailed elements and replacing them with more simple geometric shapes that resemble the appearance of the real car while being more suitable for 3D printing and wind tunnel testing, see figure 3.3.



(a) Section below the rear wing before simplifications. The red parts will be removed.



(b) Section below the rear wing after simplifications. Some parts have been removed and some parts have been smoothed out.

Figure 3.3: CAD model of the section below the rear wing before and after simplifications.

The sidepods of the Formula Student car included a cooling fan in the original CAD model, which would have made the 3D printing process more complicated. To simplify the manufacturing, the detailed fan geometry was removed and replaced with a solid block having the same overall dimensions and position. This allowed the external aerodynamic shape to be preserved while avoiding unnecessary printing complexity.

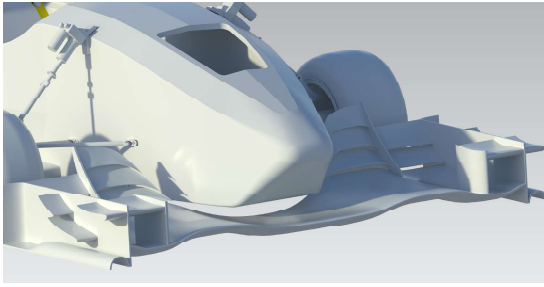
Further simplifications were made to improve the 3D printing process. The digital CAD model provided by the CFS team included an open cavity in the front section of the car. This opening was closed in the modified version of the model, see figure 3.4. The same area is also closed on the real-world racing car, which made this modification both practical and realistic for the printed model.

3.1.2 Modular Components

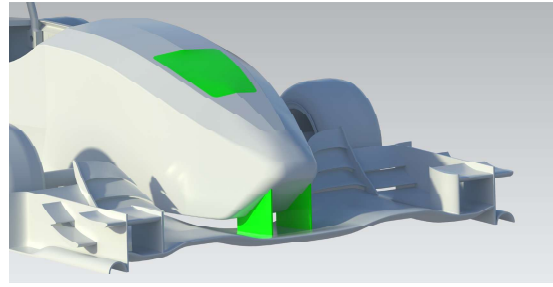
This subsection describes the process of making the front wing, rear wing and driver modular components.

3.1.2.1 Front Wing

The front wing, which was one of the components detached from the monocoque in the CAD file provided by CFS, was attached by adding two brackets to the nose cone and connecting them to the wing, see figure 3.4.



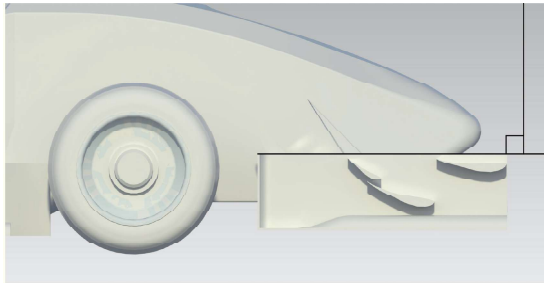
(a) The front wing before being mounted to the monocoque. The hole in the monocoque is unfilled.



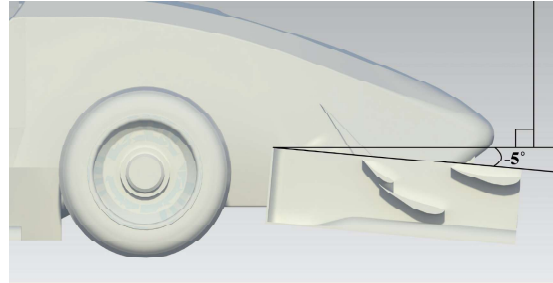
(b) The front wing after being mounted to the monocoque. Two brackets connecting the front wing to the nose cone have been added. The hole in the monocoque has also been filled. These additions can be seen by the green coloring.

Figure 3.4: CAD model of the front wing before and after it being mounted to the monocoque.

Since the wing was designed to be modular, a second version of the wing was also tested. This would allow for testing of the modularity system as well as give more data to analyze. This version of the wing differed from the original by being tilted 5° downwards, see figure 3.5.



(a) Side view of original wing setup.



(b) Side view of tilted second wing setup version. The wing has been tilted 5° downwards.

Figure 3.5: CAD model of the two different front wings used in the tests.

To comply with CFS's requirement of making the front wing modular, a T-slot connection was designed between the front wing and the monocoque. The T-rail was created on the monocoque, while the corresponding T-slot was added to the front wing, see figure 3.6. By using this sliding connection, the front wing can be mounted and removed in a simple and repeatable manner while maintaining a stable position relative to the rest of the model. This solution also ensures that the connection can withstand the aerodynamic loads during wind tunnel testing without failure.

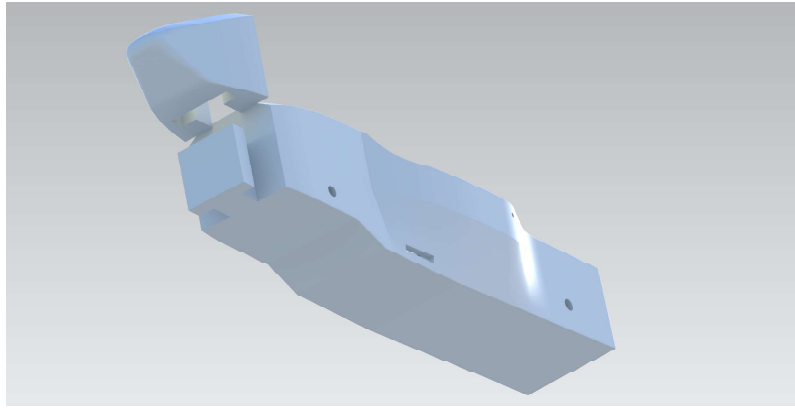
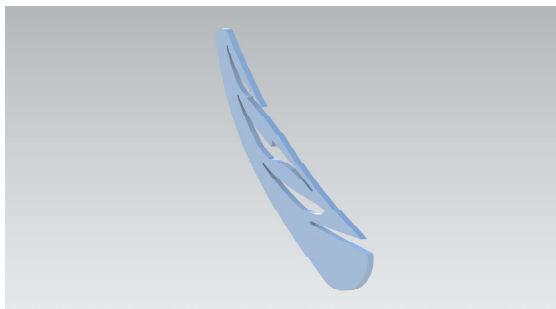
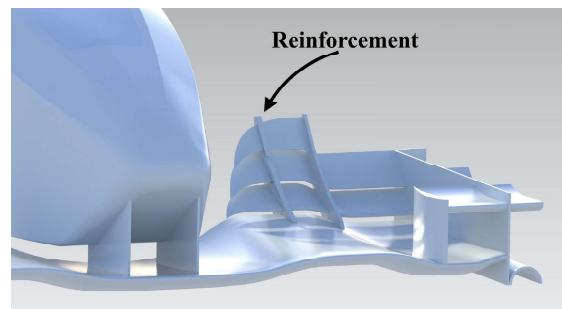


Figure 3.6: A view of the front wing modularity.

To ensure sufficient structural stability of the wing, a reinforcing structure was added to the smallest wing segments which can be seen in figure 3.7. You can also in figure 3.7 see the wing segments with the reinforcement added.



(a) Reinforcing structure.

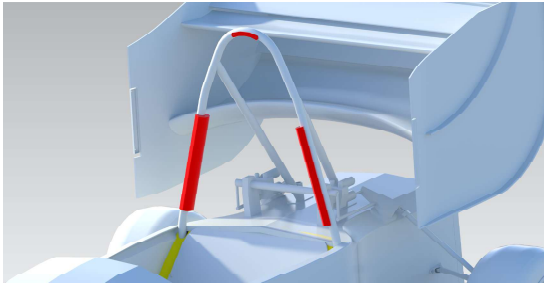


(b) The reinforcement added to the front wing.

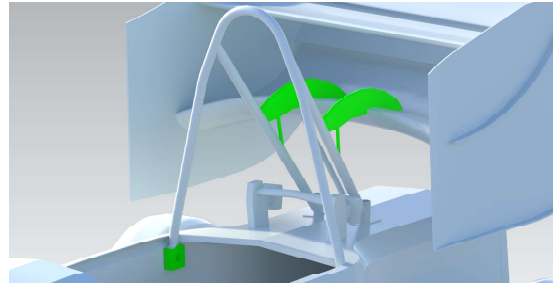
Figure 3.7: CAD model of the reinforced structure separated and attached to the front wing.

3.1.2.2 Rear Wing

Like with the front wing, the rear wing was not attached to the rest of the car in the CAD-file provided by CFS. To solve this, the rear wing was connected to the main hoop by two supporting structures positioned behind the roll cage and with two struts connecting the wing to the monocoque. See figure 3.8 and 3.9 for the mounting of the wing. By attaching the rear wing in this way, the overall stiffness of the rear wing was improved, ensuring that it could withstand aerodynamic loads during testing and maintain its original position, similar to the full-scale car.

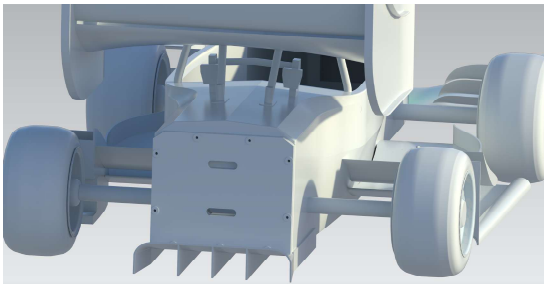


(a) Front view of the rear wing before being mounted. The red indicate the parts that were removed during the CAD work.

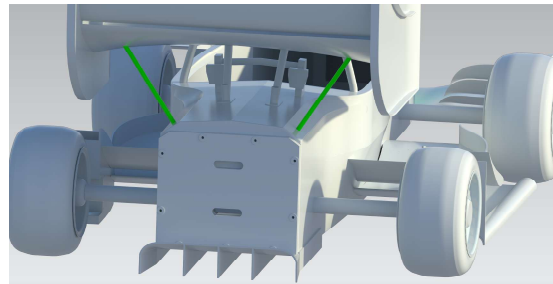


(b) Front view of the rear wing after being mounted. Two brackets connecting the rear wing to the main hoop have been added as seen by the green parts. The rear wing modularity has also been implemented.

Figure 3.8: Front view of the rear wing before and after being mounted to the rest of the car.



(a) Rear view of the rear wing before being mounted.



(b) Rear view of the rear wing after being mounted. Two rods connecting the wing to the monocoque have been added as seen by the green color.

Figure 3.9: Rear view of the rear wing before and after being mounted to the rest of the car.

To allow the rear wing to remain modular and interchangeable, the rear wing and rollbar assembly was made as a single detachable piece. Then the wing could be removed by removing the roll cage using screws, see figure 3.10. Two screws connect the main hoop to the monocoque where they meet below the rear wing and two additional screws connect them behind the driver in the cavity of the monocoque. These locations were selected because they do not influence the external aerodynamic surfaces or the structural integrity of the model. By introducing the separation of the parts at this point, the rear wing can be attached and detached more easily and without compromising the aerodynamics of the model.

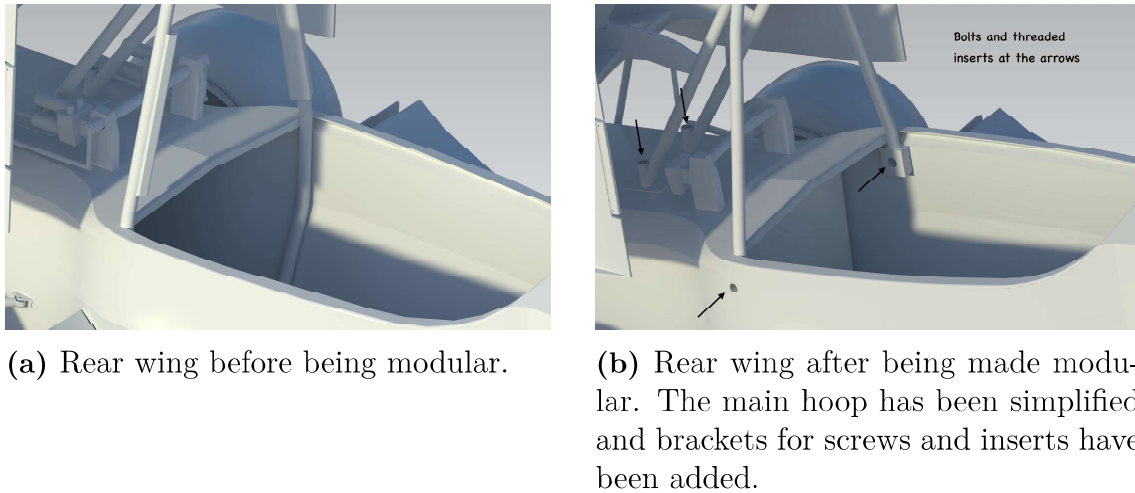


Figure 3.10: CAD model of the rear wing and main hoop assembly before and after it being made modular.

3.1.2.3 The Driver

Since FS competitions include both driverless events and events with a driver. A model of a driver was chosen to be modular. This would allow for testing the aerodynamics of both the driverless and driver events. A CAD-file featuring a model of the driver was provided by CFS. The driver was modified by creating a long square extrusion on the base of it, see figure 3.11. This extrusion allows for the driver to be inserted into a hole in the monocoque (figure 3.11) for easy installation and removal. The length of the hole and the nature of the forces on the driver will prevent it from flying into the wind tunnel.

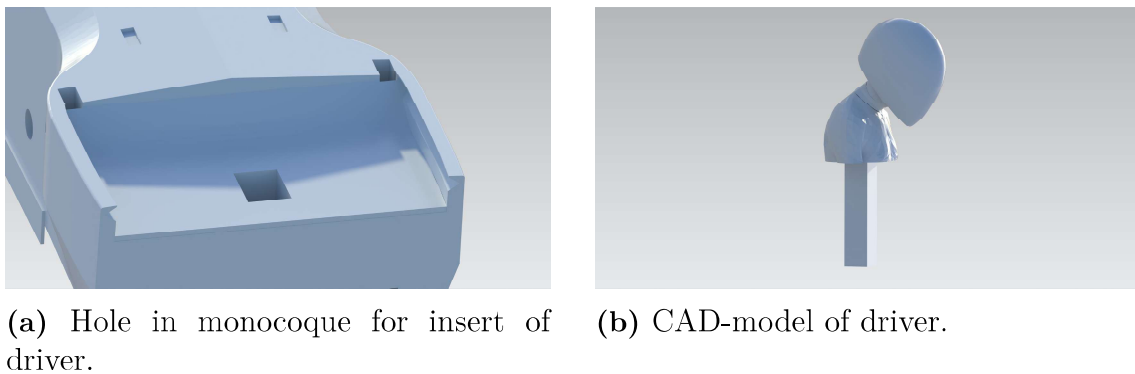


Figure 3.11: CAD model of the hole in the monocoque for the driver and also the driver model.

3.1.3 Preparations for 3D-printing

In order to be able to 3D-print the components for the car. The components had to be split into several smaller subparts. This subsection covers the splitting of those parts. This section also features a section about the scale of the 3D-printed model.

3.1.3.1 Scale

When deciding on the scale of the car, there were three major limitations to adhere to. The first one being the size of the Chalmers wind tunnel, the second being the size at which the available 3D-printers could print and the third being the force limitation of the wind tunnel balance at a maximum of 300 N.

The test section of the wind tunnel at Chalmers has a cross sectional area of $1.6m^2$ and a length of $3m$ [34]. The largest 3D-printer available for use in this project has the following dimensions: $600 * 600 * 660mm$. With these two limitations, and blockage effects considered, a final scale of 1:5 was appropriate. To ensure geometric similarity, this scale factor was applied to every measurement and component of the car. The scale was calculated with the assumption that a blockage ratio Λ of 15% would be a good balance between blockage effects and the ease of using a large model. The following blockage calculations were done:

$$A_M = S \cdot \Lambda$$

Where:

- Frontal area of the model A_M
- Cross sectional area of the wind tunnel test section $S = 1.6m^2$
- Blockage ratio $\Lambda = 0.15$

This gave a frontal area of $A_M = 0.24m^2$ which with the following calculation gave a scaling factor of 0.22 which was rounded to 0.2 for a scale of 1:5.

$$A_M = \lambda \cdot A_C \Rightarrow \lambda = \frac{A_M}{A_C}$$

Where:

- Frontal area of the full scale car A_C
- Scale factor λ

To see whether a 1:5 scale would meet the force requirement, its mass was estimated by extracting the total volume of the full scale car in Siemens NX and then multiplying it with the density of PLA, which is the plastic used to 3D-print the model. It was also multiplied with the scaling factor to the power of 3 to account for the smaller scale. The infill percentage was also included, see equation (3.1).

$$m = \rho_{PLA} \cdot V \cdot \lambda^3 \cdot infill = 1240 \cdot 0.5832 \cdot (0.2)^3 \cdot 0.15 = 0.867kg \quad (3.1)$$

Where λ is the scale factor, ρ_{PLA} is the density of the plastic used and V is the volume of the full scale car. Note that this is an approximation of the final weight. The weight will increase some due to the walls having much higher infill than the rest of the car. The model will also feature bolts, inserts and axles which will significantly increase the weight. If these additional parts are assumed to add 150% to the weight the final weight becomes:

$$0.867 \cdot 2.5 = 2.17kg$$

This is a very rough estimation and probably overestimated. The overestimation is a conscious choice in order to have a built in safety factor.

The force exerted on the wind tunnel balance could then be calculated to be $F_g = mg = 2.17 \cdot 9.81 = 21.31$ N. Although this estimate excludes the effect of downforce on the downward force subjected to the balance, the margin of error is substantial (300 N is the limit of the balance) and therefore the chosen scale works. This also means that every CAD-part and assembly was scaled by a factor of 0.2 before proceeding to 3D-printing or CFD.

After assembly, the car was weight was confirmed to be $2.01kg$ which is within the requirements.

3.1.3.2 Components Below The Rear Wing

The simplified, but still complex, geometry below the rear wing, see figure 3.12, was separated from the monocoque and printed separately to make the printing easier. See figure 3.13 for a view of the separate printed piece.

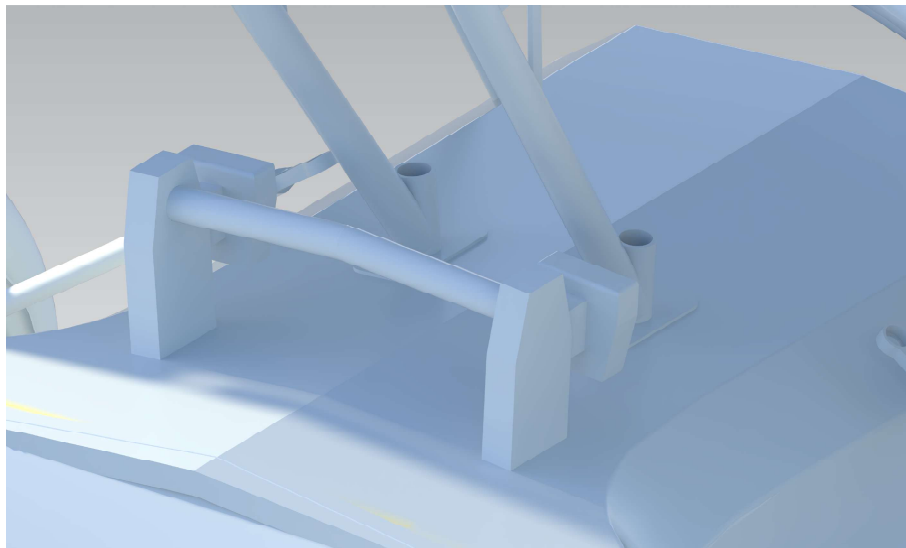


Figure 3.12: The components below the rear wing.

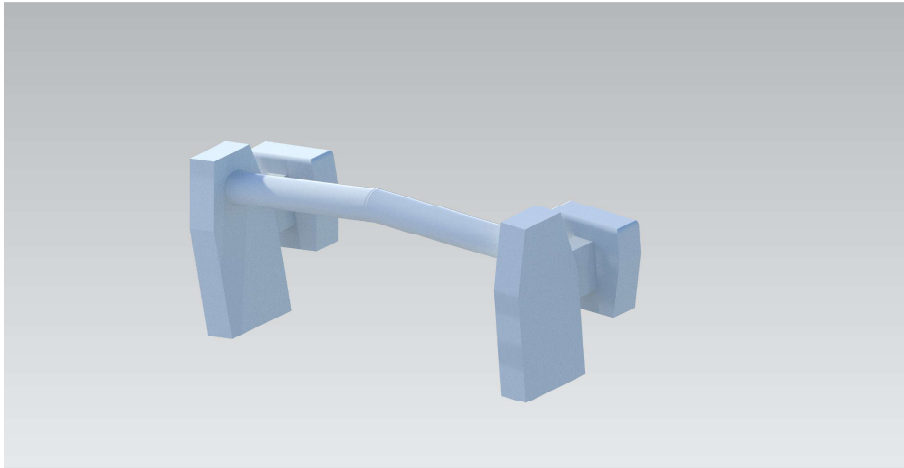


Figure 3.13: The components below the rear wing as they will be 3D-printed.

3.1.3.3 Sidepods

If the monocoque were to be printed along with the sidepods as one piece, it would be too big for the available 3D-printers and the scale would have to be reduced. Less than ideal printing orientations would also have to be used. Because of this, the sidepods were separated from the monocoque and printed separately. To be able to attach them to the monocoque at a later stage, a square extrusion was created on each sidepod, with a corresponding hole in the monocoque. This ensures that the placement of the sidepods gets exactly right during assembly. See figure 3.14 for a model showing the separated sidepod.

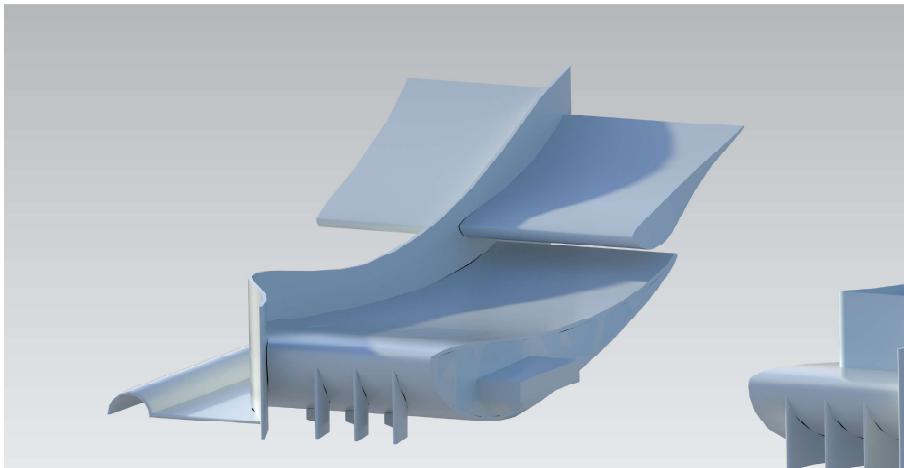
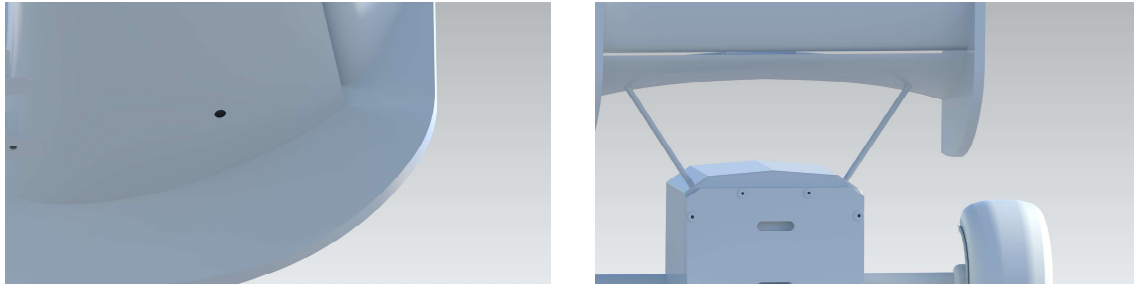


Figure 3.14: CAD model of the sidepods showing the extrusion for a later connection to monocoque.

3.1.3.4 Rear Wing Assembly

The rear wing assembly contains the main hoop, the wing itself and the supporting rods at the back of the wing. Most of this assembly was printed as one piece. The two supporting rods at the back of the wing however, were made from metal and

then attached to the rear wing by being inserted into holes in the wing and the monocoque, see figure 3.15. Corresponding holes were made in the monocoque to accommodate the rods. The rods were only press-fitted into the monocoque because the downforce of the rear wing would press them into place in their holes and thus the need for a more secure attachment was not necessary.



(a) Holes under wing for attachment of rods. (b) The two supporting rods connected.

Figure 3.15: CAD model of the hole for the rods and also showing the placement of the two rods when added to the car.

3.1.3.5 Front Wing

The front wing assembly (front wing and nosecone) was too big to print as one piece. Doing so would also result in non-optimal print orientation regarding the quality of the aerodynamic surfaces. Because of this, the wing and nose cone were split in the middle and printed as two separate parts. With the help of two squared extrusions on one piece and corresponding holes in the other piece, the two halves can then be connected and secured with glue. Figure 3.16 shows the split rear wing ready for printing.

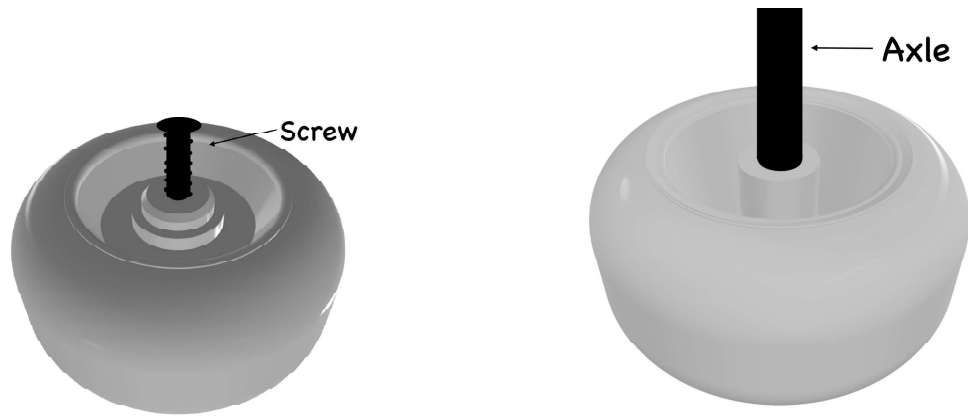


(a) Right half of front wing. (b) Left half of front wing.

Figure 3.16: CAD model of the two front wing halves that later will be connected.

3.1.3.6 Wheels

To make the 3D-printing process easier, the wheels were 3D-printed separately and then attached to the wheels by a screw going through the wheel into the axle, see figure 3.17. The axle will be internally threaded.



(a) Outside view of the wheel showing the screw going in.

(b) Inside view of the wheel showing the axle in its place.

Figure 3.17: A complete view of the assembly of the wheels to the axles.

3.1.3.7 The Monocoque

For the monocoque to be able to withstand aerodynamic loads and also the weight of the components that will be inserted and attached to the body, additional reinforcements were implemented. The monocoque was originally hollow, but the regions around the created holes were reinforced by filling the perimeter of each hole and the surrounding area. This was done to ensure that the holes remained stable and did not deform under load. Furthermore, the wall thickness of the monocoque was increased to improve the overall rigidity of the structure where many components will connect. The cockpit of the monocoque was filled solid to improve strength in the middle. See figure 3.18 for complete overview of the monocoque.

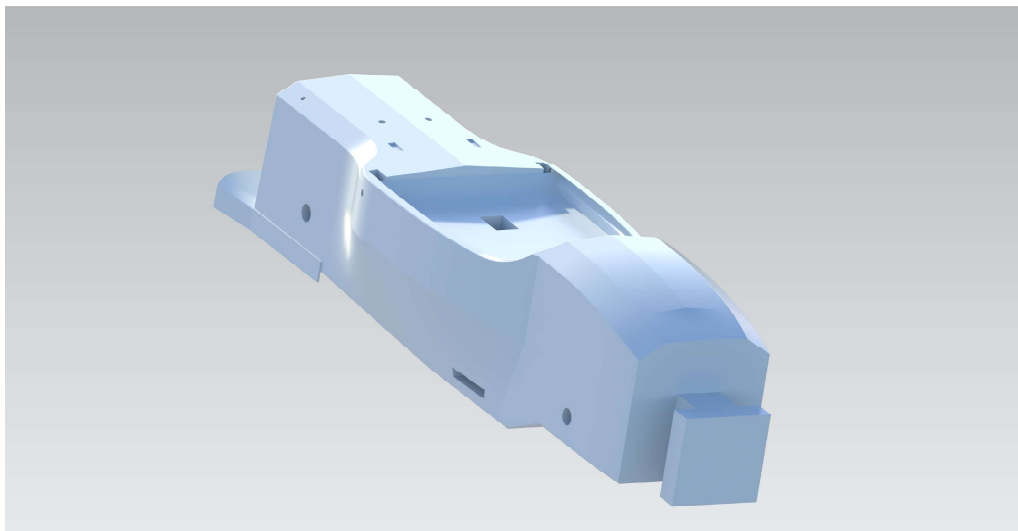


Figure 3.18: A view of the monocoque with all the holes and features.

This approach was chosen instead of fully solidifying the monocoque, as doing so

would significantly increase printing time and material usage. According to recommendations from an expert in 3D-printing from the FUSE lab, reinforcing only critical regions provides sufficient structural integrity while reducing both printing time and plastic consumption by approximately half.

The monocoque also featured holes for mounting various components such as holes for inserts, holes for the axles and a hole for the driver to be inserted in which is shown in section 3.1.2.3 and also figure 3.18.

3.2 CFD

This section will cover the method used for the CFD-part of the thesis. The CFD simulations were performed in Siemens STAR-CCM+ and a starting file was provided from CFS. This file contained a full simulation of the aerodynamics of the 2025 CFS car. It also featured a moving ground, side wind and other features not relevant to this thesis. This meant that the first part of the CFD-method was to modify the provided simulation file to include the 2026 CFS car and to represent the Chalmers wind tunnel.

The physics models used in the CFD-simulations that were not changed are shown in table 3.1. All simulations ran for 5000 iterations and all the extracted values were an average of the last 500 iterations. A numerical verification to see whether the residuals, C_l and C_d had converged was also conducted. The reason as to why the average of the last 500 iterations was used instead of the value of the last iteration was that the values were oscillating with the iterations. This is normal behavior but to avoid using inaccurate values from the peaks and valleys of the oscillation, an average was used.

Table 3.1: Physics Models Used in STAR-CCM+ Simulation.

Category	Model
Space Dimension	Three Dimensional
Time Model	Steady
Material	Gas
Density Model	Constant Density
Flow Solver	Segregated Flow
Flow Type	Turbulent
Turbulence Model	K-Omega (SST Menter)
Governing Equations	Reynolds-Averaged Navier-Stokes
Wall Treatment	All y+ Wall Treatment
Wall Distance	Enabled
Gradients	Enabled
Solution Interpolation	Enabled
Mesh Treatment	Cell Quality Remediation

An initial velocity of $35m/s$ was chosen with regards to three factors. The first

factor is dynamic similarity. To be able to fully compare the results between the simulations, the wind tunnel and extrapolate them to the full scale car, ensuring dynamic similarity is crucial. Since the scales of the car in CFD and the wind tunnel are the same, dynamic similarity between them can be achieved by simply using the same velocity of the air. However, for the scale model to be dynamically similar to the full scale car, the scale also has to be factored in and the velocity can be calculated using Reynolds number. The calculation below shows how the velocity needed for the scale models (wind tunnel and CFD) to be dynamically similar to the full scale car was calculated. In this calculation, subscript M represents the scale model and C the full scale car.

$$Re_{model} = Re_{car} \text{ (Condition for dynamic similarity if geometric similarity is achieved)}$$

$$\Rightarrow \frac{\rho_M L_M}{\mu} = \frac{\rho_C L_C}{\mu} \Rightarrow v_M L_M = v_C L_C$$

Where L is the characteristic length and $L_C = 5 \cdot L_M$

$$\Rightarrow v_M = 5 \cdot v_C$$

According to Gupta and Saxena [35] the average speed of a formula student car on track is $11m/s$. This gives:

$$v_M = 5 \cdot 11 = 55m/s$$

This means that the optimal speed to run CFD and wind tunnel tests of the scale model would be $55m/s$ (it could be argued that the average track speed is not what should be optimized for). However due to the two other factors, it was decided not to use this speed.

The second factor that contributed to the initial value of the velocity is the capabilities of the Chalmers wind tunnel. According to *Chalmers Laboratory of Fluids and Thermal Science* [34] the maximum speed of the tunnel is $60m/s$. this meant that whatever speed was chosen with regard to the other factors would be possible.

The main restricting factor for not running at $55m/s$ was the structural stability of the model. When scaled down and printed, some wing sections of the car became less than one millimeter thick. This thickness combined with the fact that these surfaces produce downforce and a lot of drag means that the force on them is large. To not break the model, or the wind tunnel in the process, the velocity of $35m/s$ was chosen as a compromise between dynamic similarity, quality of the wind tunnel flow and the structural rigidity of the model.

3.2.1 The File Used in The CFD Simulations

The modification done to the provided file from CFS can be reduced to three major categories being car modifications, wind tunnel modifications and mesh modifica-

tions.

The modifications done to the file regarding the car were to remove the previous year CAD-model and replace it with the current car. Two identical simulation files were created at this point, one with the unmodified car from CFS to act as a reference and one with the modified and simplified car. With these simulations done, the two remaining simulations with the driver and the tilted wing were created. These models were scaled down to a 1:5 scale earlier in CAD.

The modifications made to the wind tunnel were to first scale it by a factor of 0.2 like the car. Then, the wind tunnel was lengthened to a total (scaled down) length of 18m with the front of the car a distance of 3.5m from the inlet. The height of the wind tunnel was increased as well and became a total of 2m high. See figure 3.19 for the dimensions of the modified wind tunnel. The wind tunnel was enlarged (especially lengthened) because the non moving ground meant that a boundary layer would form on the ground. This would make the flow around the car and especially the wake more complex and a longer wind tunnel would have a better chance at capturing this behavior. Some tests using a shorter wind tunnel were made, but this made convergence harder to reach, thus the lengthening of the tunnel.

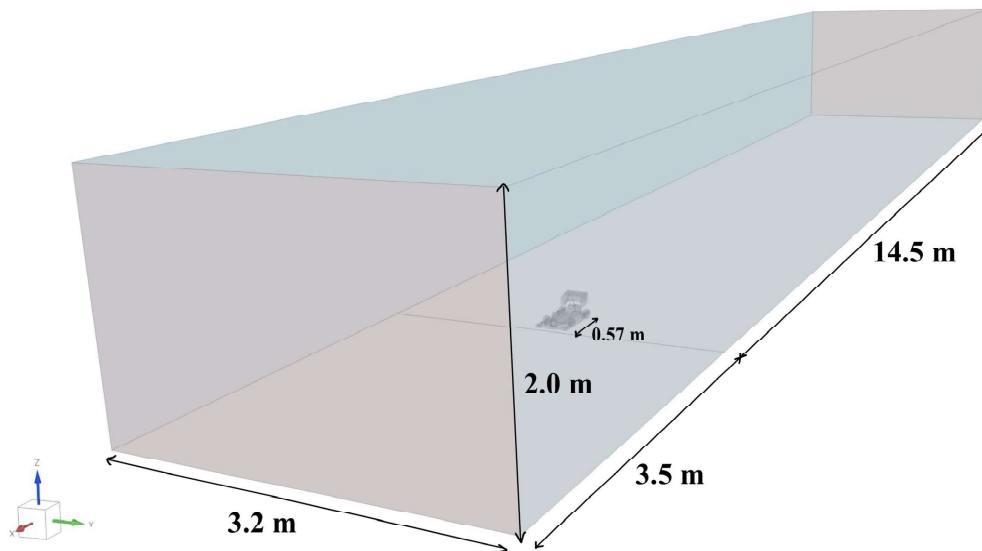


Figure 3.19: Model of the wind tunnel used in the CFD-simulations showing the dimensions of the tunnel and the car.

Since the original bottom surface of the wind tunnel had moving ground, it had to be completely revamped. This meant that the ground surface was split into two sections, see figure 3.20, where the section in front of the car was assigned a slip

boundary and the rear section a no-slip boundary, see figure 3.2 for a summary of the boundary conditions.

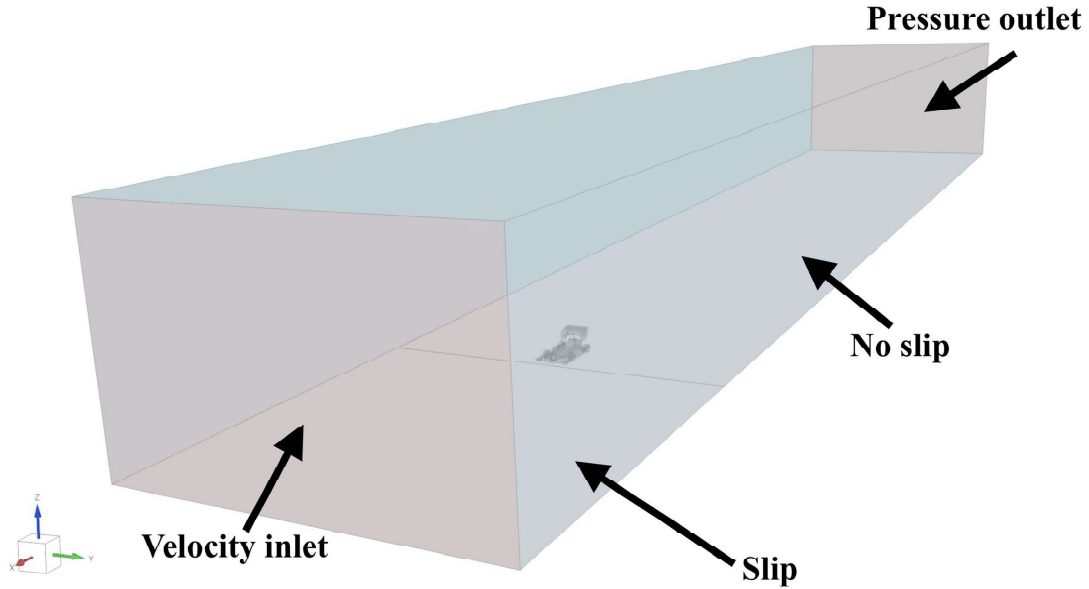


Figure 3.20: Model of the wind tunnel used in the CFD-simulations showing the types of boundaries on the wind tunnel surfaces. The unnamed surfaces are symmetry planes.

Table 3.2: Boundary Conditions Used in STAR-CCM+ Simulation.

Region	Condition	Value
Wind tunnel inlet	Velocity inlet	[35,0,0] m/s
Wind tunnel outlet	Pressure outlet	0 Pa
Wind tunnel sides and ceiling	Symmetry plane	-
Wind tunnel floor in front of car	Stationary slip-wall	-
Wind tunnel floor behind the car	Stationary no-slip-wall	-
Car	Wall	[0,0,0] m/s

The reason for using a slip surface in front of the car is to avoid having a large boundary layer build up in front of the car. A no slip surface in front would, according to section 2.1.3.2, generate a large boundary layer which would build up in front of the car. This would mean that the velocity of the air when it reaches the car would vary greatly between the underside of the car and the top of the car. A slip surface avoids this problem. The reason for using a no slip surface below and behind the car is that this correctly represents the conditions in the wind tunnel where there is a stationary ground.

3.2.1.1 Mesh Convergence

To assess the validity of the mesh used, a mesh convergence study was done. This was done by comparing the results from five different mesh sizes. The original base size of $6.0mm$ was scaled using equation 3.2 which scales the volume according to a percentage of the original, k . In this mesh study, k was chosen to be 0.5 and 1.5 in order to try both a 50% smaller and larger cell volume.

$$\text{new base size} = \text{original base size} \times k^{1/3} \quad (3.2)$$

This yielded two new base sizes of $4.76mm$ and $6.87mm$. Two even coarser meshes with base sizes of $11mm$ and $13mm$ were also chosen for a more rigorous study. In order to evaluate the results from the mesh study of these meshes, the average of the last 500 iterations of the drag- and lift coefficients (C_d and C_l) for all meshes was plotted against the cell count. This allowed for a comparison of how the coefficients relate to the size of the mesh. With these values, an assessment on whether the solution was mesh independent or not was made.

3.2.1.2 Mesh

Apart from the car and wind tunnel modifications. The mesh was adjusted as well. The mesh study yielded a base size of $6mm$ which matches CFS base size (but scaled). In addition to this, the volumetric refinements to the mesh around the wings, wake, and other regions were scaled by the same factor, enlarged a bit and moved to positions which work with the 2026 car, see figure 3.21. The volumetric refinements were made bigger as initial simulations showed that the resolution around the car was too low. Prism layers were also enabled around the no slip surface of the wind tunnel ground. To further refine the mesh, a curve control refinement was introduced to better follow the complex curves of the car.

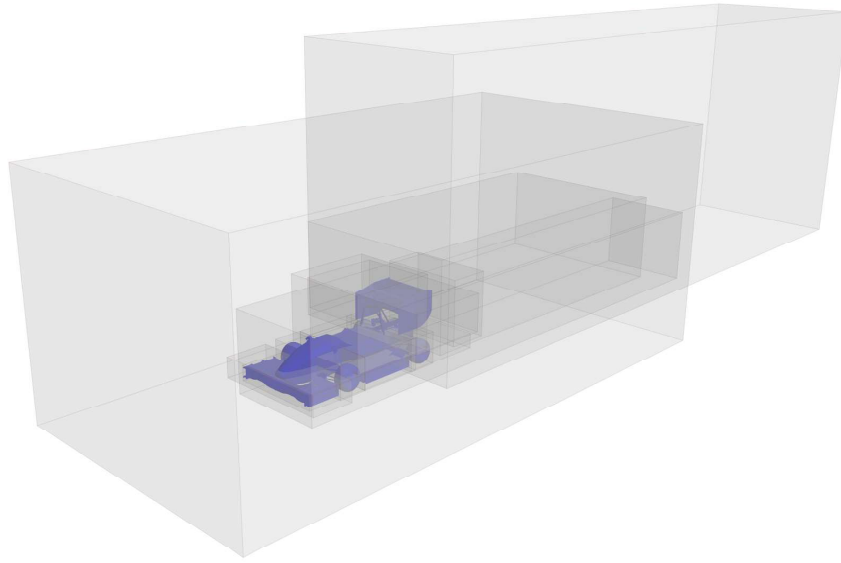
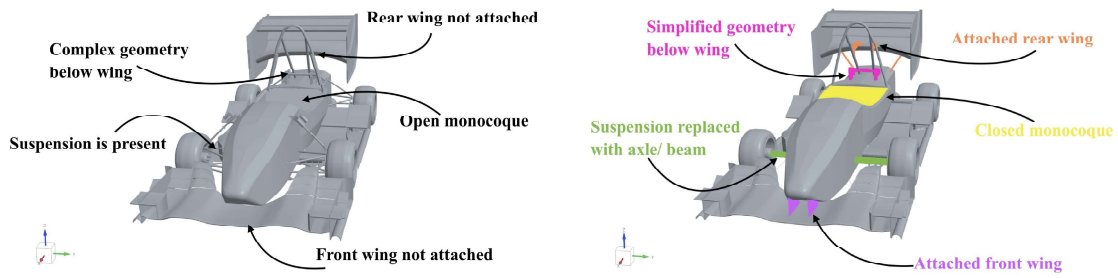


Figure 3.21: A visual representation of the various volumetric refinements used in the simulations. The car is displayed in blue.

Further, the automated mesh used *surface remesher*, *Automatic surface repair*, *Trimmed cell mesher* and *Prism layer mesher* as the main meshers. In addition to this, the meshing workflow included a surface wrapper which made the car "watertight". After the surface wrapper operation was done, a subtract operation was used to subtract the wrapped car from the wind tunnel domain so that the surrounding airflow could be simulated.

3.2.2 Simulations

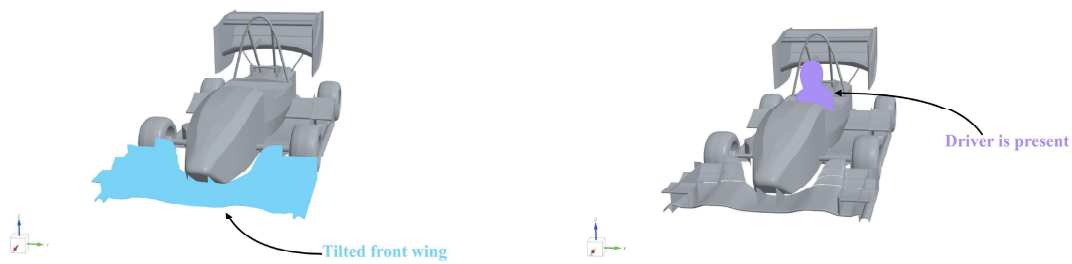
This subsection will cover the four alternative simulations that were a part of this thesis. Those are the reference which were used in combination with the simulation of the simplified car to see whether the simplified car was accurate in predicting the flow of the real car. The other two simulations were done on the simplified car but one had a driver present and the other featured the tilted front wing. These two simulations were done to get a better understanding of the aerodynamics of the car to be able to suggest improvements. Note that all simulations were done with the scaled down car and with the modified file described in section 3.2.1. The cars used in the simulations are shown in figures 3.22 and 3.23.



(a) The car used in the reference simulation. The annotations shows the characterizing features.

(b) The car used in the simulation of the simplified car. The annotations show changes compared to the reference car.

Figure 3.22: Views of the cars used in the reference simulation and the simulation of the simplified car.



(a) The car used in the simulation with the tilted wing. Changes compared to the simplified car are annotated.

(b) The car used in the simulation with the driver. Changes compared to the simplified car are annotated.

Figure 3.23: Views of the cars used in the the reference simulation and the simulation of the simplified car.

3.3 3D-Printing and Assembly of The Model

This section of the thesis presents method regarding the 3D-printing of the components of the car, as well as the assembly of those parts into a complete model.

3.3.1 Workflow

The workflow going from CAD files to the completed model consisted of the following:

First, the CAD files (.prt) from Siemens NX of the separate components were exported as STL files. They were then imported into the slicing software *Prusaslicer*, and were sliced and then printed at various 3D-printers at eXPerimentverkstaden and FUSE lab at Chalmers university of technology. The prints were then finished

by removing supports and smoothing some surfaces with sand paper. After this, the components were assembled into a complete model, see figure 3.24.

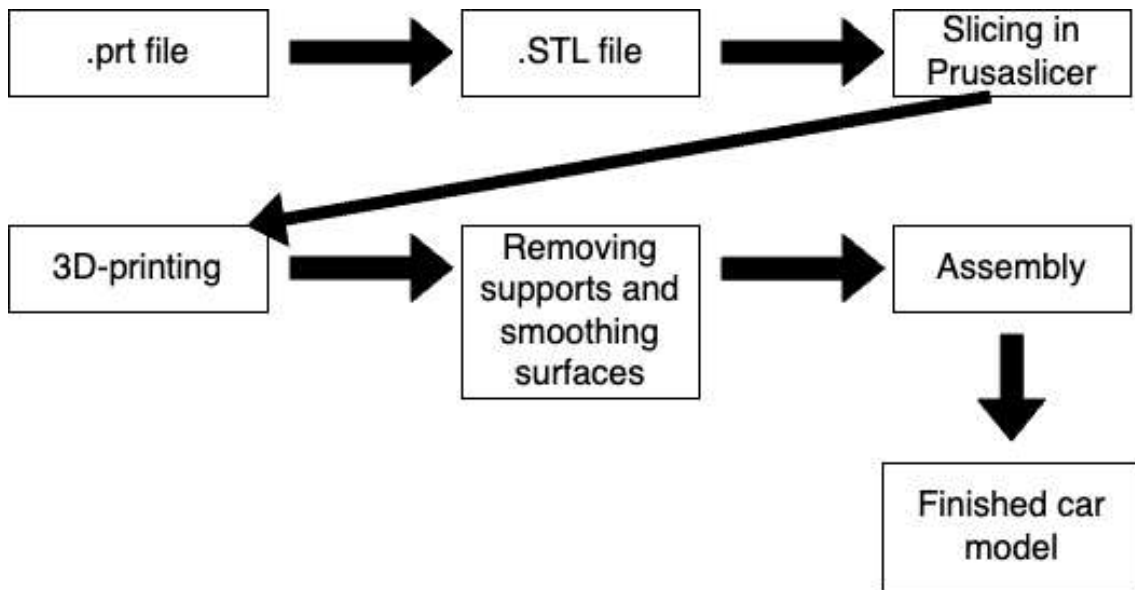


Figure 3.24: Visualisation of the workflow regarding the 3D-printing.

3.3.2 3D-Printing Setup

The 3D-printing setup used varied from component to component. This is because some parts needed higher quality, while others were not as crucial. There were also different 3D-printers available when the printing was done, which meant that different 3D-printers were used for different parts and thus the settings varied. However, the material was chosen to be PLA for every part and the infill to be 15%. PLA was chosen as a material due to its availability at eXPerimentverkstaden at Chalmers. Furthermore, the infill pattern was chosen to be grids for all prints.

3.3.3 Assembly

As mentioned in earlier parts of the thesis, not all parts of the model were 3D-printed. These parts are presented in table 3.3.

With all the parts bought or 3D-printed, the model could be assembled. The assembly consisted of attaching the parts by gluing, using bolts and sliding the front wing in. The rear wing assembly was attached with bolts and inserts while components below the rear wing were glued to the monocoque. The two separate parts of the front wing were also glued together, and the sidepods were glued to the monocoque. See figure 3.25 for a view of the half-completed car and figure 3.26 for a view of the completed car.

Table 3.3: Non 3D-printed parts of the model.

Part	Dimension	Amount
Axle/ beam	$\varnothing 10 \times 277$ mm, <i>M5</i> internal thread	2
Bolts (Wheels)	M5x20mm	4
Bolts (Front mount of rear wing)	M3x8mm	2
Bolts (Rear mount of rear wing)	M3x16mm	2
Threaded insert (Front mount of rear wing)	M3	2
Threaded insert (Rear mount of rear wing)	M3	2
Rods supporting rear wing	$\varnothing 3 \times 6$ mm	2

**Figure 3.25:** A view of the half-completed 3D-printed car. In this state, the car is missing the sidepods, both wings and the optional driver.



Figure 3.26: A view of the completed 3D-printed car. In this figure, the car is displayed with the original front wing and without the driver.

After assembly, the front wing were not as solid as expected. The smallest wing sections deformed substantially with low force. To solve this, the front wing supports discussed in section 3.1.2.1 were installed as shown in figure 3.27.

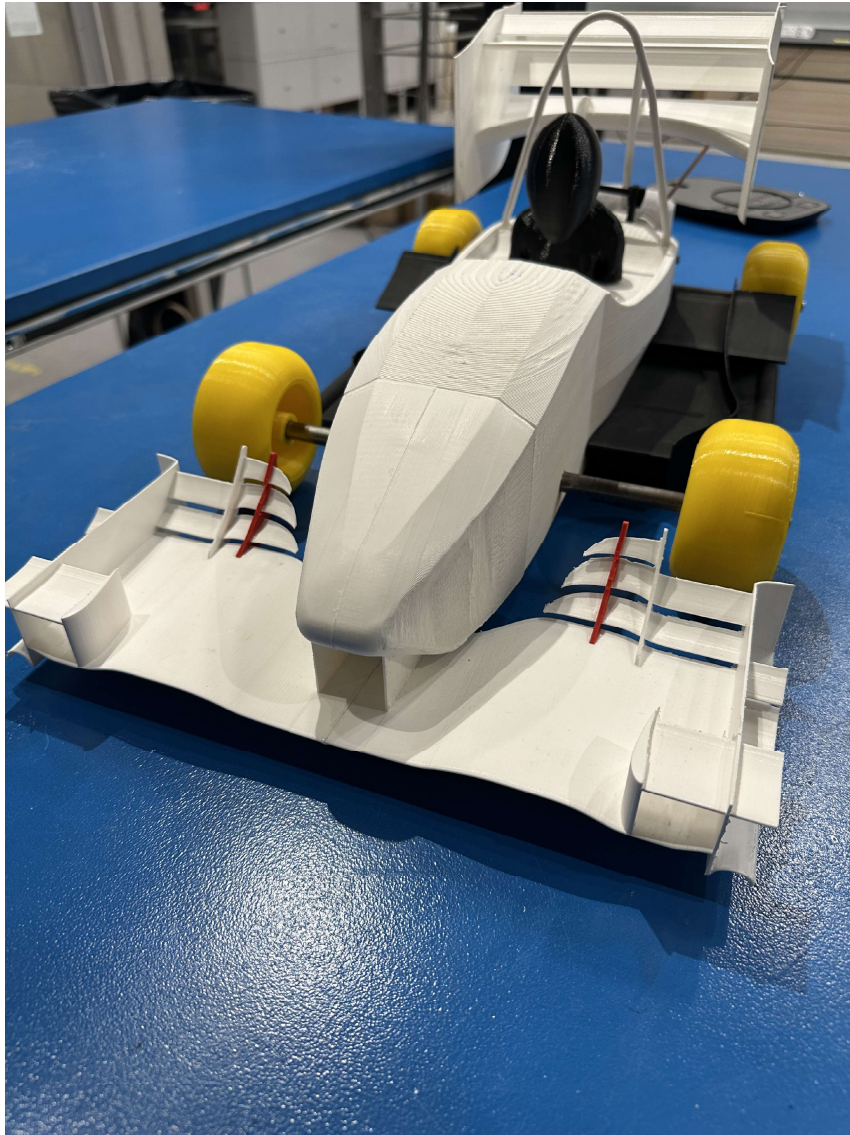


Figure 3.27: A picture of the 3D-printed car model showing the front wing supports in red.

3.4 Wind tunnel

The experimental work in this study was carried out using the Chalmers aerodynamic wind tunnel. As stated by [34], the tunnel is a closed-circuit, low-speed wind tunnel with a test section measuring approximately $3.0 \times 1.8 \times 1.25$ m and a maximum velocity of up to 60 m/s. It is designed to provide low turbulence flow conditions, typically below 0.1%, which makes it suitable for aerodynamic measurements.

3.4.1 Pre-Testing Setup

Before the wind tunnel testing, a dedicated mounting setup was prepared to ensure stable and repeatable measurements. A custom platform provided by CFS was installed in the test section. This platform was designed to fit the wind tunnel and included mounting points aligned with a jig located beneath it with long bolts for securing the scale model by putting them into the wheels of the car, where holes had been specifically designed for this purpose. This ensured that the model remained fixed during testing. See figure 3.28 for an overview of the circular platform and the jig beneath.

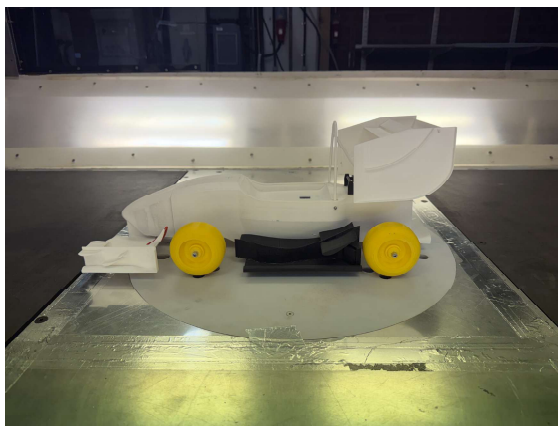


Figure 3.28: Custom mounting platform and jig used to secure the scale model in the wind tunnel.

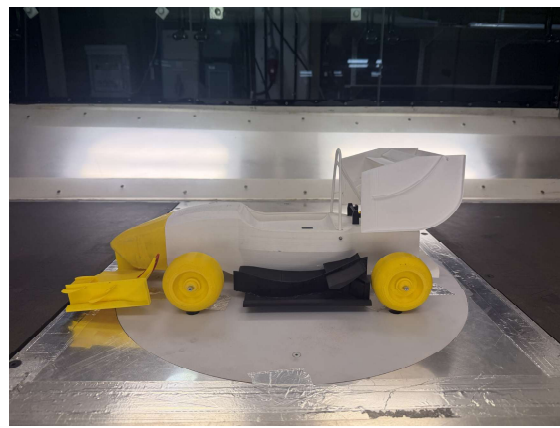
To minimize measurement errors, particular attention was given to sealing the setup. All gaps and openings that could allow air to leak out of the test section were blocked, as such leakage could disturb the flow and affect the results. Additionally, the positioning of the model within the wind tunnel was carefully adjusted to ensure correct alignment and height relative to the incoming flow. See figure 3.29 for a view of the complete setup.

3.4.2 Wind Tunnel Testing and Measurements

During the wind tunnel testing, three different configurations of the scale model were evaluated. The first configuration consisted of the simplified scale model without a driver. The second configuration included the same simplified model with a driver installed. In the third configuration, the front wing was replaced with an alternative design featuring a different angle, while the driver was removed. These configurations were chosen to investigate the influence of individual components on the aerodynamic behavior of the vehicle. See figure 3.29 for a view of the different configurations.



(a) First configuration (simplified model).



(b) Second configuration (With tilted wing).



(c) Third configuration (With driver).

Figure 3.29: Different configurations of the scale model tested in the wind tunnel.

The testing procedure began with the first configuration. A Reynolds sweep was performed by gradually increasing the wind speed from 5 m/s to 35 m/s in increments of 5 m/s. At each velocity step, the aerodynamic forces were recorded and saved for further analysis. From these force measurements, C_D and C_L were calculated. The results were also corrected for blockage effects in order to improve the accuracy of the measurement. This allowed for the evaluation of how the aerody-

dynamic coefficients varied with flow conditions.

After completing the sweep, the second and third configurations were tested at a constant wind speed of 35 m/s to allow for direct comparison between the different setups. The wind speed was then reduced to approximately 15 m/s to perform qualitative flow visualization. During this stage, a rod with an attached string was introduced into the test section to observe airflow patterns and identify vortex structures around the model.

4

Results

This chapter will include the results of the thesis. This includes data from CAD, CFD and from the wind tunnel. A comparison between the CFD and wind tunnel data is also present.

4.1 CAD

This section will cover the results from the CAD-part of the thesis.

4.1.1 The Complete Simplified Car

All the changes made to the components of the car described in 3.1 resulted in a complete modified assembly of the car. To illustrate this result, a view of the modified car is shown in figure 4.1.

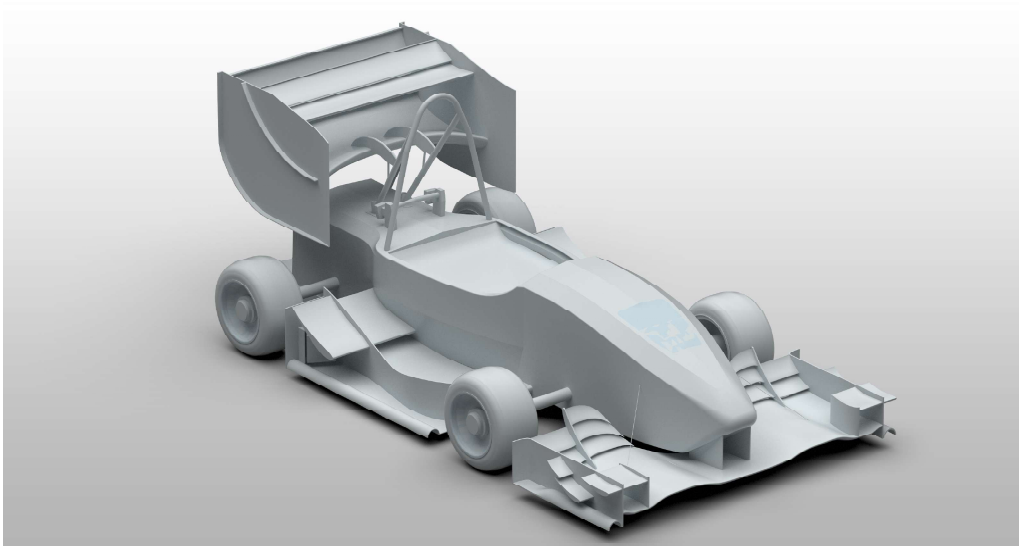


Figure 4.1: A final assembly of the complete simplified car.

4.2 CFD

This subsection covers the results of the CFD simulations done in Siemens STAR-CCM+. All simulations were done using the resulting mesh from the mesh study.

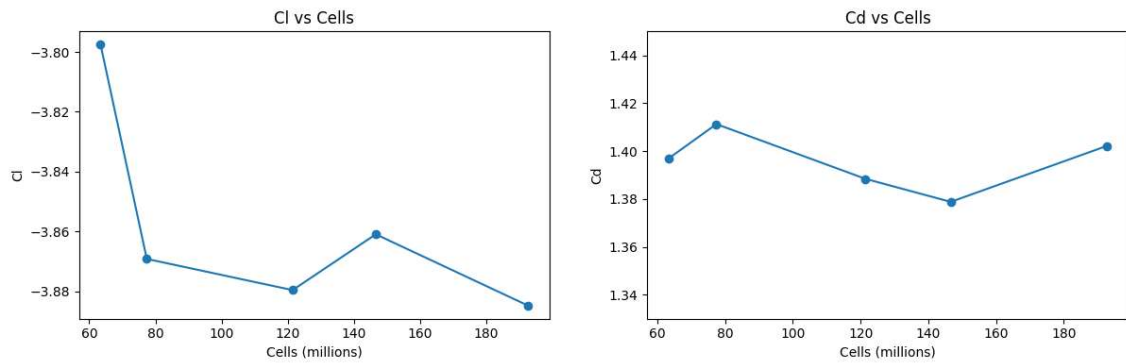
4.2.1 Mesh Study

The reference car simulation was executed with varying degrees of cell counts in the mesh. These results were the basis of the mesh study. The results are summarized in table 4.1.

Table 4.1: Mesh study results.

Base size [mm]	No. of cells	Avg. C_d (last 500 it.)	Avg. C_l (last 500 it.)
13	63,418,820	1.397	-3.797
11	77,408,258	1.411	-3.869
6.87	121,474,621	1.388	-3.880
6.00	146,634,577	1.379	-3.861
4.76	192,700,184	1.402	-3.885

To visualize these results, the lift and drag coefficients were plotted against the cell count of each mesh. This can be seen in figure 4.2. Note that these values are the average of the last 500 iterations.



(a) Lift coefficient C_l vs mesh size

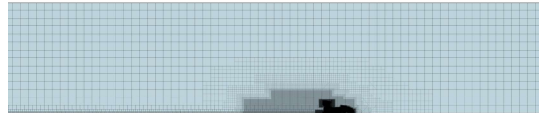
(b) Drag coefficient C_d vs mesh size

Figure 4.2: Mesh study showing variation of lift and drag coefficients with mesh resolution.

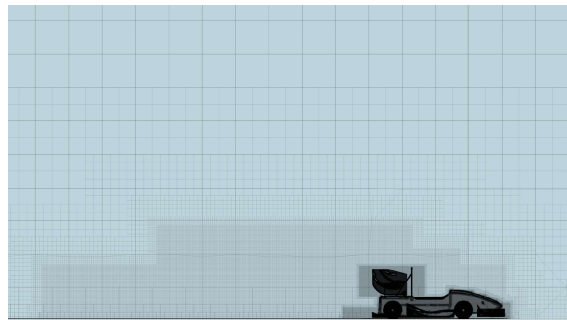
The plots show that both the lift and drag coefficients vary only slightly with increasing cell count in the mesh after the initial coarsest mesh with around 60 million cells. While the results do not show strictly monotonic convergence of the results, the overall variation between meshes remains small after the initial one. This suggests that the solution is approaching mesh independence, but it may not be fully there yet. The C_l plot in figure 4.2 does show signs that mesh independence is reached at around a cell count of 80 million (base size 11 mm). However, since C_d does not show the same stabilizing behavior, a finer mesh was deemed appropriate. However, it should be noted that while C_d does not clearly show convergence, the differences between values are very small. This could mean that C_d was mesh independent even before 60 million cells.

The 6.00mm base size was therefore chosen as a suitable compromise between computational cost and solution accuracy. Compared to the coarser mesh, it should

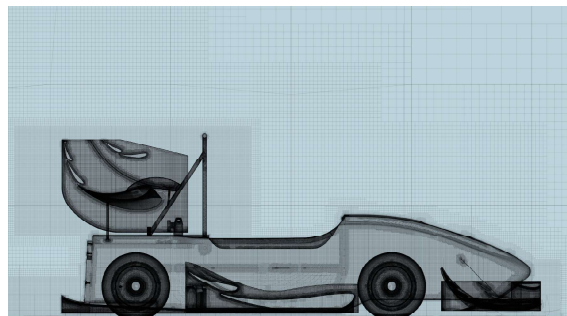
provide improved resolution while having lower computational cost than the finer mesh which does not yield a consistently improved solution. This base size gave the mesh displayed in figure 4.3.



(a) Zoomed out view of the mesh. Note that not the whole wind tunnel is present in the image.



(b) A medium-zoom view of the mesh.



(c) A zoomed in view of the mesh around the car.

Figure 4.3: The mesh used in the STAR-CCM+ simulations at different zoom-levels.

4.2.2 Forces and Coefficients From CFD

The forces and coefficients obtained from the simulations in CFD is presented in table 4.2. These values are an average of the last 500 iterations.

Table 4.2: Forces and coefficients for the different cars in CFD.

Configuration	C_l	C_d	Downforce [N]	Drag Force [N]
Reference	-3.861	1.379	118.2	42.2
Simplified	-3.913	1.396	116.6	41.6
With tilted wing	-3.510	1.389	105.5	41.6
With driver	-3.835	1.374	116.0	41.6

When looking at the results, one can see that all configurations generate similar drag while downforce generation varies more. This is especially true for the car with the tilted wing. Comparing its values with the ones from the car with the original wing, C_d decreased with 0.50% while C_l decreased with 10.30%. This indicates that the tilted wing has a strong negative effect on the generation of downforce.

4.2.3 CFD-Scenes

This section will cover the most important results that could be obtained from the scenes made in CFD. Focus will be on presenting differences and important features rather than a complete representation of the results from all configurations.

The figure below (4.4) shows the pressure coefficient distribution over the simplified car, and the car with the driver.

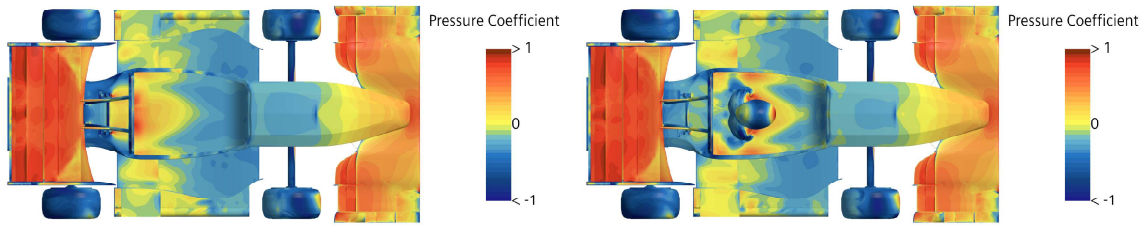


(a) Front view of the pressure coefficient distribution for the simplified car. (b) Front view of the pressure coefficient distribution for the car with the driver.

Figure 4.4: Front views of the pressure coefficient distribution of the simplified car and the car with a driver.

The figures show that the driver causes additional high pressure regions at the upstream part of the car, likely introducing additional drag. The reference and tilted wing scene had no visible differences from the scene with the simplified car.

Figure 4.5 below shows the pressure coefficient distribution from a top view. Again, only the simplified car and the car with a driver are shown since the reference and tilted wing configuration did not differ from the simplified car.

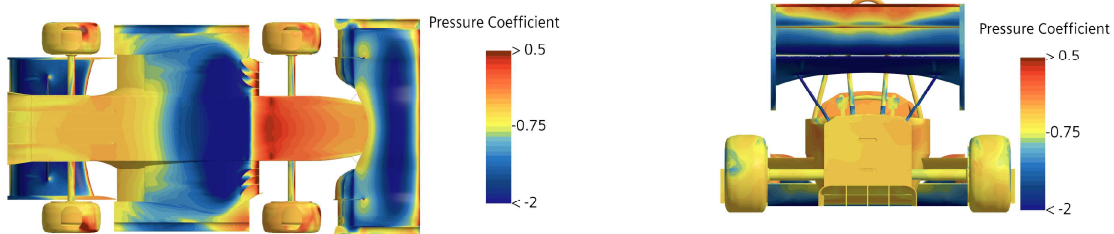


(a) Top view of the pressure coefficient distribution for the simplified car. (b) Top view of the pressure coefficient distribution for the car with the driver.

Figure 4.5: Top views of the pressure coefficient distribution of the simplified car and the car with the driver.

By comparing the car pressure coefficient distribution with and without a driver, one can see that the driver removes the high pressure area behind the monocoque void and moves it in front of the driver.

Figure 4.6 below shows the pressure coefficient distribution below and behind the simplified car. The other configurations are not displayed since they had no visual difference compared to the simplified car.

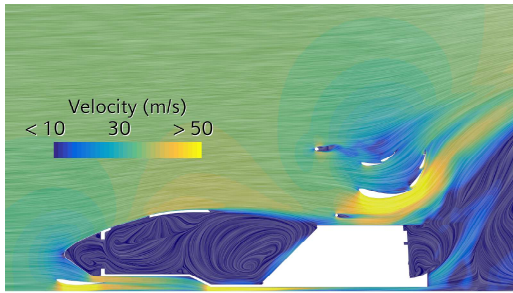


(a) Bottom view of the pressure coefficient distribution for the simplified car. (b) Rear view of the pressure coefficient distribution for the simplified car.

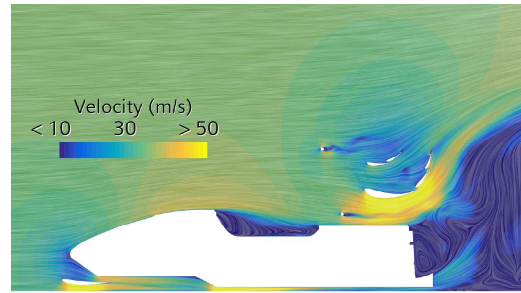
Figure 4.6: Bottom and rear view of the pressure coefficient distribution of the simplified car. Note that the scale is different compared to figure 4.5 to highlight the differences.

The blue color on the floor and behind the car indicates low pressure. This is to be expected and contributes to pressure drag on the rear and downforce generation below the floor.

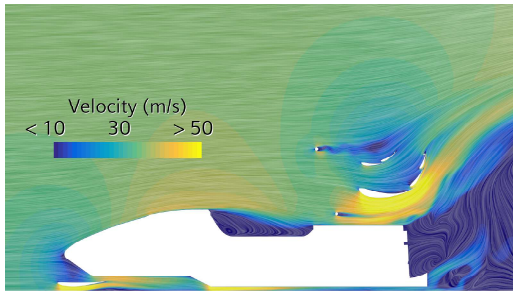
Figure 4.7 below shows the velocity distribution around each configuration of the car from a cut side-view.



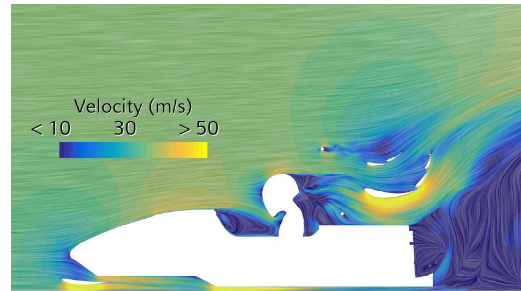
(a) Velocity scene of the reference car. A non-closed monocoque causes air to flow inside the car.



(b) Velocity scene of the simplified car.



(c) Velocity scene of the car with the tilted front wing.



(d) Velocity scene of the car with the driver

Figure 4.7: Velocity scenes of the four configurations of the car.

When comparing the four configurations, it is clear that the flow is very similar. However, subtle differences can be observed. For example, the tilted front wing generates a low velocity (high pressure) area just behind it on the ground. This likely reduces downforce. The driver also generates a large wake behind it which interferes with the rear wing, creating a smaller high velocity area below it. Also, the reference car with the open body, interferes with the flow to the rear wing, creating a smaller high velocity area compared to the simplified car.

Figure 4.8 below shows a streamline scene. This scene shows the airflow around the side of the car and how a vortex forms at the rear wing.

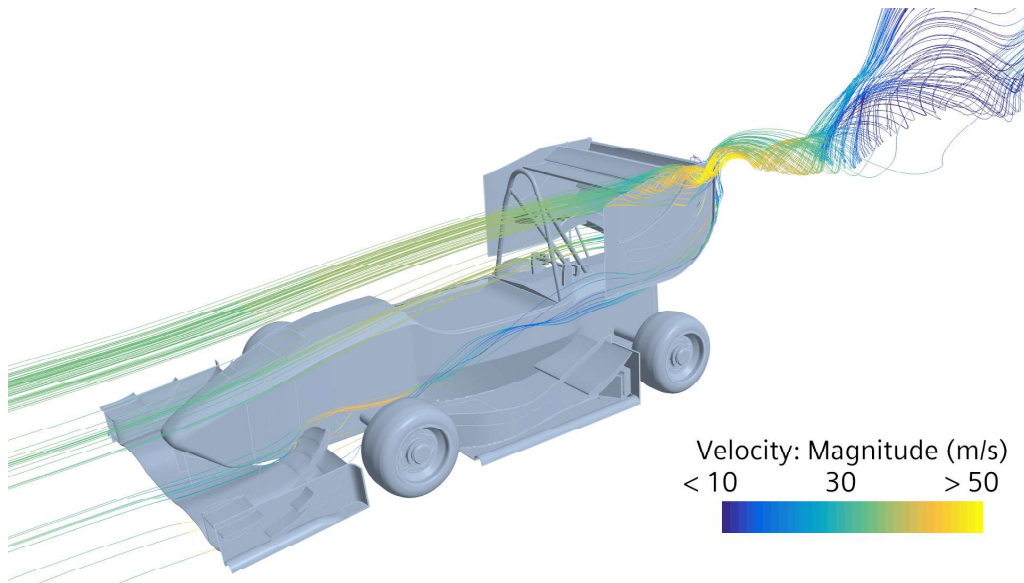


Figure 4.8: A streamline scene of the simplified car showing the path the air takes around the car.

In the figure above a wingtip vortex can be observed. This is due to the pressure gradient caused by the rear wing. This indicates normal function of the wing.

Below in figure 4.9, an iso-surface of low total pressure, indicating regions of high energy loss in the flow is displayed for the car with the tilted wing. The other configurations had no visible difference and are therefore not shown.

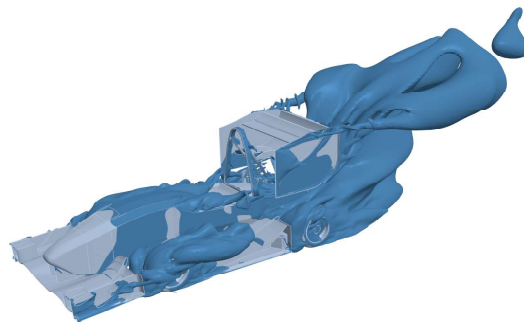


Figure 4.9: An isometric view of the car with the tilted wing showing areas with high-energy losses.

This scene shows that the rear wing is a major contributor to total drag. The front and rear wheels also contribute substantially.

4.2.4 Accumulated C_d and C_l

Plots showing the accumulated force coefficients C_d and C_l for the three configurations of the simplified model as well as the reference car are shown in figures 4.10 and 4.11.

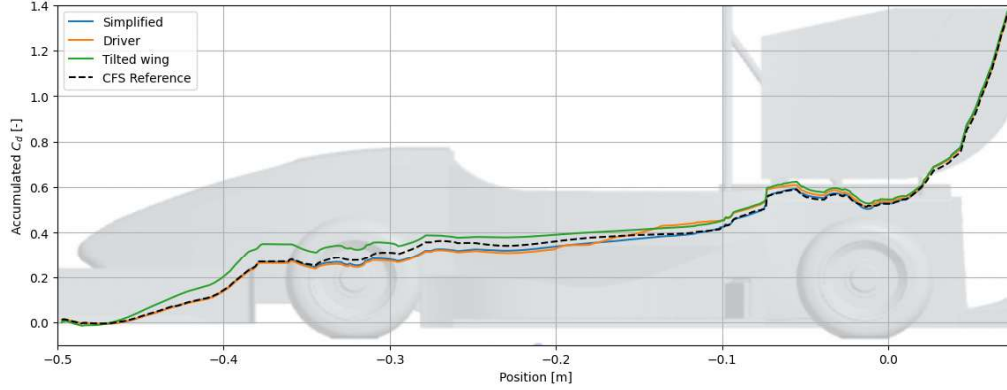


Figure 4.10: A figure showing the accumulated C_d for the different cars.

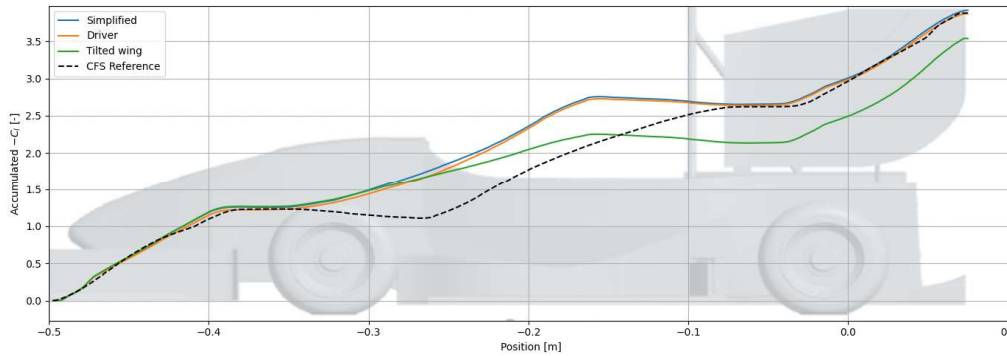


Figure 4.11: A figure showing the accumulated C_l for the different cars.

Based on these plots, it is clear that the changes made to the car, both simplifications and different configurations do not change C_d as much as they change C_l . It is also apparent that the tilted front wing primarily lowers C_l at the sidepods. Also, the tilted front wing increases drag at its location.

Furthermore, the driver increases drag only slightly, as seen by the shift in the orange line where the driver is located. What is also visible is that the reference car differs from the simplified car mainly at the area around the front suspension and the front part of the hole where the driver sits, at least regarding downforce. This is likely caused by turbulence coming from the open body of the car.

4.3 Wind Tunnel Test

The wind tunnel test data will be provided in this section. This data is divided into three parts from the different configurations tested. Those configurations are the

simplified car without driver and with the original wing, the simplified car without driver but with the tilted wing and finally the simplified car with the driver and with the original wing. The wind tunnel data is presented in table 4.3.

Table 4.3: Forces and coefficients of the 3D-printed model (corrected values).

Configuration	C_l	C_d	Downforce [N]	Drag Force [N]
Simplified	-2.224	1.119	66.6	33.5
With tilted wing	-2.335	1.089	71.0	33.1
With driver	-2.120	1.115	67.1	34.0

4.4 Comparison Between Wind Tunnel and CFD

This section presents the differences in aerodynamic coefficients and forces between CFD and wind tunnel tests. In table 4.4, the coefficients are presented while the forces are presented in table 4.5.

Table 4.4: Comparison of aerodynamic coefficients between wind tunnel (WT) and CFD.

Configuration	C_d (WT)	C_d (CFD)	ΔC_d	C_l (WT)	C_l (CFD)	ΔC_L
Simplified	1.119	1.396	0.277	-2.224	-3.913	1.689
With tilted wing	1.089	1.389	0.300	-2.335	-3.510	1.175
With driver	1.115	1.374	0.259	-2.120	-3.835	1.715

Table 4.5: Comparison of aerodynamic forces between wind tunnel (WT) and CFD. All units are in Newtons [N].

Configuration	F_d (WT)	F_d (CFD)	ΔF_d	F_l (WT)	F_l (CFD)	ΔF_l
Simplified	33.5	41.6	8.1	66.6	116.6	50.0
With tilted wing	33.1	41.6	8.5	71.0	105.5	34.5
With driver	34.0	41.6	7.6	67.1	116.0	48.9

5

Discussion

In this chapter, the results of the thesis will be discussed. This includes a discussion regarding the differences between the reference car and the simplified car. It also includes comparisons between CFD and wind tunnel tests. Finally, it also includes an analysis of the flow of the different configurations and suggestions of improvement on the geometry.

5.1 A Comparison of the Simplified Car and the Reference Car in CFD

When comparing the reference simulation with the simulation of the simplified car, it is evident that the two are similar. This can be seen when looking at the drag coefficient and lift coefficient, where the differences were $\Delta C_d = 0.017$ and $\Delta C_l = 0.052$ respectively. The first thing that this shows is that the simplifications made, such as removing the suspension, filling the monocoque void, and attaching the rear and front wing did not affect the aerodynamics substantially since the difference is minimal. This is to be expected and can be attributed to the fact that most of the contributions to drag and downforce come from the wings and sidepods, while the simplifications were mostly made to non-aerodynamic surfaces. Additionally, the changes were mostly minor and small regarding the scale of the rest of the car. This probably contributed to the fact that the differences were small.

When analyzing the accumulated C_l , which is arguably the most significant difference between the simplified car and the reference car, it shows that the simplified car separates from the reference car where the front suspension is located. This is partly due to the suspension of the reference car being removed and replaced with a beam through the wheels. This replacement led to a change in the frontal area, which directly affects C_l , see equation 2.5. Another contributing factor is that the monocoque void is filled in the simplified car but not in the reference, which will affect the flow around the hole in the monocoque, likely introducing turbulence and thus changing the behavior.

5.2 Comparison between Wind Tunnel and CFD

In theory, the wind tunnel test and the CFD simulations should yield the same results. But as shown in section 4.4 this is not the case. The change in C_d between wind tunnel and CFD was around 0.28 while the change in C_l was around 1.50. Both are quite substantial, but the discrepancies in C_l are far more severe. The causes for these differences are difficult to pinpoint exactly. This is due to the complexity of the problem as well as the difference in data between CFD and the wind tunnel, as you get a lot more data to work with in CFD. There are however quite a few things that could explain at least some of the mismatch. Those will be discussed in this section.

5.2.1 Deformations of The Model

In the CFD simulations, the geometry is assumed to be rigid, meaning that the car retains its original and optimal shape under all conditions. In reality, however, aerodynamic loads cause structural deformation. This effect is particularly relevant for the wind tunnel model, which was manufactured using PLA, a material with relatively low stiffness.

During testing, visible deformation of both the front and rear wings was observed. This is reasonable since these areas have a high-pressure coefficient, which can be seen in section 4.2. This means that the forces there are large, which causes the deformation. Such deformation alters the effective geometry of the aerodynamic surfaces, for example, by reducing the angle of attack or introducing twist. As a result, the generated downforce is likely reduced compared to the undeformed geometry used in CFD. This is because a different angle of attack could for instance cause separation of the flow and thus large wakes.

This provides a plausible explanation for part of the discrepancy between CFD and wind tunnel results, where CFD consistently predicts higher downforce. Although it was not observed, one possible explanation for the low downforce in the wind tunnel could be that the front wing deformed enough so that its ends could touch the wind tunnel ground. This would transfer some of the downforce down into the ground of the wind tunnel rather than to the balance.

Furthermore, deformation effects are not directly scalable between the wind tunnel model and the full-scale car. The material properties and structural stiffness differ significantly, meaning that the 3D-printed model is more susceptible to deformation under aerodynamic loading. Consequently, the wind tunnel results may underestimate the aerodynamic performance relative to both CFD predictions and the full-scale vehicle.

5.2.2 Surfaces

Another plausible explanation for the underprediction of downforce in the wind tunnel compared to CFD is the surface quality of the 3D-printed model. The surface finish varied between different parts and mainly the monocoque had a sub-optimal surface quality. Although the print orientation was optimized to ensure high-quality aerodynamic surfaces, the final result of the monocoque was not ideal. This can lead to earlier separation as well as introducing turbulence, thereby reducing aerodynamic efficiency.

In real-world applications, non-optimal surface finishes are common but generally undesirable. In contrast, CFD models assume perfectly smooth, mathematically defined surfaces, meaning that no surface roughness is present. This promotes a higher degree of flow attachment, resulting in more effective wing performance. The velocity fields shown in section 4.2 indicate that the flow remains attached to the wing surfaces and does not separate under the simulated conditions. However, since corresponding flow visualizations are not available from the wind tunnel tests, it is difficult to confirm whether separation occurred on the 3D-printed model. It is nevertheless a possibility, and could therefore explain part of the discrepancy between the results.

In addition, a rougher surface increases skin friction drag across the entire vehicle. This effect is not limited to aerodynamic components such as wings but applies to all exposed surfaces of the car, further contributing to the overall increase in drag observed in the experimental results.

5.2.3 Differences in Setup

Differences in the setup between the CFD simulations and the wind tunnel testing may also explain some of the observed discrepancies. In the CFD model, the car is represented as a single continuous body with perfectly defined geometry, without gaps or assembly tolerances. In contrast, the 3D-printed wind tunnel model consists of multiple assembled components. Each interface between parts introduces manufacturing tolerances, resulting in small gaps and potential relative movement between components.

These gaps may allow local flow recirculation and an increase in turbulence, which can reduce the effectiveness of downstream aerodynamic features. In addition, bolts used to attach certain components were exposed to the airflow in the physical model but were not represented in the CFD model. These may have introduced additional local disturbances in the flow field. Furthermore, the front wing reinforcing structure present in the physical setup was not included in the CFD geometry, which may also have influenced the aerodynamic performance.

Although the effects of these differences are likely to be relatively small compared to the dominant flow features, they should still be considered potential contributors to the discrepancies between the CFD and wind tunnel results.

Another difference between the wind tunnel test and CFD is the incoming air. The settings used to model the turbulence of the incoming air in CFD probably did not match the actual air of the wind tunnel to 100%. Again, the effect is probably small, but still present nonetheless.

Furthermore, in CFD the wheels were slightly lowered into the ground to simulate the slight flattening of the tires when subjected to the weight of the vehicle. The 3D-printed model was instead suspended a few millimeters above the floor of the wind tunnel. Consequently, the effective ride height differed between the two tests. This could affect the underbody flow which is important for generating downforce. It is possible that the higher ride height of the 3D-printed model was enough to greatly reduce the desired aerodynamic effect of the car floor and the front wing.

The CFD velocity fields in section 4.2 show that the airflow beneath the car accelerates. According to Bernoulli's principle, this creates a low-pressure region that generates downforce. This is how the floor generates downforce. It is also visible that the section of the monocoque just behind the front wing has more clearance to the ground and this results in less acceleration of the flow, see figure 5.1. Since this difference in height between the two sections is small and the effect is large, the effect of the slightly higher car in the wind tunnel is probably also big since the low clearance section would become a high clearance section.

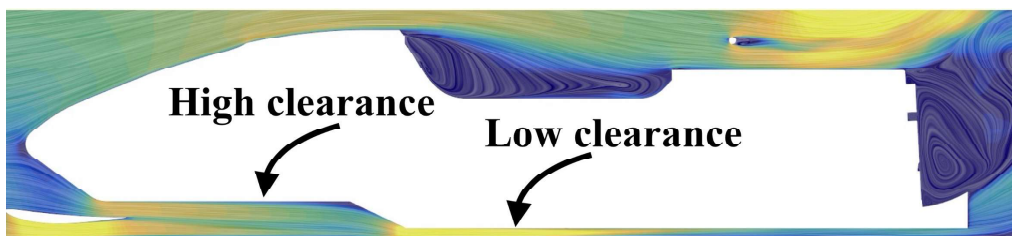


Figure 5.1: A velocity scene showing the acceleration of the flow when the clearance gets smaller.

This effect is also evident in the accumulated lift coefficient distribution (figure 4.11). In the regions with higher ground clearance, the accumulated $-C_l$ curve becomes relatively flat, indicating a low local contribution to downforce generation. Conversely, regions with lower ground clearance exhibit a steeper positive gradient, showing that these sections generate a larger proportion of the total downforce.

Another thing to note is that the simulations in this thesis were steady state and used the RANS equations, section 2.3.2. That means that the simulations had no time variation and no unsteady structures such as vortex shedding, oscillations in flow separation and wake fluctuations. In the wind tunnel however, these effects are present and could influence parameters such as C_d and C_l .

5.2.4 Inherent Errors in CFD

Another factor that may contribute to discrepancies between wind tunnel experiments and CFD results is the inherent numerical and modeling uncertainty associated with CFD. One major source of such inaccuracies is the computational mesh. If the mesh quality is insufficient, important flow features may not be properly resolved. In particular, a coarse mesh around the vehicle or in critical flow regions may fail to capture complex flow structures. Additionally, poor quality cells, such as highly skewed elements, can introduce numerical errors.

In the simulations conducted in this thesis, highly skewed cells were removed using STAR-CCM+, and the mesh quality was evaluated through a mesh independence study, which indicated convergence of the results. This suggests that the global solution is relatively insensitive to further mesh refinement.

However, local flow features may still be affected by insufficient resolution. The total pressure loss contour for the simplified car with the tilted wing (figure 4.9 shows a high-loss region that appears separated from adjacent regions. This may indicate that the mesh in this area is too coarse to accurately resolve the flow structures, leading to artificial discontinuities in the pressure loss field. Alternatively, this behavior may reflect physical phenomena, such as localized vortical structures or separation zones. Additional local mesh refinement would be required to determine whether this feature is numerical or physical in origin.

In addition to mesh-related errors, turbulence modeling plays a significant role in the outcome of CFD simulations. Different turbulence models exhibit varying performance depending on the flow characteristics, particularly in cases involving separation, transition, or strong pressure gradients. The *SST* – *k*– ω model used in this thesis is generally considered suitable for aerodynamic applications and is expected to provide a reasonable representation of the flow. However, its accuracy for the present case cannot be guaranteed.

Since only a single turbulence model was evaluated, no assessment of model sensitivity has been performed. As noted by Piferi [36], different turbulence models can, in some cases, yield significantly different results even under identical conditions. Consequently, the choice of turbulence model may contribute to the discrepancies observed between the CFD results and wind tunnel measurements.

5.3 The Effect of a Driver

When comparing the simplified car with and without the driver, one can see that both the downforce was reduced slightly, with a difference of $\Delta F_{Down} = 0.5$ N from CFD and $\Delta F_{Down} = 1.1$ N from wind tunnel data. Furthermore, the drag stayed the same (to one decimal) in CFD while it went from 33.5 N for the simplified car to 34.0 N for the car with the driver from the wind tunnel.

In other words, the addition of a driver affected drag and lift approximately the same amount. However, it is unclear whether these changes were positive or negative for the performance of the car since CFD and the wind tunnel data do not agree. Why this difference exists is investigated in section 5.2. Because of this, the results from CFD and the wind tunnel will be discussed separately in this section.

5.3.1 The Driver in CFD

In the CFD simulations. The driver caused the downforce and drag to decrease. It did also, judging by the accumulated C_l and C_d plots in section 4.2.4, change the balance of these coefficients along the length of the car. The driver causes C_d to rise slightly at its position which is expected. Why C_d decreased in total is more difficult to explain. At first glance, the introduction of a bluff object in the middle of the flow should increase drag. This occurs because it makes the frontal area higher and generates a wake causing pressure drag. This wake is also visible in the velocity scene of the car with the driver (figure 4.7). One possible explanation is that the driver alters the upstream flow field, reducing the amount of airflow interacting with the rear wing. Since the shape of the rear wing produces more drag than the shape of the driver, this could be the cause. Furthermore, when looking at the side view velocity scene of the car with the driver and the simplified car, it is possible to observe a difference in the size of the wake at the rear of the car. The driver causes the wake at the rear of the car to decrease in size which should yield lower pressure drag.

The flow redirection likely caused by the driver is also likely responsible for the reduction in C_l . One plausible explanation as to why could be that the flow gets redirected to the upper area of the rear wing, the bottom part gets reduced airflow. The balance of the rear wing segments then gets disrupted, and because the wing is designed to work as a complete piece, this imbalance causes lower performance. This redirection of flow to the top of the wing was observed in the wind tunnel where the deformation of the top wing segment was significantly larger with the driver than without.

5.3.2 The Driver in The Wind Tunnel

The data from the wind tunnel are much more scarce than from CFD. however C_l , C_d , and their corresponding forces are available. When looking at these, one can observe that the driver caused drag to increase and downforce to decrease. The fact that downforce decreased when adding the driver is consistent with theory. This is likely due to the same reasons as were discussed with the driver in CFD. This also indicates that both methods match in trend here, though not in magnitude of the coefficients and forces.

The increase in drag associated with the driver is likely explained by its role as a bluff body in the flow field. This results in strong pressure gradients between the upstream and downstream surfaces, together with a separated wake region. These effects increase pressure drag and therefore contribute to the overall rise in drag

observed in the wind tunnel measurements. This trend is more consistent with theoretical expectations than the CFD results, which indicate a reduction in drag with the driver. This discrepancy suggests that certain aspects of the CFD modeling may require further refinement. It could also be explained by the slightly smaller rear wake of the car with the driver than without, see figure 4.7.

5.4 The Effect of a Tilted Front Wing

The tilted front wing was created by modifying the original front wing angle, more specifically 5° downwards, see section 3.1.2.1 and figure 3.5. This configuration was included to evaluate how a change in a front wing affects the aerodynamic behavior, while also verifying that the modularity of the scaled model works with interchangeable parts, see figure 3.6.

5.4.1 The Tilted Front Wing in CFD

When running the tilted wing configuration in CFD, the results displayed in Table 4.2 show a decrease in C_d by 0.50%, while C_l decreased by 10.3% compared to the simplified car.. The decrease in C_l can mainly be explained by how the tilted front wing changed the flow around the front part of the car and further downstream. In the CFD results, the velocity field was similar to the original simplified car, but the tilted wing created a larger low-velocity region between the monocoque and the front wing, see figure 4.7 . This indicates that the flow around the front wing became less effective, which reduced the pressure difference needed to generate downforce.

The accumulated C_l plot (figure 4.11) indicates that the loss of downforce when using the tilted wing mostly originates around the x-position of the sidepods. This could be explained by the front wing redirecting flow away from the sidepods due to its angle. The changed flow over the sidepods could then interfere with their function and thus reduce downforce generation.

The small change in C_d indicates that the tilted wing did not have a major influence on the total drag of the car. Even though the flow field changed locally around the front wing, the main drag-producing regions remained similar to the original simplified car. Most of the drag was still generated around the rear wing and in the wake behind the car, see figure 4.9. Because of this, the change in front wing angle had a much larger effect on the lift coefficient than on the drag coefficient, see figure 4.10 and figure 4.11. Therefore, the tilted front wing mainly affected the downforce generation, while the total drag remained almost unchanged. The change in C_d that did occur was located at the site of the front wing, which was expected.

5.4.2 The Tilted Front Wing in the Wind Tunnel

When comparing this with the wind tunnel results, a different trend can be seen. The wind tunnel results are presented in table 4.3. In the wind tunnel, the tilted wing decreased C_d from 1.119 to 1.089, while C_l changed from -2.224 to -2.335 . This means that the tilted front wing slightly reduced drag and slightly increased the generated downforce in the wind tunnel test.

This does not follow the same trend as the CFD results, where the tilted front wing reduced the magnitude of C_l . The comparison between CFD and wind tunnel results is shown in table 4.4 and table 4.5. For the tilted wing configuration, the difference between the methods was 0.300 for C_d and 1.175 for C_l . The force comparison also shows that CFD predicted 8.5 N higher drag and 34.5 N higher downforce than the wind tunnel test.

The difference between the CFD and wind tunnel results can have several causes. One possible reason is that the physical front wing may not have had the same angle or position as in the CFD model due to assembly tolerances in the modular connection. Small differences in the front wing angle can affect the pressure distribution and the flow reaching the rest of the car. Another possible reason is the surface quality of the 3D-printed model. The CFD model has a smoother surface than the physical model, while the printed model has layer lines and small surface imperfections. This can change the boundary layer behavior and therefore also the aerodynamic forces.

The wind tunnel setup itself may also have influenced the results. The physical model was mounted on a platform and secured with additional support structures, while the CFD setup is an idealized representation of the test case. These differences can affect the flow under and around the car. Because of this, the wind tunnel result should not be interpreted as proving that the tilted wing is better than the original wing. Instead, the results indicate that the tilted front wing affects the aerodynamic behavior, but that the exact effect depends on both the CFD setup and the wind tunnel setup.

Furthermore, the tilted front wing has a much more aggressive angle, which redirects flow much more upwards. This causes the flow to be less optimal at the sidepods and rear wing, and their effectiveness could decrease as a result.

The steep angle of attack of the front wing could also be causing separation. This is further justifiable by looking at figure 4.7 from the CFD simulation, where a slight separation occurs. Separations at the wing surface show that it is beginning to stall, which greatly reduces performance and could be a cause for lower downforce when using the tilted wing. It should also be noted that due to slightly rougher surfaces on the 3D-printed model, a larger separation likely occurred in the wind tunnel compared to CFD, making this point even more important.

6

Conclusion

This chapter will present the conclusion of this thesis. This includes some suggestions of improvements to the car as well as some final remarks regarding the results.

6.1 The Model

The model developed and manufactured during this project was satisfactory regarding the testing conducted in the wind tunnel. It could withstand the forces acting on the components from the wind tunnel. Due to its modularity, it allowed many possibilities of testing different front- and rear wings for the purpose of finding the most optimal design for the car.

There are many ways in which the model could be improved. The surfaces, for example, in a few places were not perfectly smooth and could therefore not exactly represent the conditions that would exist on a full-scale car. For future projects or studies similar to this, it would be beneficial to try different approaches regarding the 3D-printing of the model or alternatively treating the surfaces post printing.

Another area of improvement is the structural stability of the model. Higher in-fill on delicate parts like the rear wing airfoils could reduce the deformations during testing and therefore provide more accurate results.

During this project, as will be discussed in chapter 6.2.1, issues were found regarding the sidepods. However, the model that was developed could not properly test alternative solutions to this issue since the sidepods were not modular parts. Therefore, a recommendation for future projects would be to make the sidepods modular similarly to the front and rear wings from the current model.

6.2 Suggestions of Improvement

The goal when designing a formula student car is to minimize drag and maximize downforce while making sure that airflow around the car is clean. Using the results from this thesis, we will propose some suggestions of improvements to the car.

6.2.1 Improvements to the Sidepods

From the results in the wind tunnel testing, vortices were found on the surfaces of both sidepods. Vortices lead to air being trapped in a circular motion on the surface of the sidepods, slowing the air down and preventing it from flowing onto the rear of the sidepods. This means that less air at lower speeds will hit the wings on the sidepods, generating less downforce, which is not ideal for a performance car.

There are two approaches that could be taken to try and amend this issue. You could either introduce changes to the design of the sidepods to try and make it possible for the air to flow undisturbed on the surface of the sidepods. Another approach would be to modify the design of the front wing with the goal of altering the flow of the air incoming from the front wing to the sidepods. However, you could of course also take both approaches simultaneously to find a design that optimizes both components at the same time. It is also worth noting that in real life the wheels will be spinning which might affect the flow over the sidepods.

6.2.2 Tilted Front Wing

As discussed in chapter 5.4.2 it is very difficult to draw any conclusions regarding the performance of the tilted front wing. From the results in CFD, the component is clearly a worse option. However, the results from the wind tunnel tell a different story. Since neither testing method fully represents real-world conditions, due to factors such as reduced testing velocities and the rough surface finish of the 3D-printed model, the results should be interpreted with caution. Nevertheless, the findings suggest that the new design may provide improved downforce performance and therefore warrants further investigation under more representative conditions.

Bibliography

- [1] *Global Formula Racing*. 2026. URL: <https://www.global-formula-racing.com/en/formula-student/>.
- [2] *Chalmers Formula Student*. 2026. URL: <https://www.chalmersformulastudent.se>.
- [3] *Chalmers Formula Student: Svea*. 2025. URL: <https://www.facebook.com/ChalmersFormulaStudent/photos/d41d8cd9/1375062017962282/>.
- [4] *German-Dutch Wind tunnels*. URL: <https://www.dnw.aero/services/measurement-techniques/flow-field-measurements/>.
- [5] *Resolved Analytics*. URL: https://www.resolvedanalytics.com/cfd/benefits-of-cfd-analysis?utm_source=chatgpt.com.
- [6] Andreas Borg. *Status and Trends in Large Scale CFD Applications for Vehicle Design at Volvo Car Corporation*. Tech. rep.
- [7] Yoshihide Tominaga et al. “Accuracy of CFD simulations in urban aerodynamics and microclimate: Progress and challenges”. In: *Building and Environment* 243 (Sept. 2023). ISSN: 03601323. DOI: 10.1016/j.buildenv.2023.110723.
- [8] Enrico Giacomini and Lars Göran Westerberg. “CFD Analysis of Transition Models for Low-Reynolds Number Aerodynamics”. Doctoral dissertation. Sept. 2025. DOI: 10.3390/app151810299.
- [9] Peter R.N. Childs. “Laws of Motion”. In: *Rotating Flow*. Elsevier, 2011, pp. 17–52. DOI: 10.1016/b978-0-12-382098-3.00002-0.
- [10] “Resolved Analytics”. In: (2020). URL: <https://www.resolvedanalytics.com/fluid-dynamics/conservation-of-momentum-in-fluid-dynamics>.
- [11] Siele Charles, Rotich John, and Adicka Daniel. “Numerical Solution of the Navier-Stokes Equations for Incompressible Fluid Flow by Crank-Nicolson Implicit Scheme”. In: *Applied and Computational Mathematics* 10.1 (2021), p. 10. ISSN: 2328-5605. DOI: 10.11648/j.acm.20211001.12.
- [12] Manuel Saldana et al. “The Reynolds Number: A Journey from Its Origin to Modern Applications”. In: *Fluids* 9.12 (Dec. 2024). ISSN: 23115521. DOI: 10.3390/fluids9120299.
- [13] Tom Benson. *NASA - The Drag Coefficient*. URL: <https://www.grc.nasa.gov/www/k-12/VirtualAero/BottleRocket/airplane/dragco.html>.
- [14] Tom Benson. *NASA - The Lift Coefficient*. URL: <https://www.grc.nasa.gov/www/k-12/VirtualAero/BottleRocket/airplane/liftco.html>.
- [15] Shivam R Gupta. “Pressure coefficient (C_p) equation for a compressible flow”. In: (2023). ISSN: 2349-9249. URL: www.tijer.org.
- [16] David Weyburne. *Aspects of Boundary Layer Theory*. Tech. rep. Jan. 2025.
- [17] David Weyburne. *Boundary layer*. 2016.

- [18] J. Gordon. Leishman. *Introduction to aerospace flight vehicles*. Embry-Riddle Aeronautical University, 2023. ISBN: 9798985261400.
- [19] *Wind tunnel project instructions - MTF236*.
- [20] *Siemens - Computational Fluid Dynamics simulation*. 2026. URL: <https://www.siemens.com/en-us/technology/computational-fluid-dynamics-cfd-simulation/>.
- [21] H K Versteeg and W Malalasekera. *An Introduction to Computational Fluid Dynamics Second Edition*. Tech. rep. 2007. URL: www.pearsoned.co.uk/versteeg.
- [22] Lars Davidson. *Publication 97/2 An Introduction to Turbulence Models*. Tech. rep. 2011. URL: <http://www.tfd.chalmers.se/>.
- [23] *Autodesk CFD 2024*. URL: <https://help.autodesk.com/view/SCDSE/2024/ENU/?guid=GUID-0F5C4828-9F91-46B6-A16A-2578D72DCFCC>.
- [24] W. Slater James. *NASA - Uncertainty and Error in CFD Simulations*. Feb. 2021. URL: <https://www.grc.nasa.gov/www/wind/valid/tutorial/errors.html>.
- [25] W. Slater James. *NASA - Examining Iterative Convergence*. Feb. 2021. URL: <https://www.grc.nasa.gov/www/wind/valid/tutorial/iterconv.html>.
- [26] Christoffer Johansson. *Residuals in Simcenter STAR-CCM+*. Apr. 2023. URL: <https://volute.com/simcenter-star-ccm/residuals-in-simcenter-star-ccm/>.
- [27] *Simscale - Mesh Sensitivity Study for CFD Simulations*. Dec. 2025. URL: <https://www.simscale.com/knowledge-base/mesh-sensitivity-cfd/>.
- [28] Choudhary Deepak S. *CFD Boundary Conditions - Practical Inlet & Outlet Guide*. Dec. 2025. URL: <https://gaugehow.com/simulation/cfd-boundary-conditions-inlet-outlet-guide>.
- [29] da Silva Andrade Sarah. *Understanding And Applying Boundary Conditions in CFD*. June 2023. URL: <https://www.diabatix.com/blog/understanding-and-applying-boundary-conditions-in-cfd>.
- [30] *SimFlow - Slip*. URL: <https://help.sim-flow.com/boundary-conditions/slip>.
- [31] *SimFlow - Symmetry*. URL: <https://help.sim-flow.com/boundary-conditions/symmetry>.
- [32] Longfei Zhou et al. “Additive Manufacturing: A Comprehensive Review”. In: *Sensors* 24.9 (May 2024). ISSN: 14248220. DOI: 10.3390/s24092668.
- [33] *3D printing Settings and Parameters: Types and Optimization*. Sept. 2024. URL: <https://www.raise3d.com/blog/3d-printing-settings-and-parameters/>.
- [34] *Chalmers Laboratory of Fluids and Thermal Science*. Nov. 2023. URL: <https://www.chalmers.se/en/infrastructure/chalmers-laboratory-of-fluids-and-thermal-science/facilities/>.
- [35] Sahil Gupta and Kishal Saxena. *AERODYNAMICS ANALYSIS OF A FORMULA SAE CAR*. Tech. rep. 2017.
- [36] Alessandro Piferi. “RANS-based aerodynamic optimization of a pickup truck: performance assessment and verification of add-on improvements”. Doctoral dissertation. Chalmers University of Technology, 2025.

A

Appendix 1

A.1 Python Code Used to Process The Wind Tunnel Data

```
1 import numpy as np
2 import matplotlib.pyplot as plt
3 import pandas as pd
4
5 #All variables necessary for the calculations
6 L = 0.305
7 S=1.6
8 A=np.array([4.108662e-02,4.108662e-02,4.108662e-02,4.108662e-
   -02,4.108662e-02,4.108662e-02,4.108662e-02,4.108662e-
   -02,4.108662e-02,4.170029e-02,4.144494e-02])
9 ordning=['Lucka:', 'Driver:', 'Vinge:']
10 data = np.loadtxt('BSC2026.txt', skiprows=1, usecols=(1, 2,
   3, 4, 5, 11, 6, 9))
11 densitet = data[:, 4]
12 hastighet = data[:, 1]
13 downforce = data[:, 5]
14 viskositet = data[:, 6]
15 Drag_force = data[:, 7]
16
17 #All the functions necessary for the calculations
18 def reynolds(hastighet, L, viskositet, densitet):
19     return (hastighet * L * densitet) / viskositet
20
21 def C_d(Drag_force, densitet, hastighet, A,S):
22     C_d_tunnel = (2*Drag_force) / (densitet * hastighet**2 * A
   )
23     C_d_true = C_d_tunnel * (1-A/S)**(1.288)
24     return -C_d_true
25
26 def C_l(downforce, hastighet, densitet, A):
27     C_l_tunnel = (2*downforce) / (densitet * hastighet**2 * A)
28     C_l_true = C_l_tunnel * (1-A/S)**(1.288)
29     return C_l_true
30
31 def downforce_true(C_l, densitet, hastighet, A):
```

```

32     return 0.5 * C_l * densitet * hastighet**2 * A
33
34 def dragforce_true(C_d, densitet, hastighet, A):
35     return 0.5 * C_d * densitet * hastighet**2 * A
36
37 #Plots showing the relationship between Reynolds and C_d, and
    Reynolds and C_l
38 plt.plot(reynolds(hastighet[2:9], L, viskositet[2:9],
    densitet[2:9]), C_d(Drag_force[2:9], densitet[2:9],
    hastighet[2:9], A[2:9], S))
39 plt.title("Reynolds vs C_d")
40 plt.xlabel("Reynolds")
41 plt.ylabel("C_d")
42 plt.show()

```

A.2 Wind Tunnel Balance Output

```

1 Name DP U T Pa rho nu Uhw Tb D S L l m n Scaled Voltages (E0-E8)
2 Re5 15.250134 5.022185 21.75 102330 1.209 1.507424E-5 0.000000 NaN
    -0.702769564 -0.0473880570 -1.25468150 -0.00475355566 0.146504785
    -0.00140437702 4.99911468 4.70552704 -1.42254614 4.48803295 0.181340943
    0.162530015
3 Re10 58.446546 9.831889 21.75 102330 1.209 1.507431E-5 0.000000 NaN
    -2.70584057 -0.121278307 -5.13269417 -0.0270427665 0.511748108
    -0.00472006946 19.4208125 18.0979735 -3.61800663 17.2820969 0.437630337
    0.489497094
4 Re15 131.099233 14.727741 21.86 102330 1.209 1.508397E-5 0.000000 NaN
    -6.10019142 -0.239866097 -11.7917508 -0.0630026579 1.17960795
    -0.0109470212 44.5459408 41.5683647 -8.22538935 38.9703879 0.850723032
    1.01176534
5 Re20 234.277869 19.690333 21.96 102340 1.209 1.509143E-5 0.000000 NaN
    -11.0161624 -0.457991297 -21.5239917 -0.109881882 2.11422251
    -0.0200943464 80.7253239 75.4477307 -13.9801005 70.3779087 1.63801560
    1.89680775
6 Re25 369.107706 24.725815 22.21 102340 1.207 1.511447E-5 0.000000 NaN
    -17.3370086 -0.709115276 -34.1265246 -0.174493672 3.30467769
    -0.0311201321 127.360660 119.026425 -20.9219196 110.762470 2.51544207
    2.97670338
7 Re30 534.248868 29.756815 22.40 102340 1.207 1.513177E-5 0.000000 NaN
    -24.9626031 -1.07942332 -49.8302123 -0.251793526 4.64643027
    -0.0348222863 183.745009 171.596819 -26.0675573 159.482224 3.61166515
    4.72598706
8 Simplified 728.757821 34.777000 22.79 102340 1.205 1.516710E-5 0.000000 NaN
    -34.6545907 -2.24148557 -68.8668615 -0.443974444 6.92456656
    -0.205670802 262.478799 240.023826 -47.4841445 221.447138 11.7774185
    4.63194267
9 Driver 731.109959 34.856037 23.18 102340 1.204 1.520254E-5 0.000000 NaN
    -35.1754728 -2.12847077 -69.4160406 -0.436050696 6.86490068
    -0.165453057 263.073154 241.213507 -45.6533662 224.747120 10.3496684
    5.36832550
10 Tilted_wing 733.001134 34.898852 23.14 102340 1.204 1.519910E-5 0.000000
    NaN -34.2332496 -1.67767901 -73.3925036 -0.366872426 3.63768412

```

-0.0561499035 233.554723 215.692806 36.4513010 218.487346 6.15797054 7.13024240
--

DEPARTMENT OF MECHANICAL ENGINEERING
CHALMERS UNIVERSITY OF TECHNOLOGY

Gothenburg, Sweden
www.chalmers.se



CHALMERS
UNIVERSITY OF TECHNOLOGY