

Aerodynamic analysis and design of a solar powered vehicle using CFD

A.Z. Fourie
21588635

Dissertation submitted in fulfilment of the requirements
for the degree ***Magister*** in **Mechanical Engineering** at the
Potchefstroom Campus of the North-West University

Supervisor: Dr J.J. Bosman

May 2016

Abstract

The need for solar-powered vehicles is increasing as the threat of depletion of non-renewable energy sources, such as oil, increases. This gives reason for developing electric vehicles powered by renewable energy sources, such as solar power. An important aspect for the overall efficiency of such vehicles is the aerodynamic design. To improve the design, proper analysis is essential. A generically based methodology describes the design process applied to improve the aerodynamic performance of these vehicles using Computational Fluid Dynamics (CFD) as the design tool. CFD is a valuable tool as it accelerates the design process within a 3-D flow environment. The literature survey conducted during this study describes basic fluid properties relevant to the flow around a body. It also describes the different turbulence models used in CFD and the appropriate transition model to be used. Finally, the factors of the body of a solar-powered vehicle that can be optimised to improve the aerodynamics are also discussed. The methodology followed is an iterative process comprising of the improvement of the baseline design. It was found that this iterative method enables the designer to improve the design by analysing various vehicle revisions using CFD and adjusting certain aerodynamic factors described in the literature. The aim is to minimise the design parameters with each iteration; in other words the frontal area of the vehicle as well as the drag coefficient. Other factors greatly affecting the drag coefficient are the junction fillets of the different components as well as the positions of these components relative to each other. By varying these factors, the near optimum values could be correlated with the theoretical values and based on this the design could be improved. Proper airfoil designs should also be implemented when designing the components to improve the laminar flow. The results found illustrate that the process followed for the analysis and design of the vehicle provide a suitable method for related problems in future designs. CFD is also determined to be a suitable design tool. The end result can, however, be validated experimentally, as an assumption was made that the CFD simulations were adequate, as shown by the results of the validation process. Comparing the simulation results with experimental results will further prove the validity of the study.

Acknowledgements

Firstly all praise to my heavenly Father for the strength He has given me during the duration of this study.

I would like to thank my family – parents Merwe and Christa, brothers Liaan and Stephan and sister-in-law Michelle – for their continued support and motivation throughout the duration of this study, and all my friends who also supported and motivated me.

Thanks are also due to:

Dr Johan Bosman for his valuable advice as supervisor and for the financial support required to complete this study.

Prof Jat Du Toit for help with formulating the title.

Christiaan de Wet (Aerotherm) for hours spent aiding me with countless simulations and valuable advice in this regard.

Thabo Molambo for the support and use of the HPC (NWU High Performance Computer cluster).

Kitso Epema (TU Delft's Human Power Team) for the information on the HPT's VeloX1 geometry and wind tunnel test results.

Finally, to Arno de Beer and the team for the time spent building the actual vehicle and competing in the Sasol Solar Challenge.

Table of Contents

Abstract.....	i
Acknowledgements.....	ii
Table of Contents	iii
List of figures	vii
List of tables.....	xi
Nomenclature.....	xii
Chapter 1: Introduction.....	1
1.1 <i>Background</i>	1
1.2 <i>Problem Statement</i>	2
1.3 <i>Research Aims and Objectives</i>	3
1.4 <i>Significance</i>	3
1.5 <i>Definition of key terms</i>	4
Chapter 2: Literature Survey.....	5
2.1 <i>Aerodynamic Drag</i>	6
2.1.1 <i>Drag Components.....</i>	7
2.2 <i>The Boundary Layer.....</i>	11
2.2.1 <i>Laminar and Turbulent Flow.....</i>	12
2.2.2 <i>Separation.....</i>	14
2.3 <i>Transition Modelling with CFD.....</i>	15
2.3.1 <i>The γ-Re_θ Transition Model</i>	16
2.4 <i>CFD as a Design Tool.....</i>	18

2.5	<i>Aerodynamic Design</i>	20
2.5.1	Vehicle Body Shape	20
2.5.2	Nose design	24
2.5.3	Interference Drag Reduction	25
2.5.4	Canopy Drag Reduction	29
2.5.5	Trailing Edge Thickness of the Main Body.....	31
2.6	<i>Popular Solar-powered Vehicles</i>	32
2.7	<i>Summary</i>	37
Chapter 3: Methodology		38
3.1	<i>Review Regulations and Evaluate Existing Solutions</i>	39
3.1.1	Summary of previous work	39
3.2	<i>Geometric Specifications and Initial Design Parameters</i>	39
3.3	<i>Create Concept Designs</i>	41
3.4	<i>Selection of Best Concept Design</i>	42
3.5	<i>Detailed Design</i>	42
Chapter 4: 2-D Validation: NACA 0018		44
4.1	<i>Validation Setup</i>	44
4.1.1	Mesh Configuration.....	45
4.1.2	Solver Configuration	48
4.2	<i>Validation Procedure</i>	49
4.3	<i>Validation Results</i>	50
4.4	<i>Conclusion</i>	57
Chapter 5: 3-D Validation: HPT VeloX1		58
5.1	<i>Validation Setup</i>	59

5.1.1	Mesh Configuration.....	60
5.1.2	3-D Solver Configuration.....	62
5.2	<i>Validation Procedure</i>	62
5.3	<i>Validation Results</i>	63
5.4	<i>Conclusion</i>	68
Chapter 6: Conceptual Design		69
6.1	<i>Concepts</i>	69
6.1.1	Concept based on Symmetry	70
6.1.2	Asymmetrical Concept with Long Fairings	70
6.1.3	Asymmetric Design with Short Fairings	72
6.2	<i>Comparison Setup</i>	74
6.3	<i>Comparison Procedure</i>	74
6.4	<i>Comparison Results</i>	75
6.5	<i>Conclusion</i>	78
Chapter 7: Detail Design.....		79
7.1	<i>Component Analysis</i>	79
7.1.1	Analysis Procedure.....	79
7.1.2	Main Vehicle Body	79
7.1.3	Fairing Profile Selection.....	82
7.1.4	Fairing Design	86
7.1.5	Canopy Design	92
7.2	<i>CFD Analysis of Detail Design</i>	99
7.2.1.	Model Based on Near Optimum Values.....	102
7.2.2.	Effect of Increasing Fillet Radii on Fairings	105
7.2.3.	Effect of Reducing Fillet Radii to Below Theoretical Optimum.....	107

7.2.4.	Applying the Fillet Radius Theory to Canopy Junctions	109
7.2.5.	Effect of a Large Canopy Junction Fillet.....	110
7.2.6.	Effect of Diverting Flow around the Rear Fairing	110
7.2.7.	Applying Fillet Radius Theory to the Driver Cockpit Fairing	113
7.2.8.	Effect of Removing Sharp Floor Edges on Fairings.....	114
7.3	<i>Detail Design Results</i>	116
7.4	<i>Conclusion</i>	117
Chapter 8:	Conclusion and Recommendations	119
8.1.	<i>2-D Validation</i>	119
8.2.	<i>3-D Validation</i>	119
8.3.	<i>Detail Design</i>	120
8.4.	<i>Recommendations for Future Work</i>	120
Bibliography		122
Annexure A:	2-D Validation Summary Report	128
Annexure B:	3-D Validation Summary Report	138
Annexure C:	Detail Design Analysis Summary Report	148
Annexure D:	Final Remarks	169

List of figures

Figure 1 - Solar-powered vehicles (Tour de Sol 1987) [4].....	5
Figure 2 - Aerodynamic drag and rolling resistance against vehicle speed [7].....	7
Figure 3 - Comparison between (a) a bluff body and (b) a streamlined body [8].....	8
Figure 4 - Wake produced by different body shapes [10].....	9
Figure 5 - Induced drag and lift [12]	10
Figure 6 - Boundary layer along a thin flat plate [14].....	12
Figure 7 - Natural transition process [15]	14
Figure 8 – Schematic of flow separation at a wall [17]	15
Figure 9 - Summary of how camber effects the lift force on the body in freestream and in close ground proximity [5]	21
Figure 10 - Tapering of the trail edge of a bluff body (boat-tailing) [30].....	22
Figure 11 - Body at trim [32].....	23
Figure 12 - Airfoil contribution to longitudinal stability [33]	23
Figure 13 - Junction flows between the fairing and the body [35]	26
Figure 14 - Inclination angle of an appendage [5]	27
Figure 15 - Fillet radius along an appendage [5].....	28
Figure 16 - The upper and lower bounds of the canopy drag as a function of the L/h ratio [5]	30
Figure 17 - (a) Illustration of baseline canopy with $L/h = 6$. (b) Base line canopy with a blunt nose. (c) Baseline canopy with a blunt tail. (d) Baseline canopy with a blunt nose and tail [5]	31
Figure 18 - Trailing edge thickness plotted against the percentage increase in drag area [5]	32
Figure 19 - Nuna6 solar vehicle [43]	33
Figure 20 - Nuna7 solar vehicle [44]	34
Figure 21 - Tokai University solar-powered vehicle [45].....	35
Figure 22 - Solar Team Twente's "The Red Engine" solar-powered vehicle [42].....	36
Figure 23 - Domain with boundaries defined.....	45
Figure 24 – Quadrilateral 2-D mesh.....	46

Figure 25 - Overset mesh: airfoil rotated to an angle of attack of 8°	47
Figure 26 - Prism layer on the leading edge of the NACA 0018 airfoil	47
Figure 27 - Mesh refinement for the 2-D simulation at an angle of attack of 0°	49
Figure 28 - Lift and Drag coefficients, 8° angle of attack	51
Figure 29 - Lift coefficient compared with the Drag coefficient, $Re = 0.7 \times 10^6$	52
Figure 30 - Lift coefficient compared with the angle of attack, $Re = 0.7 \times 10^6$	53
Figure 31 - Separation point on airfoil	54
Figure 32 - Skin friction coefficient plot for a 2-D NACA 0018 airfoil, 0° angle of attack	55
Figure 33 - Pressure coefficient of the 2-D NACA 0018 airfoil, 8° angle of attack.....	56
Figure 34 - Separation bubble formation on the upper airfoil surface, 8° angle of attack.....	56
Figure 35 – Human Power Team’s VeloX1 [57]	58
Figure 36 - Domain description for the 3-D validation case	60
Figure 37 - VeloX1 geometry.....	61
Figure 38 - Mesh refinement for VeloX1 trailing edge flow	61
Figure 39 - VeloX1 prism layer on surface mesh.....	62
Figure 40 - STAR-CCM+ force monitor plot	64
Figure 41 - VeloX1 transition location with the ventilation ducts taped [57]	65
Figure 42 – VeloX1 transition location by VSAero [57].....	66
Figure 43 - VeloX1 transition location by STAR-CCM+, turbulent kinetic energy plot.....	66
Figure 44 - HPT VeloX1 pressure coefficient by VSAero.....	67
Figure 45 - VeloX1 pressure coefficient by STAR-CCM+.....	67
Figure 46 – First concept design (symmetrical).....	70
Figure 47 – Two long fairings with driver to the right hand side of the vehicle.....	71
Figure 48 – Front view of the second concept.....	72
Figure 49 – Asymmetric design with three fairings	73
Figure 50 – Front view of the third design, showing the effect that the shorter fairings has on the frontal area.....	73
Figure 51 - Turbulent kinetic energy of the first concept.....	75

Figure 52 - Bottom view of the symmetric concept vehicle.....	76
Figure 53 - Turbulent kinetic energy of the belly section of the second concept.....	77
Figure 54 - Turbulent kinetic energy of the belly section of the third concept.....	78
Figure 55 – Modified JS1 sailplane wing profile used for main body	80
Figure 56 - Pressure coefficient curves of main body profile	80
Figure 57 - Main body of the vehicle	81
Figure 58 - Rounded leading-edge of the main body.....	82
Figure 59 - Laminar flow on leading-edge of main body.....	82
Figure 60 - RUD20 airfoil	83
Figure 61 - YS900 airfoil.....	83
Figure 62 - NACA 16015 airfoil	84
Figure 63 - NACA 16015 and YS900 airfoils combination	85
Figure 64 - Pressure distribution along the chord length of the NACA 16015 YS900 airfoil.....	85
Figure 65 – Fairing Configuration 1.....	86
Figure 66 - Transition locations of flow on fairing Configuration 1.....	87
Figure 67- Pressure coefficient plot of Configuration 1.....	87
Figure 68 – Fairing Configuration 2.....	88
Figure 69 - Transition locations of flow on fairing Configuration 2.....	89
Figure 70 - Pressure coefficient plots of configuration 2	90
Figure 71 – Fairing Configuration 3.....	91
Figure 72 - Transition locations of flow on fairing Configuration 3.....	91
Figure 73 - Pressure coefficient plots of configuration 3	92
Figure 74 - Canopy 1 turbulent kinetic energy plot	93
Figure 75 - Pressure coefficient plot of canopy 1.....	94
Figure 76 - Canopy 2 turbulent kinetic energy plot	95
Figure 77 - Pressure coefficient plot of canopy 2.....	96
Figure 78 - Canopy 3 turbulent kinetic energy plot	97
Figure 79 - Pressure coefficient plot of Canopy 3.....	98

Figure 80 – Mesh independence	100
Figure 81 - Detail design simulation boundary definitions.....	101
Figure 82 - Laminar flow on upper part of the baseline model	103
Figure 83 - Laminar flow on right side of the baseline model	103
Figure 84 - Laminar flow on lower part of the first vehicle model	104
Figure 85 - Laminar flow on upper surface of the model after changing the fillet radii to 60 mm	106
Figure 86 - Transition at fairing leading-edges of the revised model	107
Figure 87 - Transition at fairing leading-edges of the 30 mm fillet radii model.....	108
Figure 88 - Canopy wake of the present model	109
Figure 89 - Laminar flow on the sixth model.....	111
Figure 90 - Laminar flow on inside of fairings	112
Figure 91 - Fairing wake due to the modified front fairing.....	112
Figure 92 – Increase in frontal area due to larger fillets on the right hand fairing (left side in above image)	113
Figure 93 - Rounded lower edges of fairings.....	114
Figure 94 - Improved laminar flow on lower edge of fairings.....	115
Figure 95 - Improved laminar flow on rear left fairing	115
Figure 96 - Drag coefficients of the vehicle model when moving relative to the ground	117
Figure 97 - NWU Solar Car competing in the Sasol Solar Challenge 2014	169
Figure 98 - Sirius X25 headed to Kroonstad on Day 1 of the Sasol Solar Challenge 2014.....	171
Figure 99 – Sirius X25 travelling on the N1 highway between Kroonstad and Bloemfontein, Free State.	171
Figure 100 – On route to Hanover in the Karoo.....	172
Figure 101 - Sirius X25 completing a loop in the Karoo	172
Figure 102 - The vehicle on its way to Port Elizabeth, Eastern Cape.....	173
Figure 103 - Doing a loop between Heidelberg and Witsand in the Western Cape	173
Figure 104 - Control stop in Heidelberg, Western Cape	174
Figure 105 - Sirius X25 and team at the finish in Cape Town.....	174

List of tables

Table 1 - Definition of key terms.....	4
Table 2 - Relative sizes of the drag components and variables to which each is proportional [5]	11
Table 3 - Atmospheric Conditions	48
Table 4 - Drag Area Results Comparison	65
Table 5 - Conceptual design comparison.....	75
Table 6 - Drag force results of the stationary vehicle.....	116
Table 7 - Drag force results of the moving vehicle	116

Nomenclature

A	Frontal area [m ²]
c	Chord length [m]
$C_D A$	Aerodynamic drag area [m ²]
C_D	Aerodynamic drag coefficient [-]
C_L	Lift Coefficient [-]
C_m	Pitching Moment Coefficient [-]
C_p	Pressure Coefficient [-]
$C_{d,frontal}$	Drag coefficient based on frontal area [-]
ΔC_d	Change in drag coefficient [-]
ΔD_j	Drag increase due to junctions [N]
D	Aerodynamic drag force [N]
D_{ind}	Induced drag force [N]
D_{int}	Interference drag force [N]
$D_{p,sep}$	Pressure drag force [N]
$D_{p,BL}$	Boundary layer drag force [N]
D_{skin}	Skin drag force [N]
δ	Boundary layer thickness [m]
h	Height [m]

L	Length [m]
P_{loc}	Local pressure [Pa]
P_{∞}	Pressure outside the boundary layer [Pa]
ρ	Air density [kg/m ³]
Re	Reynolds number
$Re_{\theta t}$	momentum thickness Reynolds number
t	Maximum thickness of body airfoil [m]
t_{te}	Thickness of trailing edge [m]
U_{∞}	Free stream velocity [m/s]
ν	Kinematic viscosity [m ² /s]
V	Flow velocity [m/s]
V_{loc}	Local velocity [m/s]
V_{∞}	Velocity outside the boundary layer [m/s]
x	Distance from leading-edge [m]

Acronyms

CAD	Computer Aided Design
CFD	Computational Fluid Dynamics
CPU	Central Processing Unit
DNS	Direct Numerical Simulation

HPT	Human Power Team
LCTM	Local Correlation-based Transition Model
LES	Large Eddy Simulation
NACA	National Advisory Committee on Aeronautics (USA)
RANS	Reynolds Averaged Navier-Stokes
SST	Shear Stress Transition model
SV	Solar-powered Vehicle

Keywords

Computational Fluid Dynamics, CFD, aerodynamics, aerodynamic drag, analysis and design, solar-powered vehicle, drag coefficient, STAR-CCM+

Chapter 1: Introduction

1.1 Background

Road vehicles are one of the primary transportation systems and are largely powered by oil, a non-renewable source of energy. The threat of depleting non-renewable energy sources has encouraged research into finding alternative sources of energy. Converting solar energy into electricity is such an alternative source. This has induced the application of solar energy to power electric vehicles.

The aerodynamic drag component of a vehicle's design plays an important role in its performance and has to be optimised [1]. Solar panels on typical solar-powered race vehicles usually generate about 1500 W of electricity, equivalent to the amount used by a hair drier. For this energy to be used effectively, the drag resistance of the vehicle should be optimised to be as low as possible. It therefore encourages research to be conducted in optimising the aerodynamic designs of these solar-powered vehicles.

A wind tunnel as a design tool is essential in the aerodynamic development of road vehicles, including solar-powered vehicles, but building models to test in a wind tunnel is time consuming and expensive. A wind tunnel in this case may not always be an economical and viable tool. Computational Fluid Dynamics (CFD) can be used as an alternative to wind tunnel testing. CFD codes simulate detailed fluid flow problems and allow these problems to be analysed during the early stages of the detail design [2]. This can possibly result in reduced costs and early detection and avoidance of possible problems during the design and manufacturing processes.

CFD was therefore used to analyse the baseline design for a solar-powered vehicle (SV) designed at the North-West University. The results of the first generation SV were used as the baseline from which the next generation SV was designed. The vehicle is designed and built in order to compete in the Sasol Solar Challenge. Both local and international solar car teams compete in the South African Sasol Solar Challenge endurance competition held on the South African national roads.

New regulations in the competition meant that the aerodynamic design of the first generation vehicle had to be modified. These rules included a change in the vehicle dimensions as well as requiring a larger canopy for increased driver comfort.

1.2 Problem Statement

With recent developments in photovoltaic solar power technology, solar cell performance has increased significantly, reaching efficiency values as high as 40%. However these efficiencies come at a price and are therefore too expensive for most applications.

For increased endurance performance of the solar car, one of the options currently available is the improvement of the aerodynamic shape of the vehicle.

The current (first generation) solar car design of the NWU has typical features that increase the aerodynamic drag significantly. These features include a swing-arm rear suspension that protrudes from the body. This increases the frontal area and which increases the overall drag coefficient.

Solar vehicles which performed well in the past, have drag coefficient values of less than 0.1. The current solar vehicle design has a drag coefficient of almost 0.25, meaning that it is much less efficient than the current world leaders' designs. The teams that consistently compete for the number one position at the World Solar Challenge every other year, complete the challenge at an average speed in excess of 80 km/h, with top speeds of about 110 km/h. During the previous solar challenge, the current solar vehicle could only maintain an average speed of about 35 km/h and maintain a top speed of 110 km/h for a very limited time (less than one hour). The current design of the NWU can be improved in many areas, each of which is a study in its own. The most significant improvement can be achieved by improving the aerodynamic design.

In order to improve the aerodynamics of the current vehicle, the complete 3-D flow field around the vehicle has to be analysed. Analysing this 3-D flow field can show how the combination of the different body components increases the drag of the vehicle. This can be in terms of reduced laminar flow regions on the body due to increased interference and poor body orientation. Using CFD, the aerodynamic drag coefficients of the solar vehicle can be determined by calculating the aerodynamic force on the surface of

the vehicle. In order to correctly determine these drag coefficients, transition locations should also be calculated accurately as both laminar and turbulent flow regions are present at these operating Reynolds numbers. If transition is predicted incorrectly, the drag coefficients will also be inaccurate.

Recently transition models have become readily available in CFD software packages. However, before these transition models can be used to design new aerodynamic shapes, the accuracy of the numerical simulations should be determined. In determining the adequacy of CFD as a design tool for solar vehicles, a validation of the transition models must be conducted.

1.3 Research Aims and Objectives

- Validate CFD transition model for flow environments similar to solar vehicles.
- Optimise the aerodynamic design of a solar-powered vehicle within the regulations as specified by the Sasol Solar Challenge, specifically section 1 and section 2 [3], using CFD. The main factor to consider is the aerodynamic drag of the vehicle. A 5% reduction in the drag coefficient is deemed notable, if a near optimum base model (based on theory) is selected. An improvement in excess of 10% is deemed acceptable when the new design is compared to the existing vehicle design.

1.4 Significance

Many vehicle manufacturers such as Toyota and Nissan have established a market for electric hybrid vehicles. These vehicles typically have a small internal combustion engine, only used for long distance travelling or when the power supplied by the electric motor is exceeded. The vehicle will have a battery storage system to supply energy to the electric motors and will usually be charged using electricity from a grid. Many manufacturers also use solar arrays to supply electricity for applications within the vehicle such as the air conditioning and navigation systems.

The research conducted in this dissertation can therefore further the knowledge of building efficient and affordable road vehicles powered by solar energy. There is also the option to build a hybrid vehicle, but instead of charging the batteries from grid electricity, the batteries will be charged by the solar arrays. Further research will allow these production line vehicles to run solely on solar power, without any internal combustion engines.

1.5 Definition of key terms

The key terms used in this project are summarised in Table 1.

Table 1 - Definition of key terms

Computational Fluid Dynamics (CFD)	A division within fluid mechanics using numerical methods and algorithms to solve and analyse fluid flow problems.
Computer Aided Design (CAD)	Designs done using a computer to represent a model of the part or vehicle as it will be constructed.
Solar-powered Vehicle (SV)	Refers to the solar-powered vehicles constructed as a result of the research conducted in this project.
Human Power Team (HPT)	A team of students from TU Delft in the Netherlands, who built a bicycle with which they intend to break the land speed record for a human powered vehicle

Chapter 2: Literature Survey

Solar car racing first started with the Tour de Sol competition held in 1985, leading to other similar competitions worldwide [4]. These competitions were mainly aimed at university students to find innovative ways to expand their skills and develop solar technology.



Figure 1 - Solar-powered vehicles (Tour de Sol 1987) [4]

There are many different types of solar-powered cars being built by different institutions, such as the one in Figure 1. This was one of the first solar-powered racing vehicles built in 1987. Contestants from all sectors are allowed to participate, from privateers building vehicles out of the cheapest materials available to highly funded institutions such as universities having access to the latest technologies in order to build the most efficient vehicles possible.

Two methods are generally applied in the reduction process of the aerodynamic drag, namely the ground-up approach and the improvement approach [5]. The ground-up approach is where the main body is designed for lower drag coefficients and the other components are then designed within the body constraints. The improvement approach is where the designer will improve the features of an already non-aerodynamic vehicle. These processes will ensure a reduction in the drag of the vehicle.

2.1 Aerodynamic Drag

An object with a higher air resistance will require more energy to be displaced. Aerodynamic drag is one of the main factors adversely affecting the efficiency of a road vehicle or a body moving through air, and should therefore be optimised. The remainder of the resistance will be due to the rolling resistance caused by the friction between the tyres and the road surface [5].

The aerodynamic drag of a vehicle is determined by the frontal area (A), the drag coefficient (C_D), the square of the sum of the vehicle's velocity (V_{car}) and the headwind velocity (V_{head}) and the air density (ρ). The drag coefficient of the vehicle is also described as a measure of the flow quality around the vehicle. According to Tamai [5], aerodynamic drag (D) can be expressed as:

$$D = (C_D A) \frac{1}{2} \rho (V_{car} + V_{head})^2 \quad 2.1$$

V_{car} is the velocity of the car relative to the road and V_{head} is the velocity of the headwind relative to the road. The sum of these velocities will be the velocity of the air relative to the car (V). The above equation can thus be simplified to the following equation, as described by Smith [6]:

$$D = (C_D A) \frac{1}{2} \rho (V)^2 \quad 2.2$$

In order to reduce the drag of the vehicle, one has to analyse this equation. The density of the air, ρ , has an effect on the drag of the vehicle but one cannot adjust it to reduce the drag of the body. The velocity, V at which the vehicle is travelling, is always kept at a constant value which will deliver the best performance, but the other two factors do have an effect. By decreasing the frontal area, there is a direct effect on the drag of the vehicle as they are directly proportional to one another. One can further reduce the drag of the vehicle by designing a more aerodynamic shape that moves easier through air, thus with a lower C_D value.

The total drag of the vehicle, called the drag force, will be the sum of the aerodynamic drag and the rolling resistance. Refer to Figure 2 to see the relationship between the aerodynamic drag and the rolling resistance at different speeds [7].

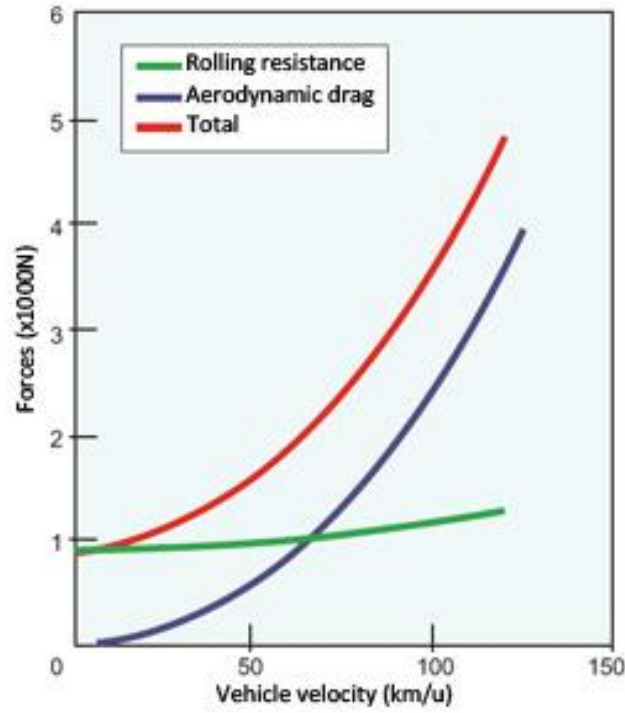


Figure 2 - Aerodynamic drag and rolling resistance against vehicle speed [7]

The drag forces will increase as the vehicle speed increases, with the aerodynamic drag having the largest effect.

2.1.1 Drag Components

The aerodynamic drag (D) of the body has several components, namely pressure drag or form/profile drag ($D_{p,sep}$), viscous friction drag or surface drag ($D_{skin} + D_{p,BL}$), induced drag (D_{ind}) and interference drag (D_{int}).

According to Tamai [5], the aerodynamic drag can thus be expressed as:

$$D = D_{p,sep} + (D_{skin} + D_{p,BL}) + D_{ind} + D_{int} \quad 2.3$$

where D_{skin} is the drag caused by the skin friction and $D_{p,BL}$ is the pressure drag caused by the boundary layer, the sum of which makes up the viscous friction drag.

Pressure Drag

Pressure drag occurs when the air flow around the body does not stay attached to the body all the way to the trailing edge of the vehicle. This phenomenon is called flow separation. When this does occur, a low-

pressure wake develops behind the vehicle. This causes a sucking force, pulling the vehicle backwards. This also results in vortices behind the body, which consumes large amounts of energy. A body in which separation occurs, is called a bluff body. Pressure drag, or separation pressure drag, is induced by surface irregularities (surface or skin friction) which causes the flow to separate from the body. The frontal area of the vehicle also has an effect on the separation of flow.

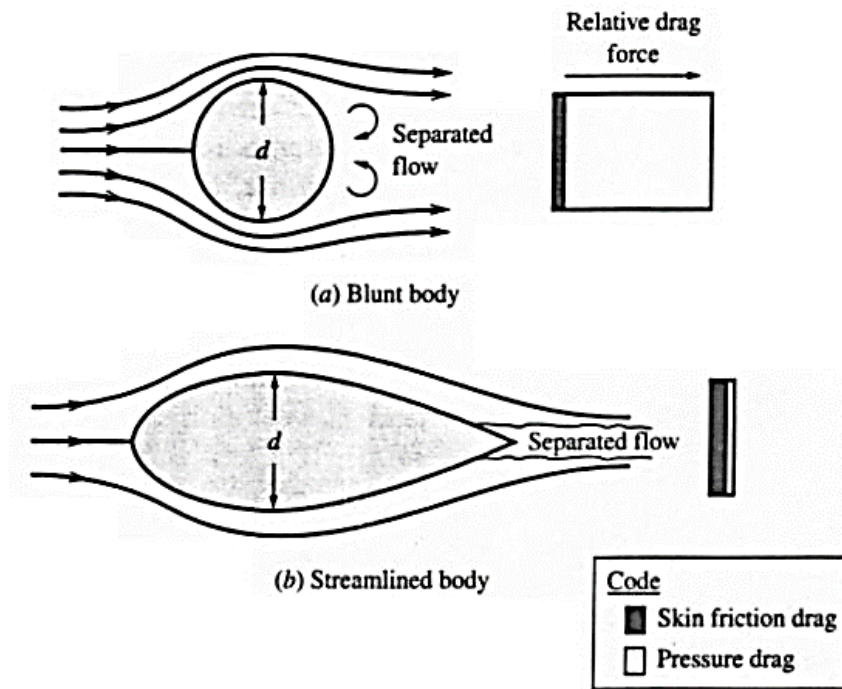


Figure 3 - Comparison between (a) a bluff body and (b) a streamlined body [8]

Referring to Figure 3 [8], it is clear that (a) a bluff/blunt body has much more pressure drag than (b) a streamlined body due to the larger wake pulling the object backwards.

Viscous Friction Drag

The viscous friction drag, also called friction drag, of the body has two components. The first of these is the drag caused by the skin friction, which is the viscous shearing of fluid molecules tangential to the surface.

In aerodynamic applications, flow separation is not ideal and the design should be such that the airflow remains attached for as long as possible. A body in which there is no, or very little, flow separation present

is called a streamlined body. It should be noted that a bluff body has five to ten times the aerodynamic drag of a streamlined body [9]. Refer to Figure 3 to see the differences in the relative drag forces.

The second component of the viscous friction drag of the body is the pressure drag due to the boundary layer. The boundary layer flow starts at the nose of the vehicle at the stagnation point, where the flow velocity is equal to zero and the pressure is at its maximum.

Fluid velocity will increase as it flows over the body and the pressure will decrease. The pressure will reach a minimum at the thickest section of the body. At this point the pressure will increase again and aim for the stagnation value, but will not be able to achieve this value as the boundary continuously thickens around the body. Therefore, full pressure recovery will never be reached. This creates a small wake behind the vehicle, having a similar effect as the wake caused by the bluff body, albeit not as large. Refer to Figure 4 [10].

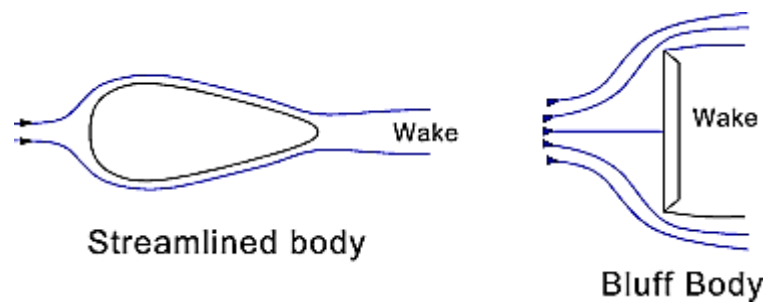


Figure 4 - Wake produced by different body shapes [10]

The sum of the pressure drag and the viscous friction drag is called the profile drag and is largely based on the wetted surface area of the body of the vehicle.

Induced Drag

The induced drag of the body is a result of the lift generated by the airflow around the vehicle [11]. Figure 5 can be used to describe the induced drag on a wing [12]. The lift force increases or decreases as the airflow around the airfoil changes. As the angle of attack of the airfoil is increased (by tilting the nose upwards), the fluid experiences acceleration over the upper surface of the airfoil. This acceleration reduces the pressure on the same surface. This also means that the airflow along the underside of the airfoil will be slower, increasing the pressure. This increase in pressure as well as the change in direction of the flow, exerts a force on the airfoil. This force is called the lift force. The increase in the lift force also means that

the resultant force (perpendicular to the chord line of the airfoil) between the lift force and the drag force is increased. As the resultant force increases, the drag force will also increase. This drag force is referred to as induced drag.

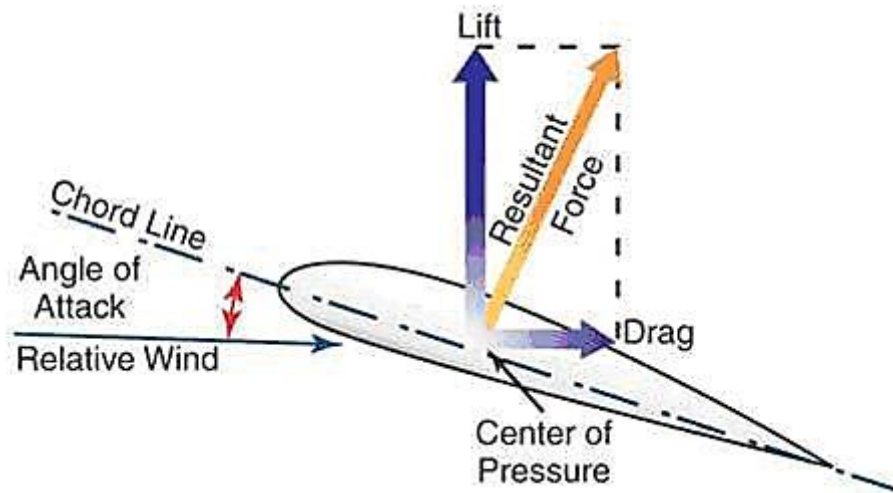


Figure 5 - Induced drag and lift [12]

This also applies to the streamlined body of a solar-powered vehicle. In this instance the angle of attack is usually zero, but the lift generated is still upward. This results in the downward force on the tires being reduced, albeit marginally. The lift induces vortices due to the pressure differences between the top and bottom of the body. These vortices use kinetic energy and this increases the drag of the vehicle. The vehicle is, however, not designed to generate lift, so induced drag is therefore not a major concern during the design procedure.

Interference Drag

Interference drag is caused by local flow disruption at the intersections of different components of the body [13]. These include the fairings-to-body and canopy-to-body junctions.

Table 2 - Relative sizes of the drag components and variables to which each is proportional [5]

[%]	Separation Pressure Drag	Viscous Drag	Induced Drag	Interference Drag
Proportional to	Frontal Area	Wetted Area	Lift	Holes, Seams, Manufacturing etc.
Bluff Body	Large	Small	Medium to Large	Small to Large
Streamlined Body	Approximately Zero	Large	Approximately Zero	Small to Large

Referring to Table 2 and to Figure 3, it is clear that the streamlined body has a much larger viscous drag, but the pressure drag has a much larger effect on the total drag of a bluff body. It is also clear that the separated flow caused by a bluff body is much larger and will have a larger suction force than the separated flow caused by the streamlined body.

Solar-powered vehicles will therefore always be designed to be streamlined in order to minimise drag.

2.2 The Boundary Layer

The boundary layer is the layer of fluid flow adjacent to the body surface. Here the effect of viscosity is important. Viscosity is the ability of the fluid to resist motion, thus the higher the viscosity of a fluid, the larger the force required to displace the fluid.

The following paragraph, in conjunction with Figure 6 [14], describes the boundary layer:

Imagine that the body moves forward such that the air velocity relative to the body surface is U_{∞} m/s, called the free stream velocity. The fluid next to the body surface does not move relative to the body surface, in other words, it has zero velocity. This is called the no-slip condition. The layer adjacent to this zero velocity layer, moves at a velocity relative to the surface of the body, but still much lower than U_{∞} . The velocities of the following layers will progressively increase until the speed is equal to U_{∞} . These layers that take the velocity from zero at the surface of the vehicle body to a value equal to the free stream velocity U_{∞} , are called the boundary layers.

The boundary layer becomes thicker as the flow progresses over the surface of the body. The thickening of the boundary layer is described in [11] using Eq. 2. 4:

$$\delta \sim \sqrt{\frac{\nu x}{U_{\infty}}} \quad 2.4$$

where the x represents the distance from the leading-edge and the ν is the kinematic viscosity. δ refers to the boundary layer thickness while U_{∞} is the free stream air velocity.

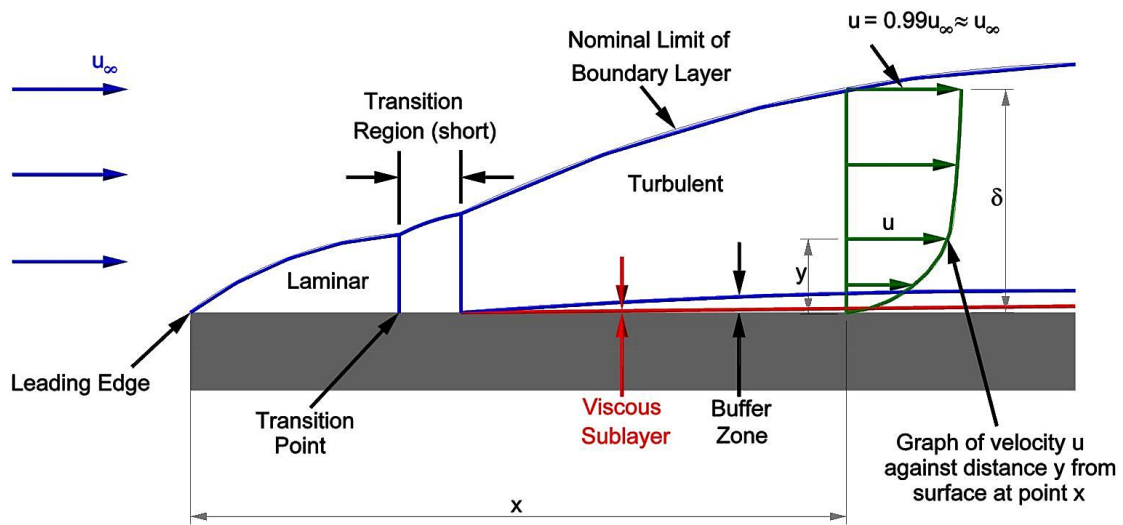


Figure 6 - Boundary layer along a thin flat plate [14]

The air flowing over the body is influenced to a greater extent by the body as the flow progresses further down the length of the body. The top layers in the boundary layer are impeded by the stationary layer next to the body surface, but still each layer will have a higher velocity than the layer beneath it. Eventually the layers will reach the free stream velocity (U_{∞}). This takes time (i.e. distance) meaning that the boundary layer has to start out thin and build on itself as the flow progresses.

2.2.1 Laminar and Turbulent Flow

Laminar flow of fluids is characterised by airflow progressing in smooth sheets without any mixing, much like the way light travels in straight beams [5]. Most flow patterns over objects start out as being laminar but then transition to turbulent flow. The point where this occurs is called transition, which is largely governed by the Reynolds Number [11].

As soon as this occurs, the fluid streams cross each other and this requires energy. The different layers absorb energy from neighbouring streams, and these then result in the lower streams to gain more energy and thus reach higher velocities. Referring to Figure 6 [14], it can be observed that the boundary layer thickens due to the increased velocities. The turbulent fluid streams also absorb energy from the free stream air, thus the turbulent boundary layer grows even thicker. Within the turbulent layer, there is still flow that remains attached to the surface of the body, called the laminar sub-layer or viscous sub-layer. This layer only accounts for about 1% of the turbulent flow, but the shear rates are much higher than the shear values in the remaining turbulent layer [5]. This means that the laminar sub-layer accounts for most of the viscous drag in the turbulent flow layer.

Natural Transition

The laminar boundary layer can become unstable at a certain critical Reynolds number, with the growth of viscous instability waves (Tollmein-Schlichting waves). This generally occurs when the free stream turbulence level is low, <1% [15]. At this stage, the waves are regarded as 2-D, but as they grow they become increasingly non-linear.

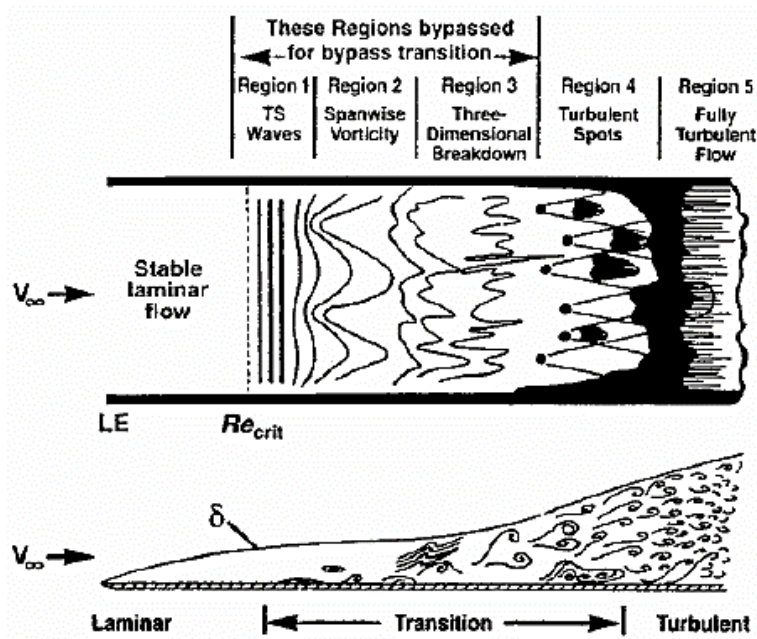


Figure 7 - Natural transition process [15]

When this happens the flow becomes predominantly 3-D and results in turbulent spots. This is the point where transition occurs. These spots will grow and mix with the laminar layer until the layer is fully turbulent (see Figure 7 [15]).

Bypass Transition

When free stream turbulence levels are high ($>1\%$), regions 1 through 3 in Figure 7 [15], which can be referred to as the linear stages; is bypassed. This means that the turbulent spots are formed directly in the boundary layer by the disturbances from the free stream flow [15]. Bypass transition is, however, not only caused by free stream disturbances. It can also be caused by the effect of the surface roughness; thus transition occurs due to disturbances at the wall of the geometry.

2.2.2 Separation

As mentioned in section 2.2.1., flow separation is the phenomenon when the fluid flow can no longer stay attached to the body, and thus transitions from laminar to turbulent flow.

To the rear of the body, as the flow progresses downwards, the pressure becomes greater (adverse pressure gradient). The pressure gradient will continue to increase and become steeper and the fluid will flow against the pressure gradient as far as possible. The fluid, however, does not have enough energy to flow all the

way to the top of this pressure hill and the boundary layer will eventually separate from the surface [16]. At this point, flow can be pushed back near the surface. Refer to Figure 8. At this separation point, the velocity gradient at the surface will become zero. It can be realised that at this point there is a stream line that leaves the body surface, dividing the forward and reverse flow. This is the line of zero velocity and the point where it leaves the body surface is called the point of separation.

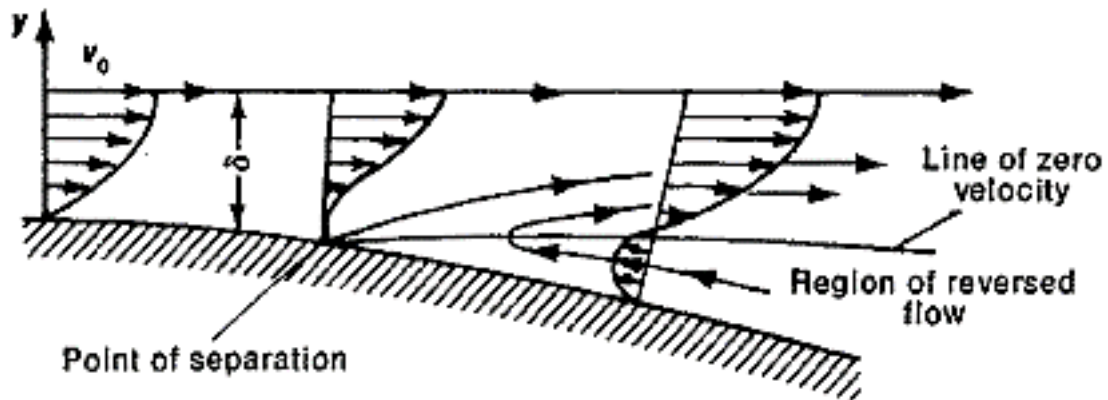


Figure 8 – Schematic of flow separation at a wall [17]

Turbulent flow layers can support much steeper positive pressure gradients without experiencing flow separation. The separation point is moved backwards when the flow is turbulent due to the energizing effect of turbulent flow, where the kinetic energy levels of adjacent layers, such as the fluid layers close to the body wall, is increased [18].

Fully separated flow results in vortices behind the body, consuming extra energy. Separation can also result in air being pushed back, eventually moving faster than the body and also consuming extra energy.

2.3 Transition Modelling with CFD

Simulating flow around bodies using CFD, is gaining popularity in modern day research studies. Reasons for this include the elimination of the use of wind-tunnels, which are expensive to operate; and by using CFD, more iterations of the study can be completed in a smaller time frame (provided the computational power is sufficient). There are many reliable turbulence models, allowing for accurate simulation results to be obtained for turbulence modelling [19]. While simulating turbulence is important, an even more important factor is the transition prediction of laminar flow on bodies.

Several simulation methods are available to compute the transition over a body, each with its own limitations. All of these methods solve the Navier-Stokes equations, but with differences. The first of these is the Direct Numerical Simulation (DNS) method. DNS solves the full unsteady-state Navier-Stokes equations [15]. This method does not require a turbulence model for turbulence closure as no Reynolds averaging is used, but in order to capture the small scales of turbulence, a very fine mesh is required; thus large computational power. Even though great advances in computational technology have been made, it is still not feasible to use the DNS method to compute transitional flow across a 3-D model.

The second method is the Large Eddy Simulation (LES) method. This method works on the same principle as DNS, which is to solve the Navier-Stokes equations, but on a much coarser mesh. This means that only the large scale eddies are directly resolved, while small scale eddies are modelled using an eddy viscosity assumption. The small scale eddies are generally more isotropic; thus simpler, universal models can be used rather than standard Reynolds stress models [20]. The LES method therefore computes the large scale eddies using a fine grid and the small scale eddies are computed using sub-grid models.

Reynolds Averaged Navier-Stokes (RANS) modelling is the third commonly used method to model the transition. This is a useful method from a design perspective, as a reasonable compromise is made between accuracy and expense [21]. This means that the method is more cost effective than previous models but will lower accuracy, though not to such an extent that the results are not reliable.

The most recent model is a correlation-based transition model called the Local Correlation-based Transition Model (LCTM) [19]. This model uses only local variables and gradients, as well as the wall distance [22]. The intermittency function developed for this model is coupled with the SST $k-\omega$ turbulence model, turning on the production term for the turbulent kinetic energy behind the transition location.

One realization of this LCTM concept is the γ - Re_θ transition model.

2.3.1 The γ - Re_θ Transition Model

Langtry described a model for transition prediction (Langtry *et al.* [23], Menter *et al.* [24], Menter *et al.* [25] and Langtry *et al.* [26]) that is based on two transport equations, using only local information. The first of these two transport equations is the intermittency, γ , which can be described as the fraction of time

during which the flow is turbulent, measured at a specific point in the transition region. The momentum thickness Reynolds number is not used to trigger the onset of transition, but rather the strain-rate Reynolds number:

$$Re_v = \frac{\rho y^2}{\mu} \frac{\delta u}{\delta y} = \frac{\rho y^2}{\mu} S \quad 2.5$$

where y is the distance from the wall, ρ is the density, μ is the molecular viscosity and S is the absolute value of the strain-rate. What is of importance with regard to Re_v is the relationship between its maximum value inside the boundary layer and the momentum thickness Reynolds number (Re_θ). As the boundary layer grows, $y^2 S$ will also increase until the critical value for Re_v is reached.

The intermittency transport equation can be summarised as in Eq. 2.6 [23]:

$$\frac{\delta(\rho\gamma)}{\delta t} + \frac{\delta(\rho U_j \gamma)}{\gamma x_j} = P_{\gamma 1} - E_{\gamma 1} + P_{\gamma 2} - E_{\gamma 2} + \frac{\delta}{\delta x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_f} \right) \frac{\delta \gamma}{\delta x_j} \right] \quad 2.6$$

Where the transition sources are defined as:

$$P_{\gamma 1} = F_{length} \rho S [\gamma F_{onset}]^{c_{a1}} \quad 2.7$$

$$E_{\gamma 1} = c_{e1} P_{\gamma 1} \gamma \quad 2.8$$

F_{length} is a correlation used to control the transition area length, while the onset of transition (F_{onset}) is controlled by the strain-rate Reynolds number, Re_v .

This model is usually coupled with the SST $k-\omega$ turbulence model (Shear Stress Transport model). Many correlation-based transition models work on the principle of momentum thickness, which is strictly a 2-D-concept [19]. However, the γ - Re_θ transition model avoids this limitation and is able to compute transition in 3-D flow models.

To couple the transition model with the SST k- ω turbulence model, the following equations are used:

$$\frac{\delta}{\delta t}(\rho k) + \frac{\delta}{\delta x_j}(\rho u_j k) = \bar{P}_k - \bar{D}_k + \frac{\delta}{\delta x_j} \left((\mu + \sigma_k \mu_t) \frac{\delta k}{\delta x_j} \right) \quad 2.9$$

$$\frac{\delta}{\delta t}(\rho \omega) + \frac{\delta}{\delta x_j}(\rho u_j \omega) = \alpha \frac{P_k}{v_t} - D_\omega + C d_\omega + \frac{\delta}{\delta x_j} \left((\mu + \sigma_k \mu_t) \frac{\delta \omega}{\delta x_j} \right) \quad 2.10$$

$$\bar{P}_k = \gamma_{eff} P_k \quad 2.11$$

$$\bar{D}_k = \min(\max(\gamma_{eff}, 0.1), 1.0) D_k \quad 2.12$$

Where P_k and D_k are the original production and destruction terms for the SST model and γ_{eff} is the maximum value between γ and the modified equation for separation induced transition, γ_{sep} .

By coupling the intermittency, γ , with the above-mentioned turbulence model (SST k- ω turbulence model), the production term is turned on. Regarding the modelling of transition, one can capture the effect of the large free-stream turbulence levels on the laminar boundary layer, as well as the related increase in the skin friction and heat transfer. In order to capture the non-local influence of the turbulence intensity, it is necessary to solve for the transition onset momentum-thickness Reynolds number (Re_θ). The turbulence intensity changes due to the decay of the turbulent kinetic energy in the free stream layer, as well as changes to the free stream velocity outside the boundary layer. The transition model therefore solves for the intermittency, γ , as well as the transitional momentum-thickness Reynolds number, Re_θ , and is thus named the γ - Re_θ transition model.

According to Menter *et al.* [19], the intermittency equation has been formulated to take into account the rapid onset of transition during the separation of flow and can also be formulated for low free stream turbulence intensity flows and cross-flow instabilities. These modifications make the transition model extremely compatible with modern CFD codes such as STAR-CCM+.

2.4 CFD as a Design Tool

The external aerodynamic simulation process for simulating the flow around road vehicles, is described by Mansor *et al.* [27] as having four important steps. These four steps are the model geometry setup, mesh

generation, numerical iterations and post processing. Yakkundi *et al.* [28] also indicates that the CAD model is generated using a suitable software package, after which it undergoes surface repair during pre-processing. This relates to Mansor's model geometry setup.

Both Mansor and Yakkundi indicate the next step as meshing of the model or mesh generation. This is the process of dividing the volume of the model into smaller elements in order to be able to solve the flow field around the vehicle. Refinements is incorporated in specific areas around the model where larger elements would not be able to capture the geometry effectively.

A mesh independence study is an important step to conduct. This ensures that the solution obtained during the simulations are independent of the mesh resolution. In other words, reducing the mesh size further, has no effect on the result. If one were to further refine the mesh after mesh independence has been reached, computation time will be increased, and the result will remain within a very small margin of the previous simulation. During this same step the different turbulence models can be compared in order to test the accuracy of the different turbulence models. The most common turbulence models used in CFD simulations are the $k-\epsilon$ and the $k-\omega$ turbulence models. Both Mansor *et al.* [27] and Yakkundi *et al.* [28] found the $k-\omega$ SST turbulence model to be more accurate.

Boundary layer conditions are of utmost importance when simulating flow around a road vehicle. The fluid domain should be defined as air. The boundary in front of the vehicle is defined as the inlet of the domain and should be defined in the CFD simulation as a velocity-inlet with the same velocity as the speed at which the vehicle is being simulated to move forward. The outlet of the domain is set as a pressure-outlet with the pressure at 0 Pa (relative to atmosphere). The inlet domain should be set at least two vehicle lengths ahead of the vehicle to ensure that the flow is fully developed (and laminar) by the time it reaches the vehicle body. The outlet domain is set at least three vehicle lengths behind the vehicle. This ensures that the separation of flow and wake generation is properly captured. The vehicle surface should be treated as a no-slip condition while the walls of the domain can be treated as free slip conditions as well as symmetry walls.

The last step in this basic methodology is to analyse the results of the simulations. The simulations have to be run until a converged solution is achieved, usually by first solving for only turbulence after which the

transition model can be activated. This enables the solution to diverge easier by removing some of the variables to be solved for at the beginning of the simulation. The results are evaluated in the step referred to as post-processing and is usually compared to the experimental results (if available) at this stage.

2.5 Aerodynamic Design

For almost all vehicles, aerodynamics play an important role, whether it is to increase fuel economy as in the case of production vehicles or to increase down force in race vehicles. Solar-powered vehicles often run on as much energy as used by a hairdryer, so in this case reducing aerodynamic drag to the absolute minimum is essential.

When considering the purpose of the vehicle, this need is further enhanced and the essentiality is highlighted. The solar-powered vehicle is built to compete against other solar-powered vehicles. All of these vehicles are only powered by energy obtained from the sun and each team will strive to reduce the energy consumption of their vehicles to a minimum; in most cases by reducing the aerodynamic drag. If this is not done, the vehicle will have a clear disadvantage. By reducing the aerodynamic drag and thereby improving the energy consumption of the vehicle, the vehicle will be able to travel further at higher speeds, and therefore have a better chance at winning the competition.

The following sections will describe how the vehicle design is affected by the aerodynamic forces on the body and how the shape of the body can be altered to reduce these aerodynamic effects. During the design process it is always important to reduce the frontal area as this has a direct effect on the drag force. A person holding his or her hand out a car window when the car is moving at a high speed, will feel a noticeable difference between holding their hand vertical (palms perpendicular to the moving air) than when their hand is rotated to be horizontal to the ground. This movement reduces the frontal area and in effect the aerodynamic drag force.

2.5.1 Vehicle Body Shape

The 3-D shape with the least amount of aerodynamic drag is that of a streamlined body [16], such as a water droplet, but due to the requirement of a large flat surface to accommodate the solar array, this droplet shape

is not suited for the body shape of a solar-powered vehicle. So the droplet shape is modified to have a flat tail section, to accommodate the solar array.

In the case of a droplet shape, the drag is at a minimum when in free stream air [29]. Bodies travelling in close proximity to the ground, such as solar-powered vehicles, experience a venturi effect as the air flows between the body and the ground plane [5].

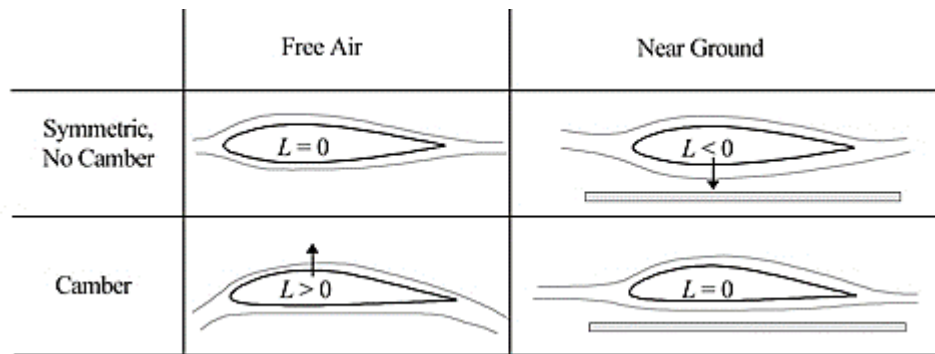


Figure 9 - Summary of how camber effects the lift force on the body in freestream and in close ground proximity [5]

The ground effect increases the downforce on the droplet shape as the stream lines around the shape become compressed between the body and the ground as is illustrated in Figure 9. The streamlines concentrate more on the underside of the body than on the top, and this venturi effect produces downforce. Cambering the shape changes the flow patterns around the body, reducing the venturi effect and therefore the downforce on the body.

When the cambered body is in freestream, the streamlines experience greater acceleration over the top of the body. This increase in velocity reduces the pressure on top of the body, resulting in lift. When a cambered body is in close proximity to the ground, this lift can be eliminated, increasing the stability of the vehicle.

Another method to reduce the drag is to change the top view of the body shape. In the case of a rectangular shape, the solar array would fit easier onto a body of constant width, making it more desirable [29]. Even though a solar array would fit onto the rectangular shape with more ease, a rectangular body shape is not the most aerodynamic. However, the drag can be reduced by tapering the trailing edge of the body. This

complicates the solar array design as the curvature of the body makes it difficult to fit a rectangular array without wasting area on top of the body.

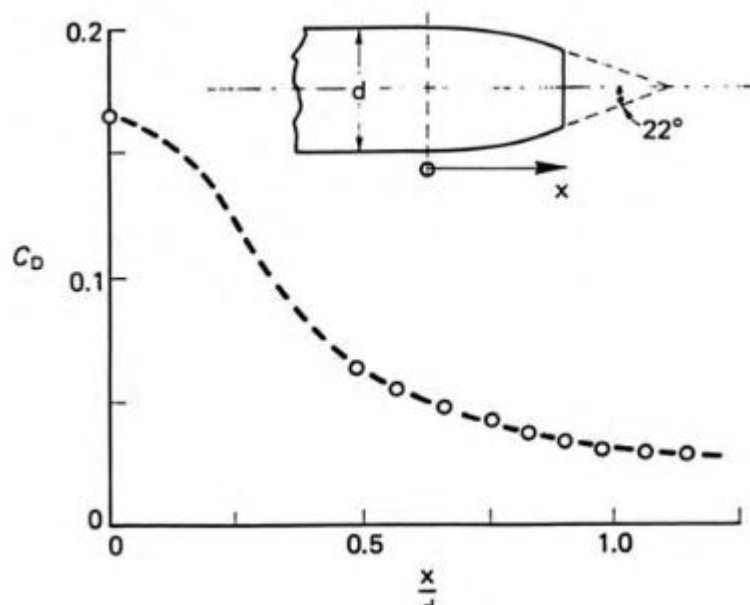


Figure 10 - Tapering of the trail edge of a bluff body (boat-tailing) [30]

Referring to Figure 10, the effect of tapering the trailing edge of a bluff body can be observed. This is commonly called “boat-tailing” [30]. Even though this is based on drag reduction of bluff bodies, the theory can be applied to the shape of the solar vehicle body as well. By tapering the body at 22° , the drag can be reduced by more than 10%. In the case of solar vehicle bodies, it is not always possible to taper the vehicle to that extent, as the competition regulations limit the length of the vehicle and space is still required to fit 6 m^2 of solar cells.

The static stability of the main body is also important, as it will affect the dynamic stability of the entire vehicle. According to McCormick [31] static stability refers to the tendency of an airplane to return to a trimmed condition when disturbed under steady conditions. It does not refer to any motion it undergoes after the initial disturbance. This theory may also be applied to the main body of the vehicle as it is essentially a wing of an airplane. For the vehicle to be in equilibrium, the forces and moments acting on the vehicle must be in a direction to constantly force the vehicle back to its equilibrium position. In other words, the forces acting on the body must balance around the centre of gravity of the vehicle. This point where the forces are in balance, is referred to as the trim point.

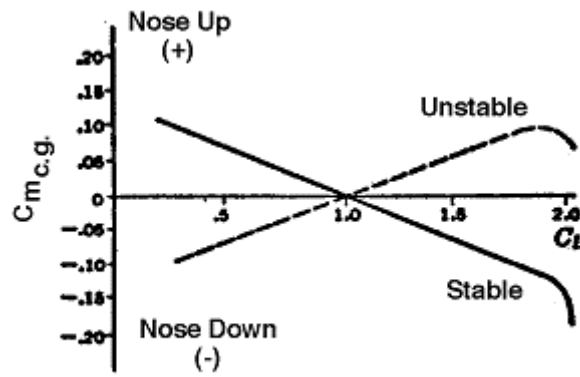


Figure 11 - Body at trim [32]

Referring to Figure 11 [32], the body will be in balance or trim when the moment about the centre of gravity is zero. This is where the lift coefficient is equal to 1 and the angle of attack is 0° . The lift coefficient must be positive at 0 angle of attack in order to generate lift (in the case of an airplane). If the angle of attack (α) is increased, in other words, an increase in the lift coefficient (C_L), and the moment increases about the centre of gravity, the body will become unstable. The opposite will happen if the moment decreases with increasing C_L . Therefore, for the body to be stable, two conditions must be met:

$$\frac{dM}{d\alpha} < 0 \quad 2.13$$

This means that the moment about the centre of gravity of the body must be negative. The other condition that has to be met for vehicle stability is that the centre of gravity must be ahead of the centre of lift. See Figure 12 [33].

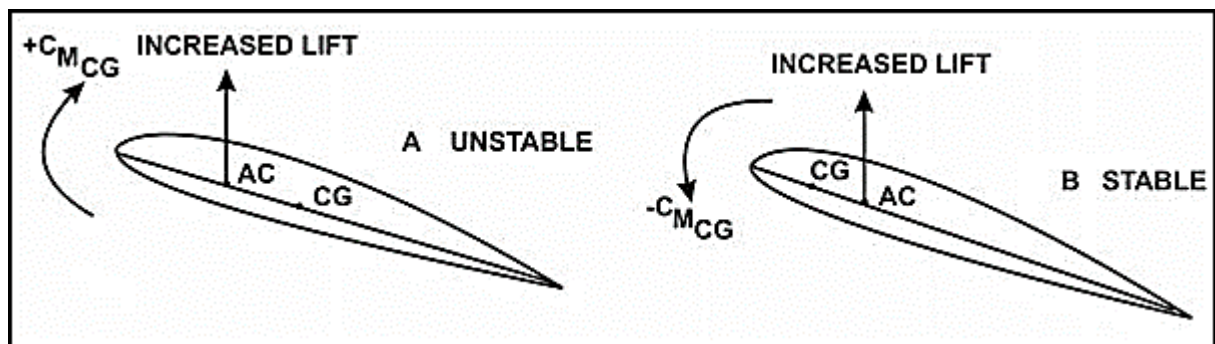


Figure 12 - Airfoil contribution to longitudinal stability [33]

If the centre of gravity is behind the centre of lift, the body will experience a positive moment. When the lift force increases with increased α , the moment about the centre of gravity will increase linearly and will

cause the body to become unstable. What eventually will happen is the front wheels of the vehicle will lose traction on the road surface, which can result in an accident. If the centre of gravity is in front of the centre of lift, the body will have a negative moment. This will have an effect of pushing the nose of the body down in the case of increased α . The vehicle's front wheels will thus remain in contact with the road surface and thus stable on the road. This can also be defined in terms of the neutral point:

$$C_{M_\alpha} = (h - h_n)C_{L_\alpha} \quad 2.14$$

This equation states that the slope of the momentum curve, C_{M_α} , is equal to the slope of the lift curve, C_{L_α} , multiplied by the dimensionless distance of the centre of gravity behind the neutral point of the body, $(h - h_n)$. However, for the body to be statically stable the momentum must be negative, meaning the centre of gravity must be ahead of the neutral point. The value $(h - h_n)$ is the static margin and must be at least 5% of the mean aerodynamic chord of the body [31].

2.5.2 Nose design

An effective way to delay the transition of flow from laminar to turbulent is to design the nose of the body to have a favourable pressure gradient. Imagine a fluid molecule of mass m flowing over the nose of the vehicle, which is usually convex in shape. This will result in the fluid molecule accelerating and this resultant increase in velocity will result in a reduction in pressure [5].

This molecule experiences a centrifugal force (mV^2/R), pushing it away from the surface of the body. V is the local streamline velocity and R is the local radius of curvature of the body. In order for the molecule to remain attached to the body, the local pressure gradient must be such as to push the molecule against the body. In other words, the centrifugal force acting on the molecule must be balanced by the pressure gradient [34]. This means that the free stream air pressure must be greater than the local pressure at the surface of the body. By increasing the centrifugal force, by either increasing the free stream velocity or by decreasing the local radius of curvature, the local pressure will decrease. The resulting effect will be that the free stream pressure will be greater than the local pressure and the molecule will remain attached to the body; thus the flow would remain attached. This attached flow could be laminar or turbulent depending on whether transition has been reached.

A negative pressure gradient results during acceleration, as the pressure at one point is lower than the point directly before it. This negative pressure gradient has the effect of stabilizing the laminar boundary layer, and is thus favourable.

The pressure coefficient (or coefficient of pressure), C_p can be defined by Eq. 2. 15:

$$C_p = \frac{P_{loc} - P_{\infty}}{\frac{1}{2}\rho V_{\infty}^2} \quad 2.15$$

According to Bernoulli's Principle, C_p can also be expressed by Eq.2. 16:

$$C_p = 1 - \left(\frac{V_{loc}}{V_{\infty}}\right)^2 \quad 2.16$$

Here P_{loc} and V_{loc} are, respectively, the local pressure and velocity values near the body surface. Looking at Eq. 2. 15 it is clear that the value for C_p is a measure of the local pressure. It is assumed that the pressure across the thickness of the boundary layer is constant, as the boundary layer is very thin. The pressure is equal to the pressure just outside of the boundary layer, P_{∞} of Eq. 2. 15. Therefore, if the value for C_p is known, the rise and fall of the surface pressures can be calculated. By tracking the slope of the C_p curve along the body, the pressure gradient curve can be determined.

Eq. 2. 16 indicates that the value for C_p must be equal to 1 when V_{loc} is equal to zero. This point is termed the stagnation point. The air accelerates over the nose of the body and this causes a rapid reduction in the pressure at the same point. Nearing the thickest part of the body, the flow will continue to accelerate, but at a slower rate. It results in further reductions in the pressure and the C_p value becomes negative. At the point of maximum thickness and minimum pressure, the pressure will start to increase again. This is called the pressure recovery region, where the C_p value progresses in the positive direction [5]. It is usually in the pressure recovery region that the flow separates from the body.

2.5.3 Interference Drag Reduction

As described in Section 2.1, interference drag is the result of irregularities on the surface body such as manufacturing defects, ventilation scoops, poorly fitted body panels, wheel arches, etc. This section describes design procedures to reduce the effect of interference drag.

Junction Flows

Junction flows refer to the junctions between the main body of the vehicle and appendages such as the wheel fairings and the canopy for the driver. The drag increase ΔD_j due to these junctions [5] can be approximated by the following equation:

$$\Delta D_j = qt^2 \left[0.75 \left(\frac{t}{c} \right) - 0.0003 \left(\frac{c}{t} \right)^2 \right] \quad 2.17$$

Where q is the dynamic pressure $0.5\rho V^2$, c is the chord length and t is the thickness of the airfoil-shaped appendage. Referring to Figure 13 [35], the chord length and the thickness of the appendage, relating to the above equation, is clearly shown.

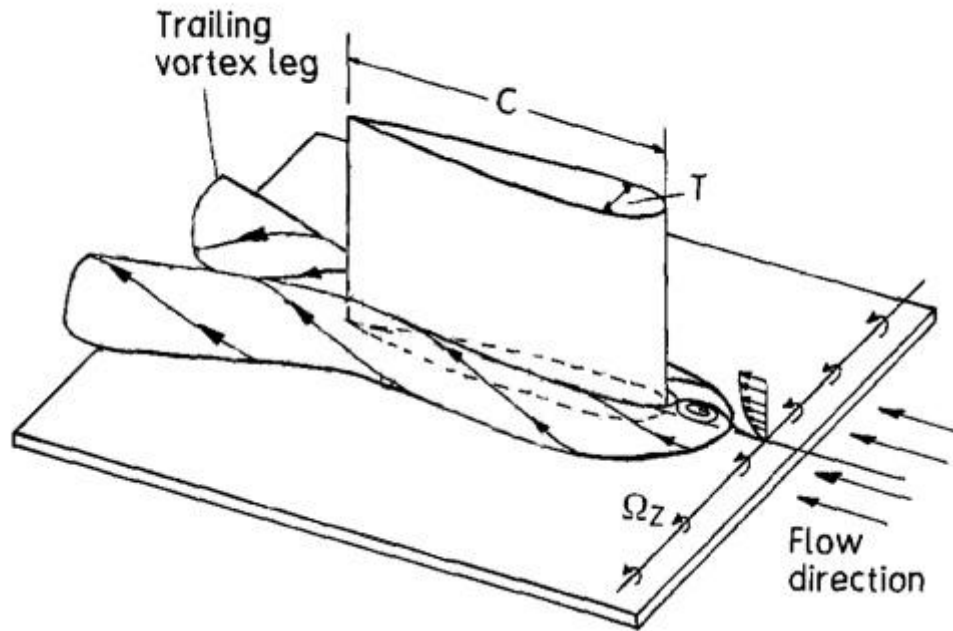


Figure 13 - Junction flows between the fairing and the body [35]

The junction flow is described by Boermans *et al.* in [36]. The flow ahead of the appendage is laminar, but this laminar boundary layer cannot overcome the adverse pressure gradient created by the junction between the body of the vehicle and the appendage, or the body-appendage junction. This adverse pressure gradient causes the laminar boundary layer to transition. The turbulent boundary layer is, in turn, forced to separate from the surface of the body due to this adverse pressure gradient. The adverse pressure gradient is formed due to the junction and the flow will roll up into a vortex at the root of the appendage, as is seen in Figure

13. These vortices are of the rotational sense of the approaching boundary layer and because of this, the vortices entrain higher speed fluid along the appendage which increases the drag in the junction region [37]. The simplest method to reduce the drag caused by junctions is to minimize the number of junctions. By aligning the fairings with the outside lateral edge of the main body, the number of junctions can be reduced, as well as the interference drag. Once these junctions have been reduced or eliminated, it is necessary to minimize the drag of the body. The key here is to eliminate flow separation.

Inclination Angle of the Wheel Fairings

According to Tamai [5] the inclination angle (θ_f), as indicated in Figure 14 [5], of the wheel fairing to the vertical plane should be as small as possible. Therefore, the leading-edge angle should rather be near vertical, as a fairing that is near perpendicular to the body has a C_d (drag coefficient) value of about 0.08, provided the thickness-to-chord ratio is about 0.43.

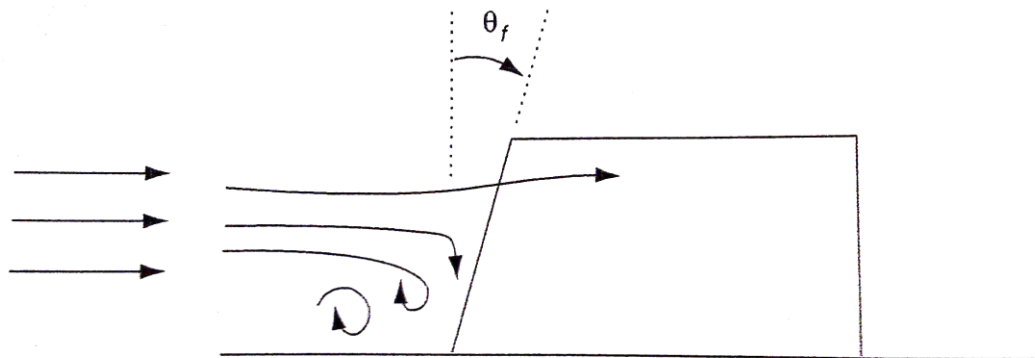


Figure 14 - Inclination angle of an appendage [5]

The C_d value increases minimally for inclination angles up to 10° . For every degree that the inclination angle exceeds the 10° mark, the drag increases linearly by about 2.5% per degree of inclination. This can be attributed to premature separation of flow, which is what should be avoided. This means that when designing the fairings, it should be done so that the fairings intersect the body perpendicularly. Simpson, however states that transition can be delayed by increasing the inclination angle of the fairing as it shifts the vortex downstream [37]. This is based on experiments conducted by Ahmed and Khan [38]. By

designing a fairing that joins the body perpendicular and sweeps back at an angle of less than 10° , it might be possible to reduce the effect of the vortex.

Fillet Radius along the Appendage

Fillets can help to reduce the interference between the body and the appendage [39]. This is achieved because the fillet eliminates or minimizes the effect of vortices that forms at a sharp intersection between an appendage and the body of the vehicle. By implementing fillets where the appendage meets the body, as indicated in Figure 15 [5], a drag reduction of as much as 20% can be experienced. The optimum fillet radius, as indicated by the above-mentioned author, is 4-6% of the chord length of the appendage. This value was originally experimentally determined by Hoerner [40].

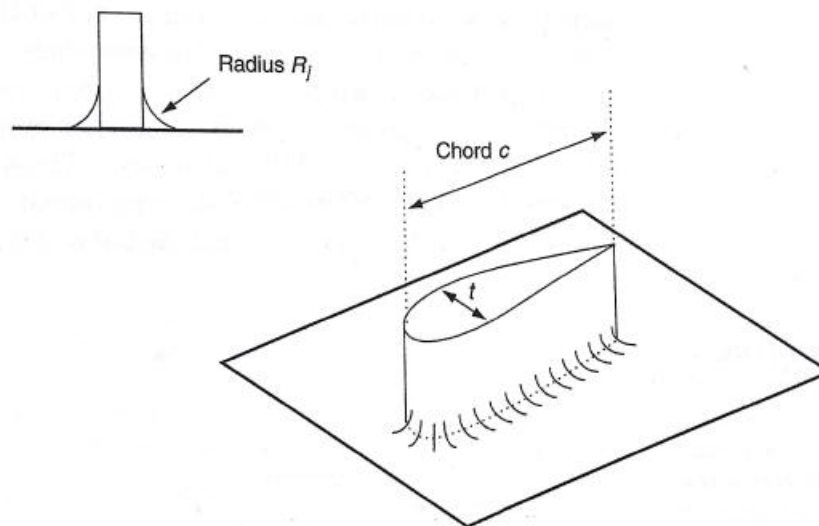


Figure 15 - Fillet radius along an appendage [5]

It was also determined that a smaller fillet radius proved to be more favourable. This was experimentally determined by Maughmer *et al.* [41], when wind tunnel tests revealed that a 10% radius on a sailplane had a measurable reduction in the drag force compared to sailplanes with 35% and 45% radii.

Leading-edge Junction Geometry

If the leading-edge of the fairing is blunt, in other words, with no fillet (Figure 13 [35]), creating a sharp transition from the fairing to the body, the adverse pressure gradient ahead of the fairing will cause the flow to separate and form a vortex. This vortex, called a horseshoe vortex, will continue along the entire length

of the fairing. This will increase the shear stress along the fairing junction. The increase in pressure drag beyond the trailing edge of the fairing is usually what increases the drag of the vehicle.

By implementing a leading-edge fillet of 63° , the separation of the flow can be eliminated. Using a fillet with a 30° angle, separation can be greatly reduced [5]. Smoothing these leading-edge fillets further helps to eliminate separation. Another method to reduce the drag, using a leading-edge fillet, is to implement a linear fillet. The linear fillet showed as much as a 7% drag reduction over a parabolic fillet, according to Maughmer *et al.* [41].

2.5.4 Canopy Drag Reduction

Bubble canopies are the most popular design and will be implemented in the design. These bubble canopies should allow the driver sufficient space to move forward with clear peripheral visibility. The canopy should also not be too large as this reduces the area available to the solar array. The points discussed in the previous subsection should also be applied to the junction between the canopy and the body.

Moving the canopy as far backwards as possible is also essential to prevent premature flow separation. Viewing the canopy from the front, it should be as narrow as possible in order to create a narrower separation bubble around the canopy, thus more laminar flow on the body.

A major design variable for canopies is the length-to-height ratio, L/h . According to Tamai [5], it was reported that certain university solar car teams claimed that their vehicles had canopies with little or no separation. These canopies had L/h ratios of about 4.5, with the highest point of the canopy at about the middle of the canopy. However, the author also states that a safe optimum L/h ratio for a canopy would be 6. This is based on data in Figure 16.

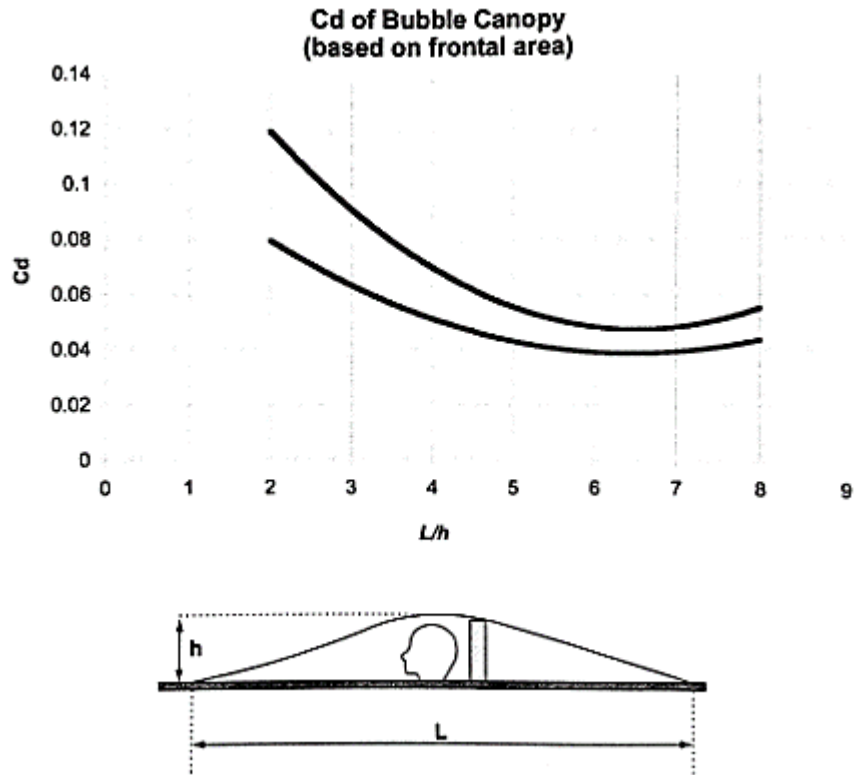


Figure 16 - The upper and lower bounds of the canopy drag as a function of the L/h ratio [5]

The drag values of canopies also change as the shape of the canopy is changed. For the above L/h ratio of 6 and a trailing section that tapers into a point, the approximated drag value for the canopy is 0.045 (based on the frontal area of the canopy) [5]. It is quite clear by referring to Figure 17 [5] that designing a canopy to have any type of blunt end will increase the drag.

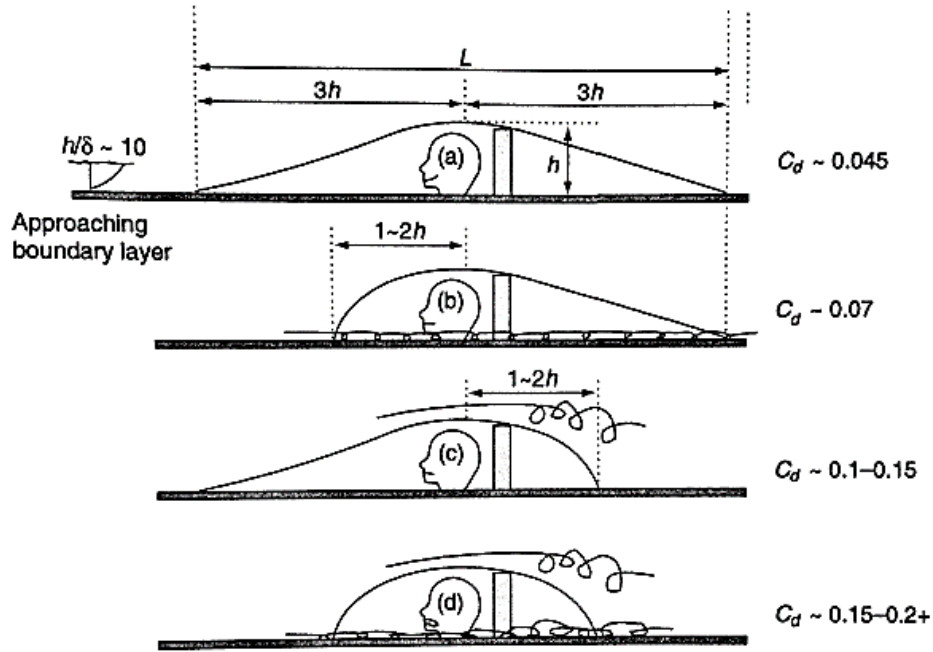


Figure 17 - (a) Illustration of baseline canopy with $L/h = 6$. (b) Base line canopy with a blunt nose. (c) Baseline canopy with a blunt tail. (d) Baseline canopy with a blunt nose and tail [5]

The L/h ratio method is an accurate design methodology to be used for the canopy design. But it should also be noted that to allow sufficient space for the solar arrays while still allowing enough space for the driver and roll bar, the L/h ratio of the canopy might be much closer to a value of 3 to 4. The shape of the canopy in terms of leading and trailing edges should also be kept in consideration.

2.5.5 Trailing Edge Thickness of the Main Body

In many 2-D, as well as 3-D simulations, it is often assumed that the trailing edge of an airfoil is perfectly sharp. However, this is not the case with solar vehicle bodies. The ideal would be to design for a knife edge trailing edge, but this is not always possible.

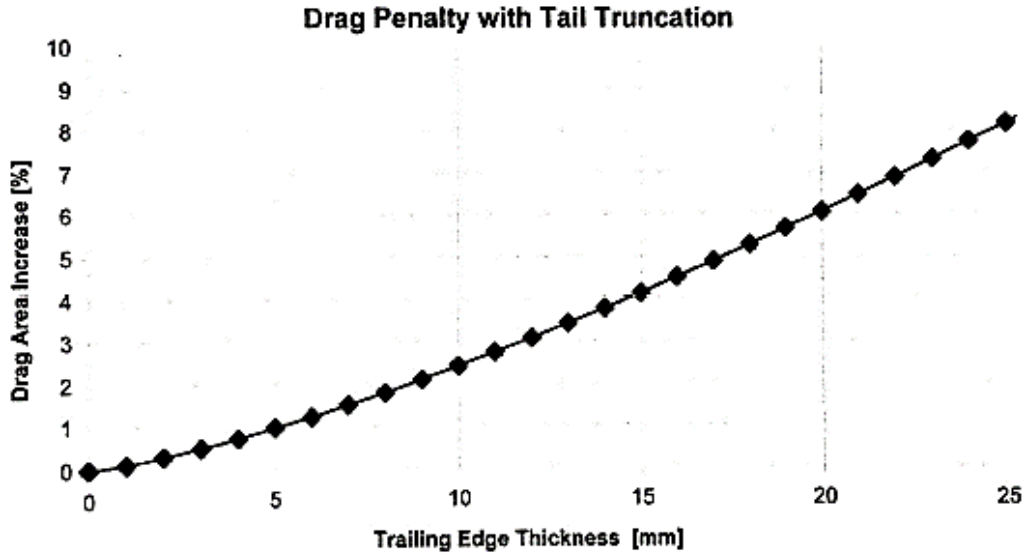


Figure 18 - Trailing edge thickness plotted against the percentage increase in drag area [5]

This drag increase caused by the trailing edge, is estimated by the equation described by Hoerner [39] and is based on the frontal area of the main body:

$$\Delta C_d = \frac{0.34}{\sqrt[3]{C_{d,frontal}}} \left(\frac{t_{te}}{2t} \right)^{\frac{4}{3}} \quad 2.18$$

where $C_{d,frontal}$ is the drag coefficient of the main body, based on its frontal area, t_{te} is the thickness of the trailing edge and t is the maximum thickness of the main body of the vehicle.

Referring to Figure 18, it is clear that a 5 mm thick trailing edge causes an increase of 1% in the drag value. An increase of 1% is regarded as significant and therefore it is best if the trailing edge does not exceed 5 mm. It would be ideal if the trailing edge of the vehicle was a knife edge, but practicality should also be considered as the vehicle has to have rear taillights, as per regulation.

2.6 Popular Solar-powered Vehicles

Over the years certain institutions have gained extensive experience in developing and manufacturing solar-powered vehicles. These institutions have competed in various competitions, including the World Solar Challenge held in Australia every second year [42]. The following teams are placed respectively first, second and third globally, based on performances at the 2013 World Solar Challenge.

Delft University of Technology: Nuna7 (Nuon Solar Car Team)

The team's car line-up is called Nuna. They have already produced six vehicles, having started in 2001. The Nuna6 (displayed in Figure 19 [43]), which competed in the 2011 World Solar Challenge, achieved an overall second position. The body of this vehicle is a sandwich construction, with face material being carbon fibre and a foam core. "TeXtreme" Carbon fibre fabric was used, which is also used in the construction of Formula 1 cars. The roll bar used was made of titanium and aramid reinforced parts were used to protect the driver during possible collisions. The vehicle had an aerodynamic drag coefficient of 10% less than the previous model, the Nuna5.



Figure 19 - Nuna6 solar vehicle [43]

The Nuna7 (Figure 20 [44]), which is the university's first four-wheel vehicle, was built to compete in the World Solar Challenge in Australia in 2013 (Figure 20 [44]). Four wheels were required according to the competition regulations; a four wheel vehicle being more stable than the three wheel designs. This vehicle uses the same body shape as the Nuna6, but to cover the wheels two long fairings were used. The two fairings are replicas of each other, wide enough for the driver to be seated between the front and rear wheels on the left hand side of the vehicle. The space on the opposite side to the driver is used to install various components such as the battery case, thus ensuring even weight distribution across the vehicle.

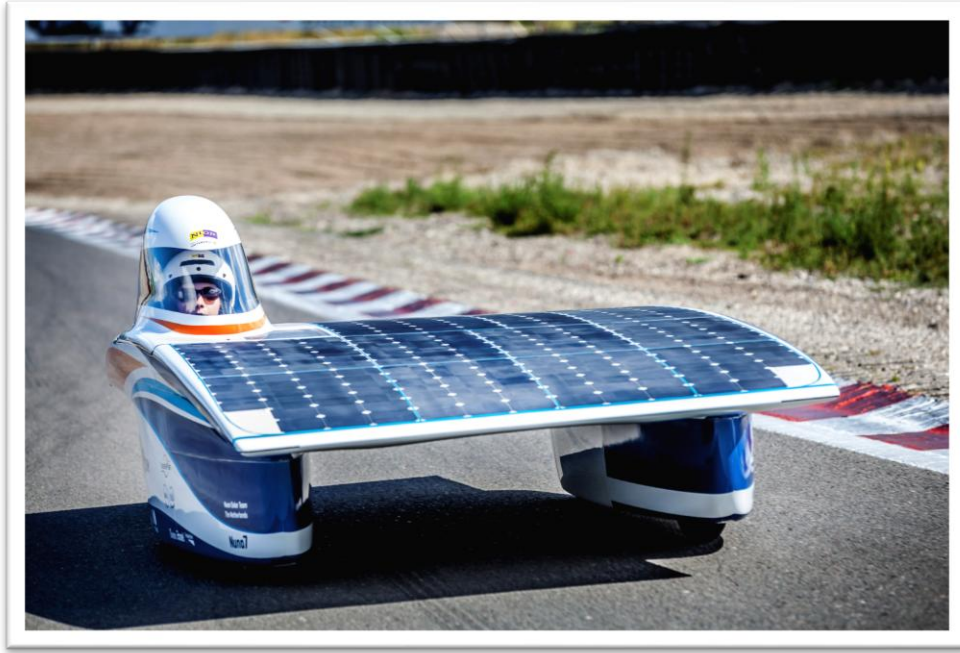


Figure 20 - Nuna7 solar vehicle [44]

The Nuon Solar Team is currently ranked first in the world for the Challenger Class of solar-powered vehicles after they finished first during the World Solar Challenge in 2013.

Tokai University: Toray

As with the Nuna range of Nuon Solar Car Team, Tokai University has always focused their designs on three-wheel concepts (see Figure 21 [45]). Tokai University, from Japan, has always been the main competition for the Nuon Solar Car Team followed by Solar Team Twente. As mentioned, the Challenger Class now regulates that vehicles must have four wheels, and Tokai University also built their first four-wheel vehicle for the World Solar Challenge in 2013.



Figure 21 - Tokai University solar-powered vehicle [45]

This vehicle is different from the Nuna7 vehicle in that it has only one long fairing, also serving as the driver cockpit. The remaining two wheels on the opposite side of the vehicle are covered by individual fairings. Once again the body shape is that of a wing airfoil, thus enhancing the laminar flow on the body of the vehicle.

Unlike the Nuon Solar Team's solar-powered vehicle, the different components of this vehicle are mounted in the main body, distributed evenly so that the centre of gravity is closer to the front of the vehicle. This enhances the stability of the vehicle.

Solar Team Twente: The Red Engine

Another popular solar-powered vehicle, dubbed the "Red Engine", was built by Solar Team Twente from the Netherlands (see Figure 22 [42]). This vehicle has a different concept to that of the other teams already discussed. The basic body shape is also that of an airfoil, but it is lofted towards the centre of the vehicle. This more rounded shape allows the driver of the vehicle to be seated in the centre of the vehicle.



Figure 22 - Solar Team Twente's "The Red Engine" solar-powered vehicle [42]

This vehicle further has a single shorter fairing covering each of the four wheels. The two rear wheels' track width is narrower than the front wheels. All the components are installed in the main body, again so that the centre of gravity is towards the front end of the vehicle.

One can compare the effectiveness of these designs with one another. This enables the most promising design features to be identified, which can then be used to enhance the aerodynamic design of the NWU's solar-powered vehicle. In section 2.4 it is indicated that the frontal area of the vehicle should be reduced as it has a direct effect on the aerodynamic drag of the vehicle. Keeping this in mind while evaluating the design of the three vehicles discussed, it makes sense to only look at the Nuna7 and the Toray. This is because both these vehicles have the driver seated on the side of the vehicle. This greatly reduces the frontal area of the vehicle and can therefore be regarded as important design feature. The fact that the Toray also has two shorter fairings on the one side, means that the overall width of these fairings are narrower than the single fairing of the Nuna7. This adds to the reduction of the frontal area. Furthermore, the friction drag between the air and the surface of the vehicle also contributes to the drag of the vehicle. The two shorter fairings covering the wheels opposite to the driver's side of the Toray means that the vehicle has a smaller wetted surface (body surface in contact with the air). Theoretically this should help to further reduce the drag of the vehicle. The Red Engine has a larger wetted surface than any of the other two vehicles discussed

and has a much larger frontal area. Theoretically this means the aerodynamic drag of this vehicle would be higher. All of the vehicles have one thing in common, namely smooth junctions between the different body appendages. This is achieved by fillets to reduce the vortex effects that the body has on the air flow past it and to extend the laminar flow.

The most promising features that can be carried forward to the design process is to move the driver to the side for an asymmetric design and to reduce the width of all fairings to the absolute minimum that is required for the proper functioning of the internal components. Effort should be made to reduce the wetted surface of the vehicle in order to further reduce the aerodynamic drag of the vehicle. Properly sized fillets should also be incorporated into the design of the junctions.

2.7 Summary

Aerodynamically, solar-powered vehicles can be improved by reducing or even eliminating the drag components contributing to the drag force on the vehicle. These include the pressure drag, friction drag, induced drag and interference drag. These components are directly affected by the shape of the vehicle and the different design factors discussed. It is important to choose the values for these factors, such as the body shape, nose and trailing edge design as well as canopy and fairing designs, as near the theoretical optimum as possible. The most promising factors of previous designs is to move the driver to the side of the vehicle, an asymmetric design, and to reduce the width of the fairings. This reduces the frontal area of the vehicle and should therefore help to reduce the aerodynamic drag of the vehicle. One should further try to reduce the wetted surface of the vehicle which can also contribute to the drag reduction. These changes are incorporated into a methodology which describes four basic steps, namely model geometry setup, mesh generation, numerical iterations and post-processing.

Chapter 3: Methodology

By researching previous methods and studies of a similar nature, a preliminary design of the vehicle can be developed that will be closer to the final design, thereby reducing the design time considerably. The most important aspects of this methodology will be covered in this section, these same aspects has already been summarised by Carroll [29]. They are directly applicable to the design process of a solar-powered vehicle. The methodology proposed in this section focuses specifically on the aerodynamics and the body design of the vehicle.

The methodology is a generic method described by Carroll [29] and is based on published methods such as those described by authors Abe and Starr [46] and Ullman [47]. The following six steps describe the design process that will be followed when designing the body shape of the solar-powered vehicle. It is adapted from the mentioned authors' works and can relate to the process described by Kissinger [48]:

1. Review of competition regulations pertaining to the body design of the vehicle and of previous works. Perform software validation.
2. Determine the initial design parameters to be used to optimise the body of the vehicle, namely the frontal area and the drag coefficient.
3. Design of preliminary vehicle body shapes (concept designs) based on solar-powered vehicle dynamics as discussed by Carroll [29] and Tamai [5]. Here the model geometry is transferred to a suitable CAD software package.
4. Analyse concept designs in terms of the drag coefficient as well as manufacturability. Select a concept design. This step can be conducted using CFD simulations, implicating the mesh generation and numerical iteration steps discussed in the literature survey would have to be applied.
5. CFD analysis (based on the steps mentioned above) of the preliminary body design of the vehicle will be conducted and models are compared during the post-processing stage. The comparisons are based on C_D and transition locations on the turbulent kinetic energy plots.
6. Revise the design by adjusting factors as described by authors mentioned in step 3 and results found in step 5. This is also part of the post-processing step as described in the literature.

3.1 Review Regulations and Evaluate Existing Solutions

The first step is to review all of the competition regulations that will have a direct effect on the body design of the vehicle. This means studying the competition regulations to ensure that the body design will conform to all regulations, making it eligible to compete in the competition. Therefore, a good understanding should be developed in order to have a good idea of what is regarded as being competitive. Existing solar vehicle designs should also be studied in order to determine reliable designs.

3.1.1 Summary of previous work

The most common design for solar-powered vehicles is a vehicle with only three wheels – commonly with two steering wheels in the front and only one driving hub motor in the rear. The most successful teams are the Nuon Solar Team from the Netherlands, Tokai University from Japan and Solar Team Twente also from the Netherlands.

These teams have many years' experience in building solar-powered vehicles, resulting in vehicles that have coefficients of drag values claimed to be below 0.1. The drag resistance caused by the vehicle's body is extremely low and thus little energy is required to overcome the resistance. More solar energy received from the sun can therefore be used to overcome the rolling resistance (also a large factor to be considered) and more power is available to maintain a higher average speed for a longer distance.

As indicated, the common design for a SV is based on three wheels. New regulations state that SVs should have four wheels in order to compete in the main classes. The reason for this change is that the three wheel vehicles are too unstable at high speeds. The organisations (Sasol Solar Challenge and World Solar Challenge) also wish to encourage the technology to advance towards production line vehicles and not to only be focused on designing newer and better race vehicles. The design for three wheel vehicles has been optimised, so the current challenge is to design a four-wheel vehicle with the same drag coefficient figures as the current three wheel designs.

3.2 Geometric Specifications and Initial Design Parameters

This design process is applicable to the body shape of the solar-powered vehicle, but other specifications of the vehicle have a direct influence on the eventual shape of the vehicle. These specifications include the

space required for the solar array, the battery capacity and the shape of the battery packs. The type of motors and suspension to be used will determine the width of the fairings covering the wheels and the shape of the canopy is determined by the space requirements for the driver.

3.2.1 Solar Array and Body Shape

The maximum width of the vehicle should not exceed 1.8 m and the vehicle may not exceed 4.5 m in length. The maximum height of the vehicle is 1.9 m. This leaves relatively limited space for the solar array; with a maximum allowable area of 6 m² (silicone mono-crystalline solar cells). The largest part of the body should therefore be relatively flat or gently curved (such as seen in an airfoil of a wing) surface. This will form the base onto which the solar cells are mounted.

3.2.2 Battery Pack

The size of the battery pack is limited according to weight and the maximum weight is 21 kg. The battery pack is usually mounted in the nose of the vehicle in order to enhance the aerodynamic stability of the vehicle. The battery pack is therefore limited by height, rather being longer and wider. If the battery pack is mounted to the centre of the vehicle, the centre of gravity of the vehicle is shifted to the rear, which can decrease the stability of the vehicle. If the battery pack is too high, it might require an increase in the height of the main body section, increasing the frontal area of the vehicle.

3.2.3 Electric Motors and Wheel Size

Two popular drivetrains are used. The simplest drivetrain used is an electric motor mounted, in the vehicle in such a position to give power to driving the wheels via a chain-link or a similar device. The other type of motor is an in-hub electric motor. The electric motor is mounted inside the rim of the driving wheels. These motors are usually at least 150 mm in width, but the more expensive models can be as narrow as 75 – 95 mm. The wider the motor, the wider the wheel fairings will have to be.

3.2.4 Suspension, Steering Mechanism and Wheel Size

Adding to the electric motor thickness, is the suspension. Numerous suspension designs can be incorporated, each design with its own share of advantages and disadvantages. A commonly used double wishbone suspension might require a fairing to be much wider and therefore will result in an increase in

the frontal area of the vehicle. The larger the wheel, the more space will be required at the same steering angle as a smaller wheel. This, again, increases the frontal area due to wider fairings. These factors should be given precedence when designing the body shape of the initial design.

3.2.5 Driver Position

The driver should be seated comfortably and have ease of access to all controls at any time. The driver should be able to move forward 45° and 25° backwards, without his/her head touching the canopy shield. This might result in a longer canopy being designed, which increases the wetted surface area of the vehicle. He/she should also have a clear field of vision around him/her and his/her eyes should be no less than 700 mm above ground level. Furthermore, the vehicle must have 4 wheels and a ground clearance of no less than 90 mm.

The initial design parameters for the frontal area and the drag coefficient were determined by reviewing existing designs of solar-powered vehicles. This was done in order to design a vehicle that would be on par with world leaders in the trade and to ensure that the vehicle would be able to compete with these other solar-powered vehicles. A value for the frontal area of the vehicle of not more than 1 m² was chosen, as well as a value of 0.1 for the drag coefficient.

3.3 Create Concept Designs

As has been mentioned, previous solar-powered vehicles of other institutions needed to be studied. This will give insight in the most popular designs and different aerodynamic designs that have been proven in a similar competitions to the Sasol Solar Challenge. These should be used as a baseline from which to design different concepts for the vehicle body shape, while constantly keeping in mind the geometric specification mentioned in Section 3.2 of this chapter.

A number of different conceptual designs were evaluated and compared with each other in order to find the design with the most favourable aerodynamic properties. The main areas focused on during the conceptual designs, were the fairing configurations and the position of the driver, while still giving precedence to the reduction of the frontal area and minimizing the effects of the interference and profile drag.

3.4 Selection of Best Concept Design

The drag force on the bodies of the different concept ideas, will be determined according to the simulation model developed during the validation procedure. The results of these simulations should then be presented to the whole team. During this step, the designer should point out the advantages as well as the disadvantages of the different concept designs. The team will then have to discuss the pros and cons of each of the designs and a consensus should be reached as to which design is best in terms of aerodynamics, manufacturability and overall appearance.

3.5 Detailed Design

This step can only proceed once the concept design has been chosen. This is an iterative process and the different aspects of the body will have to be adjusted in order to find the optimum shape with the best aerodynamic properties. The development of the detail design is discussed in Chapter 7 of this study, once the simulation procedure has been validated in the following chapters.

A preliminary body design for the solar-powered vehicle is developed from the selected concept design. This design has to conform to the regulations discussed and the chosen design parameters. By referring to the literature of Carroll [29] and Tamai [5], CAD parts of the different components of the vehicle, such as the fairings and canopy, can be made. The main body section, onto which the solar array is integrated, is chosen as the basis of the design. Around this part, the other components will be designed and integrated according to the above-mentioned literature.

During this process the minimisation of the frontal area of the vehicle should constantly be given precedence, while conforming to the regulations mentioned. Factors that will increase the frontal area of the vehicle include the width and height of the canopy in order to allow enough headroom for the driver; and space for a roll bar to protect the driver in the event of a collision. The front wheels are used to steer the vehicle and there should thus be enough space inside the fairings to allow the wheels to turn without the suspension and brake mechanism touching and possibly damaging the inner walls of the fairings. The body must also be smoothed and sharp corners must be avoided to reduce the effects of interference drag.

These sharp corners can cause vortices to form which will increase the pressure drag of the vehicle.

Therefore, all edges are smoothed by the use of fillets.

Chapter 4: 2-D Validation: NACA 0018

Validation is the process of determining the degree to which a model is an accurate representation of the real world [49]. This is one of the most commonly used definitions for validation and can be broken down into several parts. Validation is a process and can thus not be conducted in only one step. The process also requires that the accuracy should be determined, meaning that error and uncertainty will be factors [50].

When a model has been validated, it is implicated that the validation process is reliable and can be applied to similar models under the same conditions. It can thus be used on other models to make accurate assumptions of reality. It also gives assurance that the model has been tested against well-defined criteria, such as experimental results obtained from wind tunnels, for example.

The 2-D validation process was therefore conducted in order to gain confidence in whether the transition model would iterate correctly at low Reynolds numbers. This then meant that the transition model could be applied in more complicated models, such as 3-D models. The transition model used in the validation of flow around a 2-D airfoil was the γ - Re_θ transition model, coupled with the $k-\omega$ SST turbulence model. CD-adapco's STAR-CCM+ was used as the solver.

4.1 Validation Setup

The geometry used was the NACA 0018 airfoil profile. The coordinates for this airfoil were obtained from Profili 2.0 software [51]. This airfoil was used because of the availability of wind tunnel experimental results as described by Timmer [52].

The coordinates of the airfoil were used to create a solid part in SolidWorks. A solid block, representing the wind tunnel, was also created here, after which the airfoil volume was subtracted from the wind tunnel volume. The fluid volume in the wind tunnel was thus represented by a solid part. The test domain for the validation procedure was selected near the experimental values that have been described by Timmer [52]. The domain size was 2.6 m in length, 1.8 m wide and 1.25 m high, with an octagonal face as shown in Figure 23. The wing model had a chord length of 0.25 m.

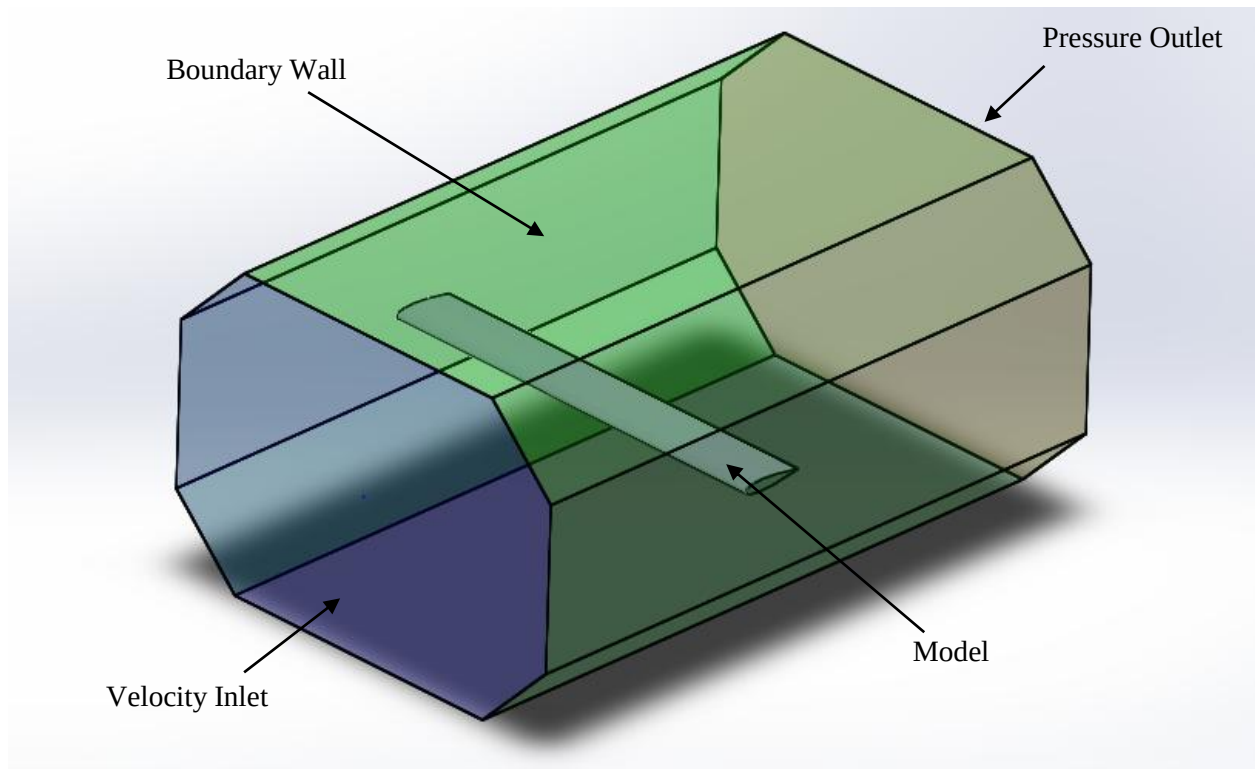


Figure 23 - Domain with boundaries defined

The inlet boundary wall is treated as a velocity inlet, with the velocity set to 40 m/s. This will then represent the speed at which the air flows past the model, with the model standing still relative to the ground. This was also the method used by Mansor *et al.* [27]. For the model chord length of 0.25 m, this velocity setting relates to the Reynolds number of 0.7×10^6 that was used in the experimental setup. The outlet boundary is set to a pressure outlet with the pressure setting at 0 Pa. Both these settings have been discussed in the literature. The boundary walls are treated as solid walls, and not as symmetry planes, because the test setup was conducted in a closed domain with walls exhibiting a no-slip condition.

Due to the symmetric nature of the problem, only half of the geometry was used to generate the 3-D mesh; in other words, half of the fluid volume. The geometry was divided into two regions to enable the use of an overset mesh.

(Refer to Annexure A for a summary report of the CFD simulation).

4.1.1 Mesh Configuration

The volume mesh in a simulation is a mathematical description of the geometry used in the problem being solved. The mesh is constructed from mesh entities such as vertices, faces and cells [53].

The code used to construct the mesh was STAR-CCM+, as already specified. There are different meshing techniques available to create these mesh constructions, but for the purpose of this study it was decided to use the trimming method. The trimming method provides fast and high quality meshes [54]. This method constructs hexahedral cells to represent the volume of the geometry.

Furthermore, the surface remesher and the prism layer mesher models were also activated. It is important to note that the stretching mode for the prism layer mesher is set to wall thickness, while the rest of the functions are kept to the default settings. In the case of the surface remesher, proximity refinement is disabled as is automatic surface repair. The minimum surface quality must also be set to a value of zero (0). Per region meshing is also set to true.

The cells on the xy-plane at $z=0$ was retained and the remaining volume cells in the 3-D mesh was removed from the simulation. By doing this, the 3-D mesh was converted to a 2-D mesh (quadrilateral cells), thus simplifying the simulation and reducing the CPU time required to run the simulation. To properly capture the trailing edge flow, volumetric control regions were incorporated to refine the mesh behind the airfoil, as shown in Figure 24. Volumetric control regions increases the number of cells and computation time required substantially, but increases the accuracy of the solution [55].

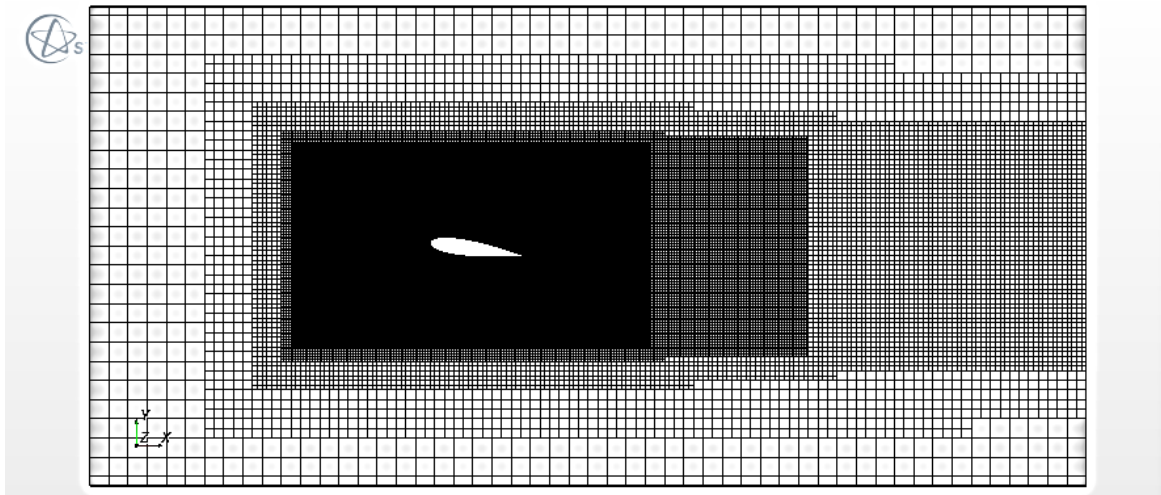


Figure 24 – Quadrilateral 2-D mesh

Comparing the simulation results of an airfoil with the results in the literature, it is required to simulate the airflow around the airfoil at a range of different angles of attack. An overset mesh was thus incorporated into the simulation. This is illustrated in Figure 25, where the overset mesh is rotated through 8° . The

overset mesh function enabled the angle of attack to be changed by simply rotating the overset mesh region, thus the airfoil, and running the simulation again.

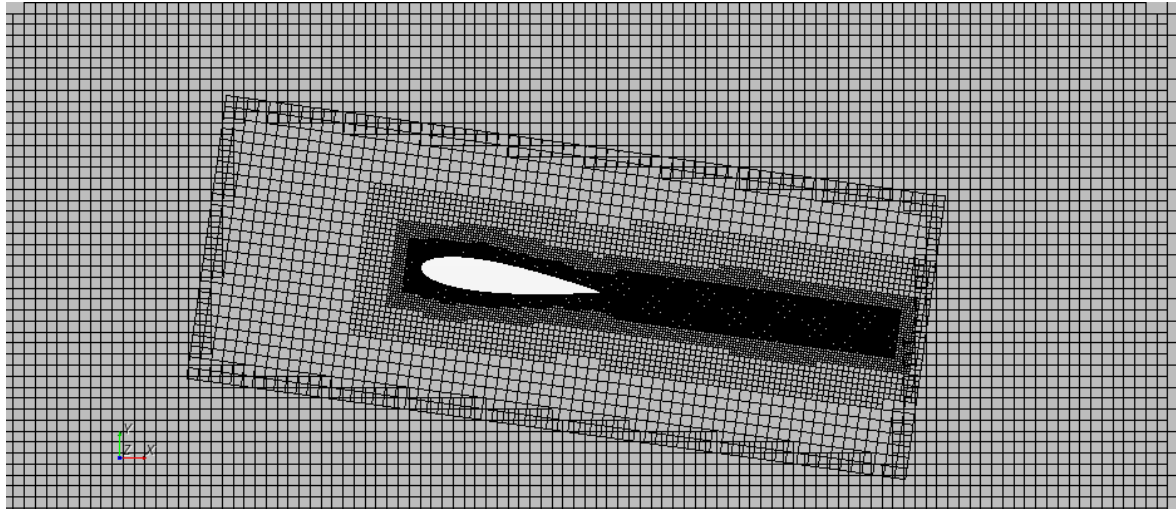


Figure 25 - Overset mesh: airfoil rotated to an angle of attack of 8°

In order to capture transition, a very fine, layered mesh, called a prism layer, must be used on the boundary wall of the airfoil (Figure 26).

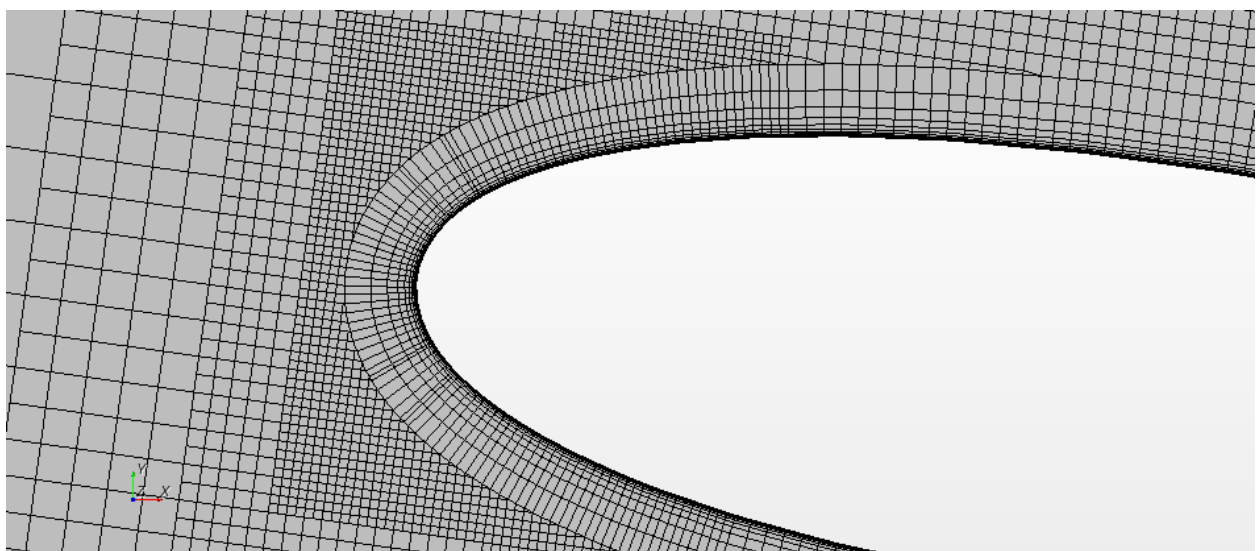


Figure 26 - Prism layer on the leading edge of the NACA 0018 airfoil

The mesh can, however, not be too fine as this will mean more cells and therefore the simulation will take longer to solve. A coarser mesh was therefore first generated and then refined.

4.1.2 Solver Configuration

In order to determine the aerodynamic coefficients of the airfoil, the solver had to be set up with flow conditions matching those of the wind tunnel tests. Standard atmospheric conditions were accepted for the simulation setup, as summarised in Table 3.

Table 3 - Atmospheric Conditions

Pressure	101325	Pa
Density	1.225	kg/m ³
Temperature	288.16	K
Dynamic Viscosity	1.85508x10 ⁻⁵	Pa.s

The wind tunnel tests from Timmer [52] were conducted at Reynolds numbers of 0.3×10^6 and 0.7×10^6 . The Reynolds number is calculated by

$$Re = \frac{\rho V D}{\mu} \quad 2.19$$

where ρ is the air density, V is the free stream velocity of the fluid, D is the length of the object and μ is the dynamic viscosity of the fluid.

In the literature the pressure coefficient plot is also described for an angle of attack of 8° and a Reynolds number 0.7×10^6 . Therefore, the simulations were conducted at a range of angles of attack at a constant Reynolds number value of 0.7×10^6 . With a chord length of 0.25 m, the corresponding free stream velocity should therefore be 40 m/s.

The boundary walls were kept stationary for the simulation and defined as non-slip. This was to simulate the boundary walls of the wind tunnel.

To properly predict the aerodynamic coefficients, a transition model was used, as already noted. To determine whether the solver had converged, the plots for these aerodynamic coefficients were also monitored for convergence.

4.2 Validation Procedure

Before the simulation results can be compared with the experimental results, mesh refinement has to be applied to determine whether mesh independence has been reached. Mesh independence refers to the point where the solution is independent of mesh resolution, meaning that the solution is not improved any further as the mesh is refined. As the mesh is refined, the number of volume cells is increased and therefore the computing time required to solve the simulation. If the mesh is refined further without any significant increase in the accuracy of the result, time is wasted. A base size should then be determined that will be used to obtain the results which will be compared to the experimental results.

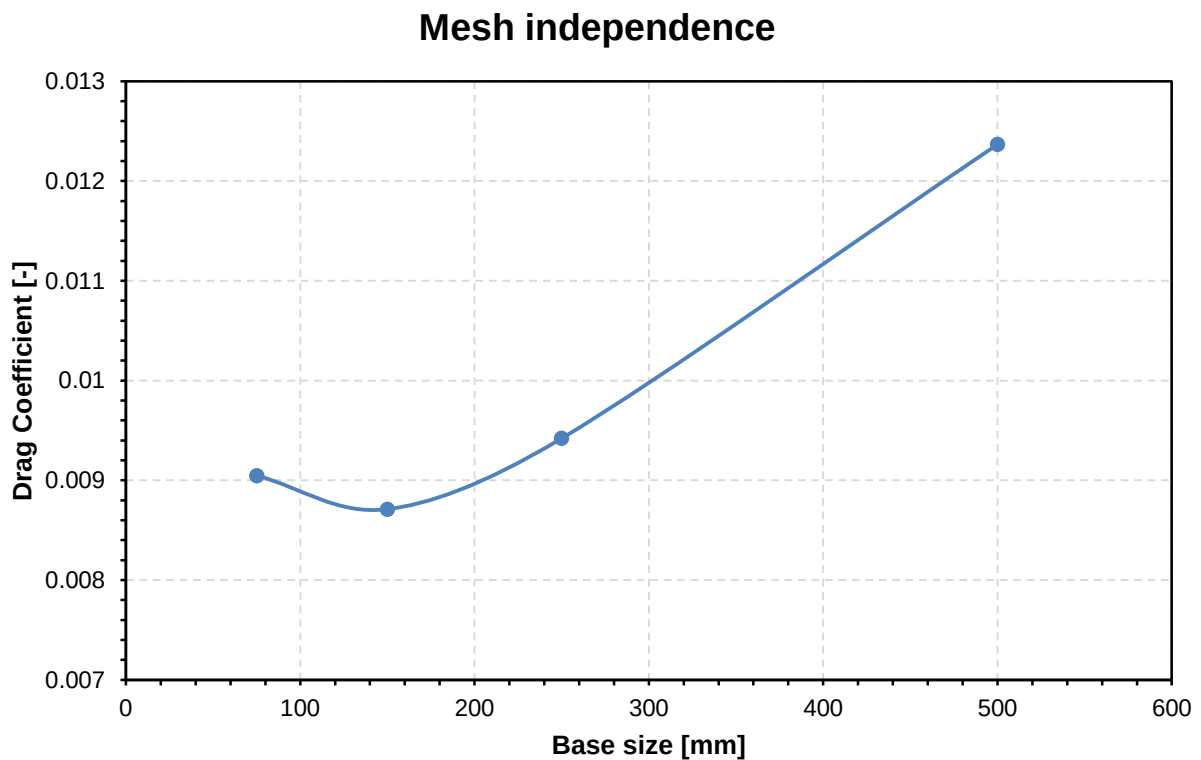


Figure 27 - Mesh refinement for the 2-D simulation at an angle of attack of 0°

Referring to Figure 27, the drag coefficient decreases as the base size of the volume cells is decreased. This occurs up until a certain point, at a base size of 150 mm, after which point the drag coefficient starts to increase again. This is as a result of a too fine mesh. Reducing the base size, reduces all of the cells in the model. Refinements should be incorporated in specific areas in order to properly capture the drag coefficient of the model. The cells might become smaller than the tessellations of the model and this will cause the

solution to become unstable. It will have difficulty converging and therefore the drag coefficient value will increase again.

Timmer [52] compared the experimental results from the wind tunnel with plots generated using RFOIL [56]. RFOIL is a modified version of the MIT-developed XFOIL code and uses non-linear amplification of the boundary layer [52].

The transition locations are given by Timmer and can therefore be compared with the results from the CFD simulations. Transition has a large effect on the aerodynamic coefficients of the airfoil and will also determine the accuracy of the simulation. Skin friction plots were used to determine the transition positions in the CFD simulation and then compared with the literature.

In the literature it is also reported that at an angle of attack of 8° , laminar separation starts on the upper surface. This is revealed in the pressure coefficient plot of the airfoil at an angle of attack of 8° , generated using XFLR5. XFLR5 is a program with a user interface, based on XFOIL. As mentioned, RFOIL is also based on the XFOIL code, thus it was assumed that the plots generated using XFLR5, would be nearly the same as those generated using RFOIL, as in the literature. The plot of the pressure coefficient obtained from the CFD simulation, was compared with reported findings in order to determine if laminar separation occurred at the same point.

To further determine the accuracy of the CFD simulation, the values for the aerodynamic coefficients obtained from the CFD simulation will be compared with the experimental graphs, as well as the graphs obtained from RFOIL.

4.3 Validation Results

In order to ensure that the values used for plotting data was accurate, it was important to ensure that the solution had converged (Figure 28). Monitor plots for the lift, as well as the drag coefficients, were used to determine whether a converged solution had been reached.

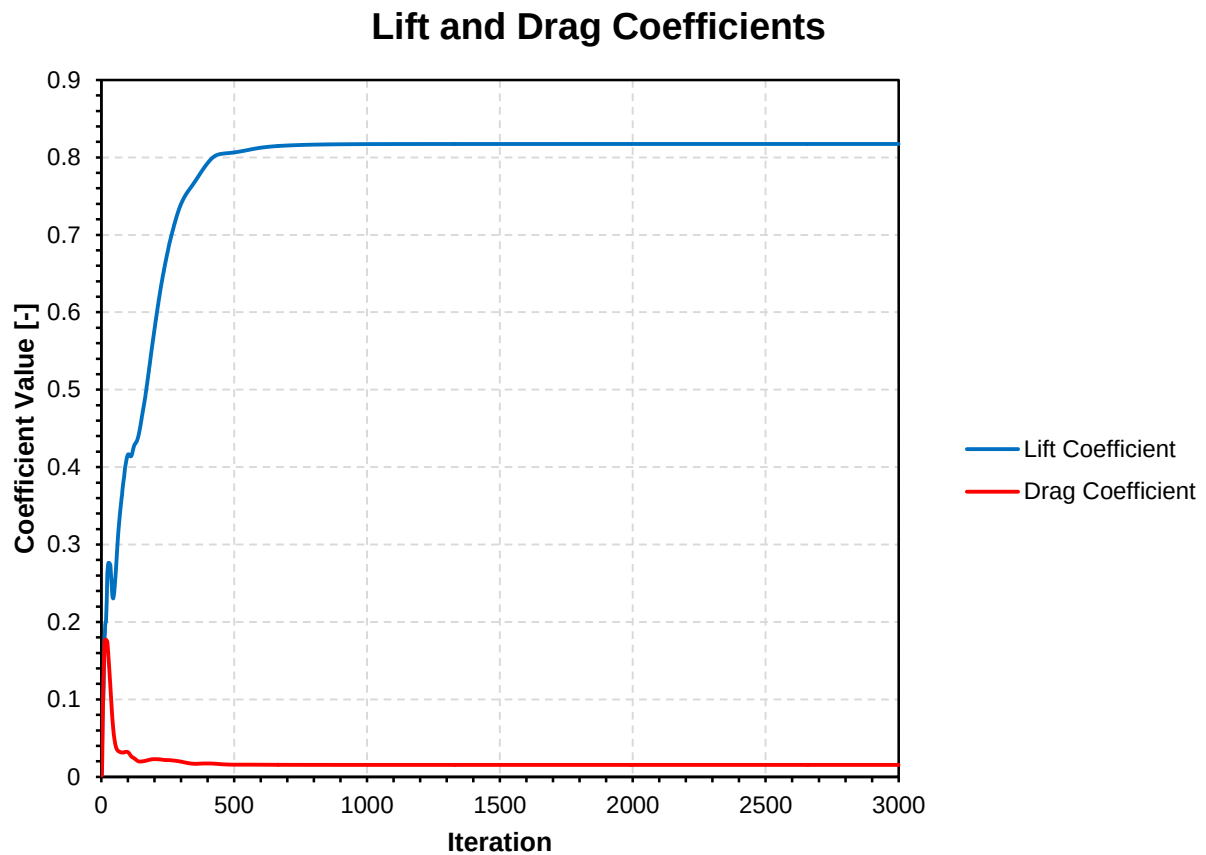


Figure 28 - Lift and Drag coefficients, 8° angle of attack

In order to compare the CFD simulation results with the experimental and the XFLR5 results, the simulation was run at angles of attack ranging from 1° to 16°. The simulations were also run using two different turbulence models available in STAR-CCM+, namely the k-omega and k-epsilon turbulence models. The airfoil overset mesh was rotated to change the angle of attack. Excel was used to plot the simulation data against the experimental data.

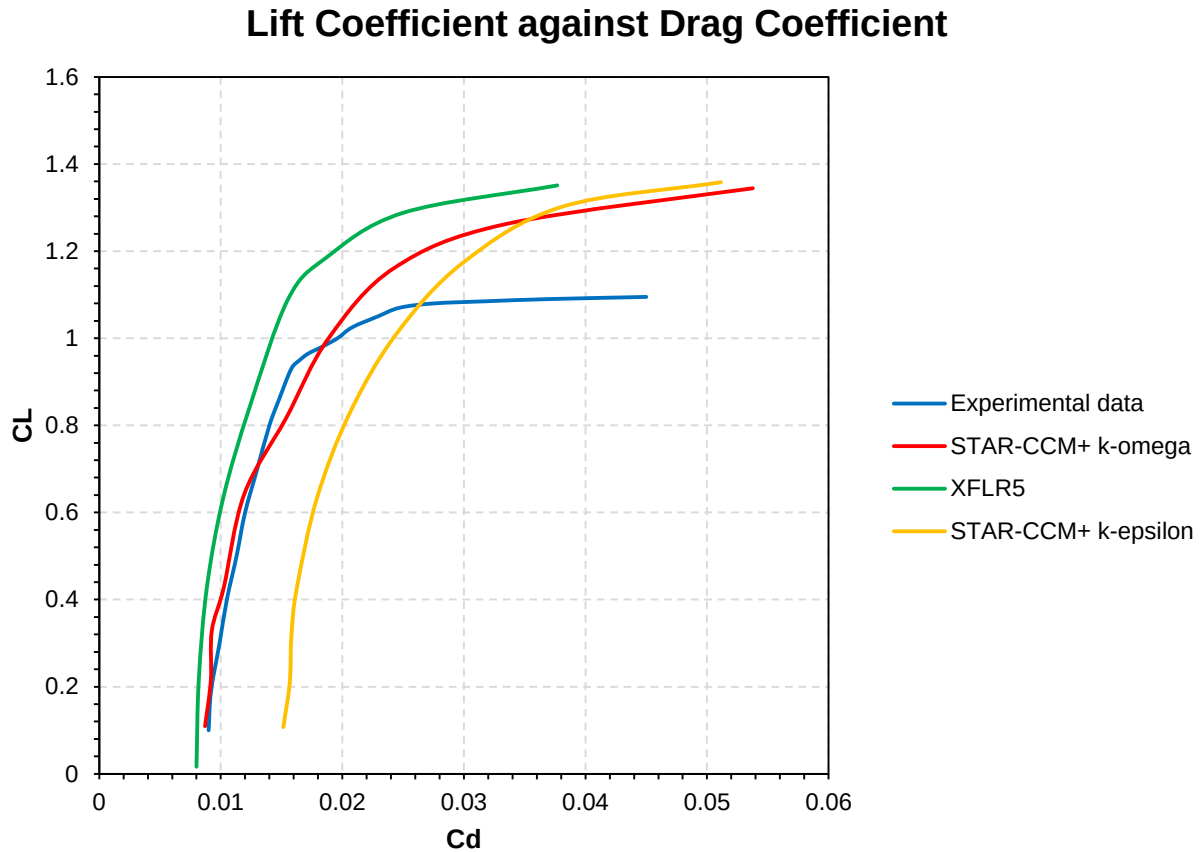


Figure 29 - Lift coefficient compared with the Drag coefficient, $Re = 0.7 \times 10^6$

Referring to a study by Ahmed *et al* [55], it is indicated that the k-epsilon turbulence model is the model typically used in simulations of external flow around bodies. It was therefore compared to the k-omega turbulence model which was discussed in the literature survey.

When the lift coefficient is compared to the drag coefficient in Figure 29, both XFLR5 and STAR-CCM+ over-predicts the values for the lift coefficient, and the drag coefficient value increases with increasing angles of attack. Even though this is the case, the same trend is obtained as in the case of the experimental data. For lower angles of attack, the CFD data and the experimental data compare well. For drag, as well as lift coefficient values for angles of attack up to 9° , the CFD and experimental values differ by less than 6%. At an angle of attack of 15° , however, the CFD data exceeds the experimental data in excess of 15%. The k-epsilon turbulence model over-predicts the drag coefficient at low angles of attack; the value for the drag coefficient being about 74% greater than the value calculated using the k-omega turbulence model.

The values for the lift coefficient were also compared with the different angles of attack. In Figure 30 it is clear that the STAR-CCM+, as well as the XFLR5, plots compare extremely well with the experimental data for low angles of attack.

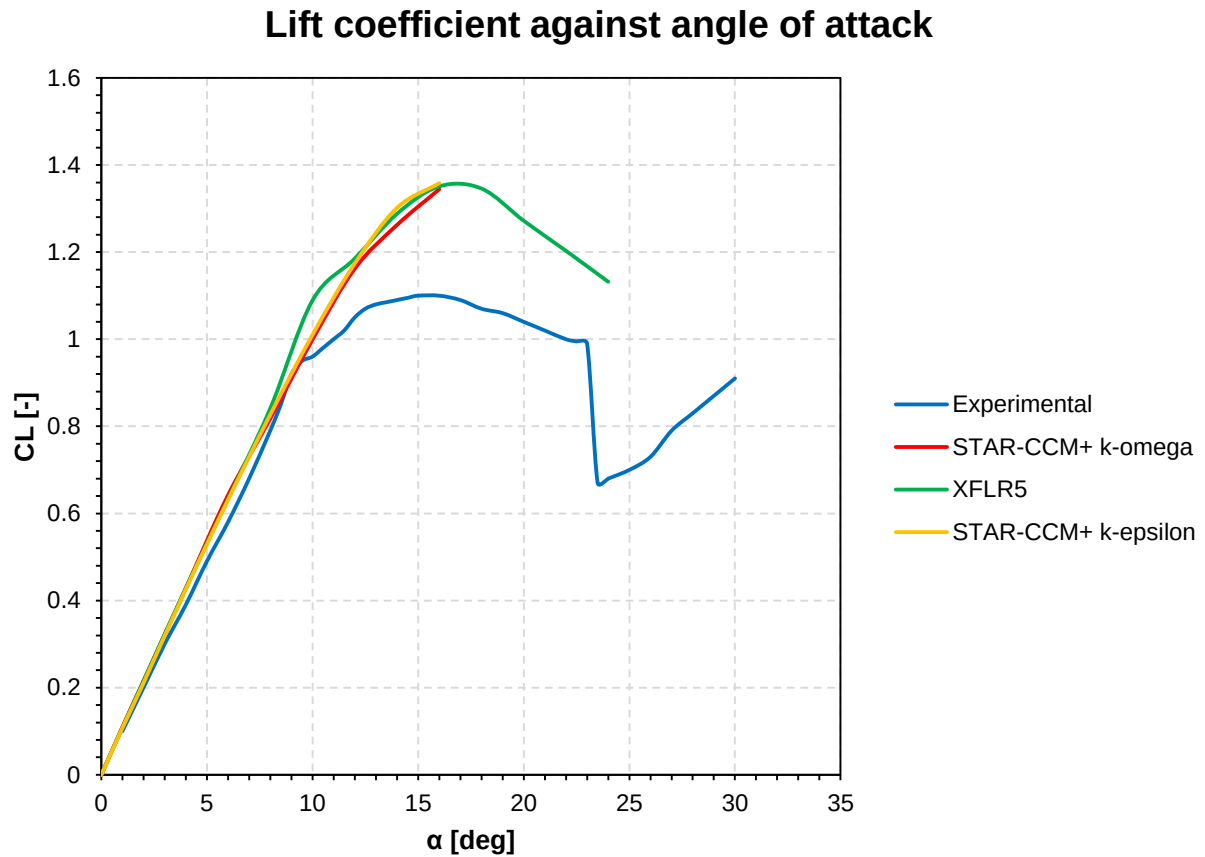


Figure 30 - Lift coefficient compared with the angle of attack, $Re = 0.7 \times 10^6$

Again, at angles of attack higher than 10° , STAR-CCM+ and XFLR5 over-predict the values for the lift coefficient, exceeding the experimental data by more than 15%. This is the case for both turbulence models used in the STAR-CCM+ simulations. The same profile is still maintained. However, the values for STAR-CCM+ never differ by more than 5% from the XFLR5 data.

The experimental tests were completed to an angle of attack of 30° , but the values for XFLR5 only extended to 24° , after which the result no longer converged. At an angle of attack higher than 16° , STAR-CCM+ had difficulty converging, oscillating around a single value. This is because the flow around the airfoil has

separated from the surface. This separation point is indicated in Figure 31. This eliminates the need for the prediction of transition as it has already occurred and the solution will therefore not converge.

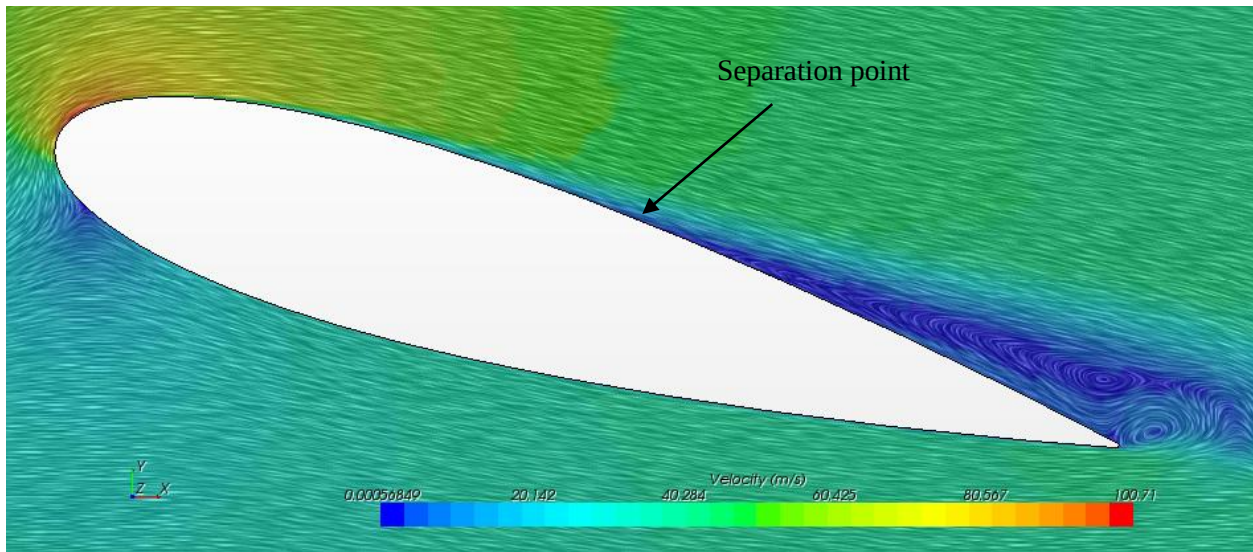


Figure 31 - Separation point on airfoil

The STAR-CCM+ values are therefore only plotted until an angle of attack of 16° . The lift coefficient at the different angles of attack is accurately calculated by both turbulence models used in STAR-CCM+, with the values remaining within 2% of one another.

It is indicated by Timmer that at 0° angle of attack, transition occurs at 54% of the cord length (c) of the NACA 0018 airfoil [52]. In Figure 32 transition occurs where the skin friction coefficient starts increasing, before decreasing later on. Here it occurs at about 0.14 m, which is 56% of the chord length. This is 2% less than the experimental value.

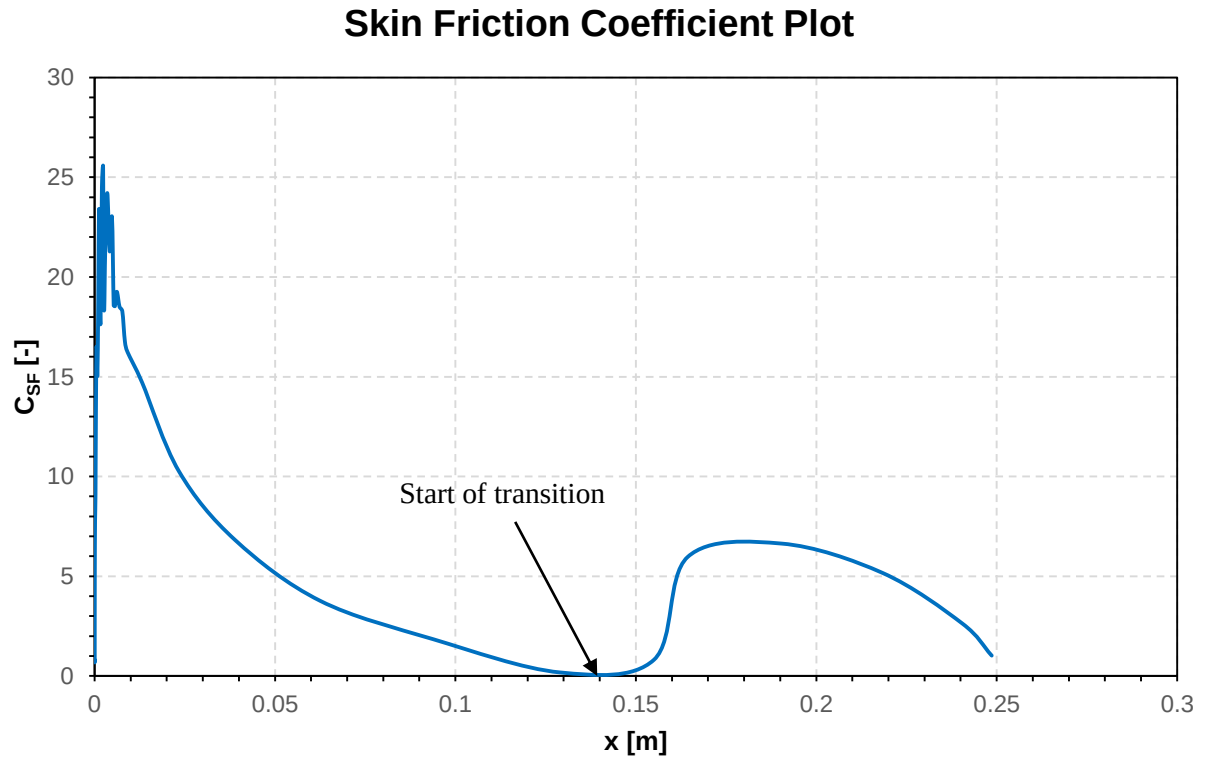


Figure 32 - Skin friction coefficient plot for a 2-D NACA 0018 airfoil, 0° angle of attack

In Figure 33 it is observed that the pressure coefficient values for STAR-CCM+ compare very well with the data obtained from XFLR5. The plot for XFLR5 data runs on the line of the STAR-CCM+ data plot, remaining within 5% of one another.

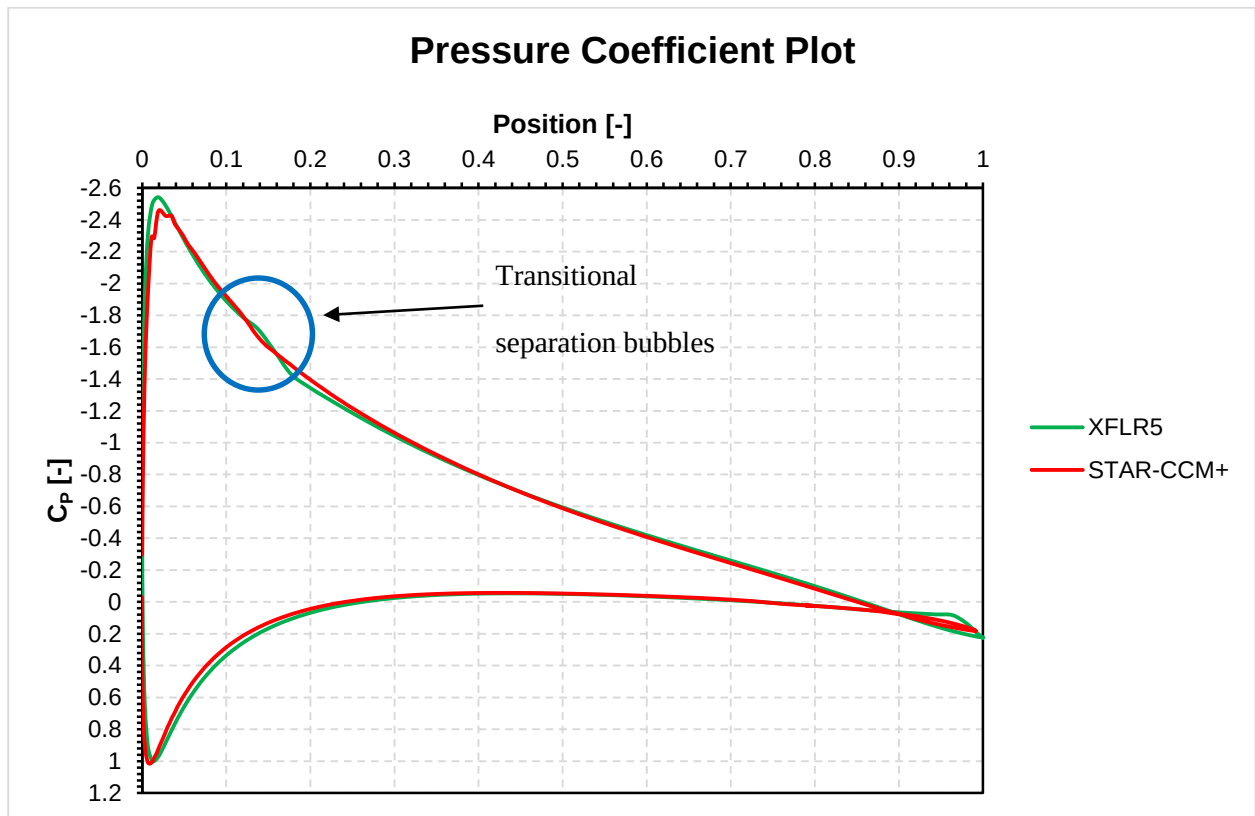


Figure 33 - Pressure coefficient of the 2-D NACA 0018 airfoil, 8° angle of attack

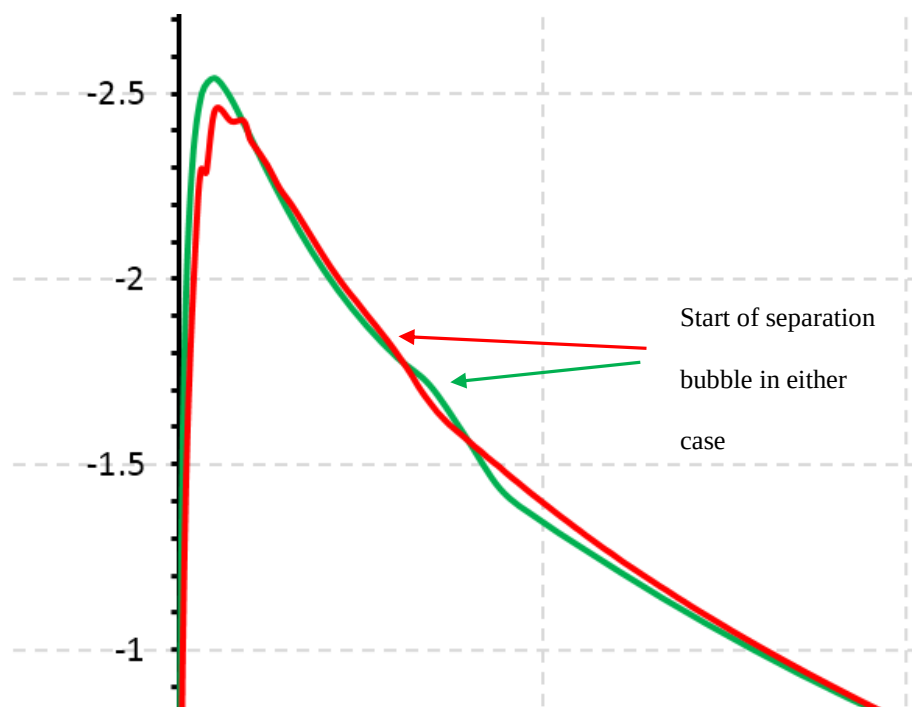


Figure 34 - Separation bubble formation on the upper airfoil surface, 8° angle of attack

In Figure 34 the separation bubble is indicated by the slight rise in the pressure coefficient for XFLR5. Here the separation bubble formation starts to form at about 15% of the chord length. Looking at the same

figure, the data from STAR-CCM+ reveals that the separation bubble starts a bit earlier, at about 13% chord length. Note that in Figure 33 and Figure 34, the length of the airfoil is not expressed in the actual chord length of 0.25 m, but as a percentage.

4.4 Conclusion

The CFD results compare very well with the experimental results from the literature at low angles of attack. The difference observed is that the CFD results are over-predicted at higher angles of attack. After separation has occurred, the CFD software has difficulty converging and the results after separation can therefore not be compared to the experimental results. The transition locations of both sets of results also compare well, keeping the above in mind. It can be concluded that the use of the specific transition models for simulating transition of 2-D profiles yields satisfactory results. The transition models can therefore be validated for a 3-D case similar to that of the solar-powered vehicles.

Chapter 5: 3-D Validation: HPT VeloX1

In order to improve on the current SV design of the North-West University, it is necessary to analyse the current design using CFD. To validate the procedure that would be used in order to analyse the current design, CFD is used to analyse the HPT's (Human Power Team) VeloX1 bicycle (see Figure 35 [57]). The HPT is a group of students from the Delft University of Technology in the Netherlands, who aims to use this bicycle to break the land speed record for this class of human powered vehicles.



Figure 35 – Human Power Team's VeloX1 [57]

A full scale model of the VeloX1 had been analysed in a wind tunnel at a velocity of 28m/s, and these results were made available by the HPT group to use for validation purposes of this study. The road test data was also supplied. It was therefore viable to do a CFD analysis on the CAD model of the VeloX1 and compare it with the experimental data.

Once the simulation is validated, the same procedure would be applied to the North-West University's SV to determine the aerodynamic drag values of the vehicle. This could be done as the driving speeds of solar-powered vehicles could also be of the order of 28 m/s, which is about 100 km/h.

The experimental results for the road tests as well as for the wind tunnels tests, were obtained from the team and permission was granted by the team to use the model in the validation process for this study.

5.1 Validation Setup

In order to properly validate the 3-D model, transition locations have to be accurately determined for 3-D flow around the HPT VeloX1 model. The transition locations of the CFD simulations will be compared with the wind tunnel transition locations, thus ensuring that the CFD simulation is accurate enough to use as a design tool for the design of the SV. Furthermore, the drag coefficient values of the simulation should also be relatively close to the wind tunnel results to further enhance the validity to use the CFD simulation as a design tool.

Once again, as in the case of the 2-D validation, STAR-CCM+ will be used to do the CFD simulation. During the 2-D validation, the γ - Re_θ transition model, coupled with the $k-\omega$ SST turbulence model, was used to determine the transition on the airfoil. These positions position were determined within 5% of the experimental values, and will therefore also be used to determine the transition locations on the 3-D model.

Regarding the geometry to simulate the fluid flow around the bicycle in the wind tunnel, a rectangular tunnel was used based on dimensions described in the literature, originally by Mansor *et al.* [27]. The dimensions of the simulated wind tunnel was not the same as the experimental wind tunnel used. A rough simulation was conducted with dimensions similar to the actual wind tunnel, but this increased the number of cells required in the simulation without a significant improvement in the results obtained. It was thus opted to use the simpler, rectangular wind tunnel in the simulation. The rectangular tunnel should be at least twice as high as the model, with the outlet boundary at least five vehicle lengths behind the model to ensure that the wake of the vehicle is properly captured. The inlet boundary should be at least two vehicle lengths ahead of the model to ensure that the flow has had sufficient time to re-laminarise before reaching the model.

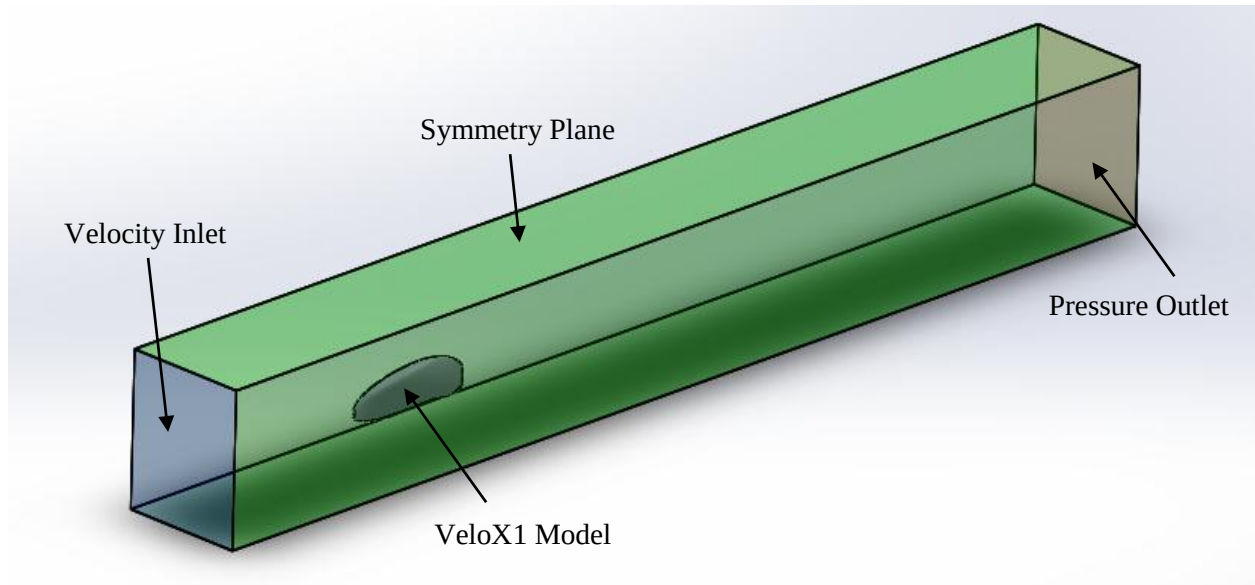


Figure 36 - Domain description for the 3-D validation case

The boundary conditions for the 3-D validation case is described in Figure 36. The inlet boundary was set to a velocity inlet as in the 2-D validation setup, with a velocity of 28 m/s. The pressure outlet is set to a 0 Pa pressure outlet condition and the walls of the tunnel was treated as fixed walls to represent a wind tunnel. Due to the symmetric nature of the model, only half of the geometry was simulated. This meant that the results would have to be doubled to get the value for the whole model. The plane separating the domain in half is treated as a symmetry plane.

5.1.1 Mesh Configuration

Different meshes were compared in order to determine which was most accurate. It is generally accepted that the results are more accurate when using the polyhedral mesher instead of the trimmer mesher. The trimmer mesh uses hexahedral cells to build the geometry, while the polyhedral mesh uses polygons with up to 14 sides. The polyhedral mesher, however, uses more cells to build the geometry. The trimmer mesh uses much less than the other meshing techniques and therefore less computing time. The trimmer mesh is therefore used to conduct the simulations.

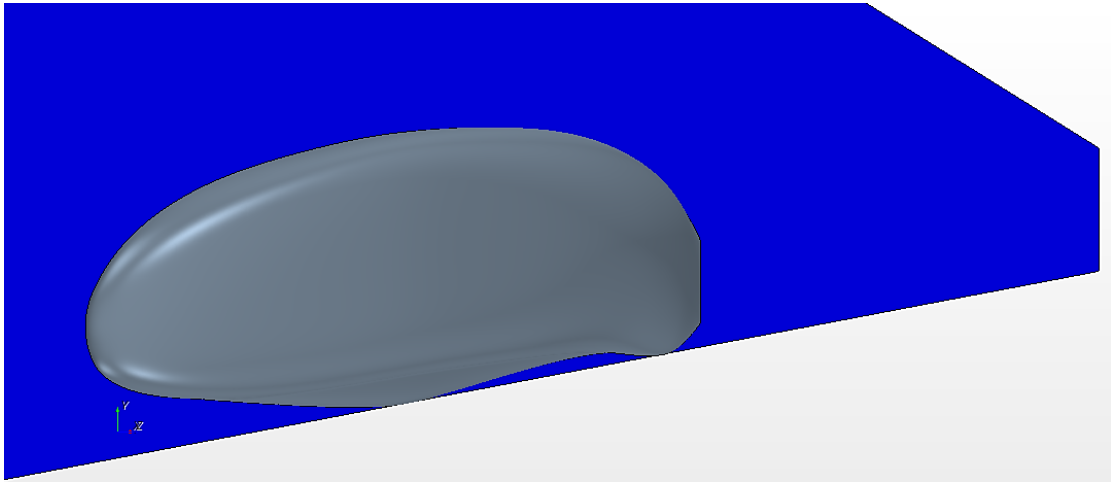


Figure 37 - VeloX1 geometry

As for the 2-D validation case, the surface remesher and the prism layer mesher models were also activated. The stretching mode for the prism layer mesher is set to a stretch factor while the rest of the functions are kept to the default settings. In the case of the surface remesher, proximity refinement is disabled, as is automatic surface repair. The minimum surface quality must also be set to a value of zero (0).

To properly capture the trailing edge flow, volumetric controls were once again incorporated in the mesh to refine the mesh behind VeloX1. This is indicated by the higher density of cubes, as shown in Figure 38.

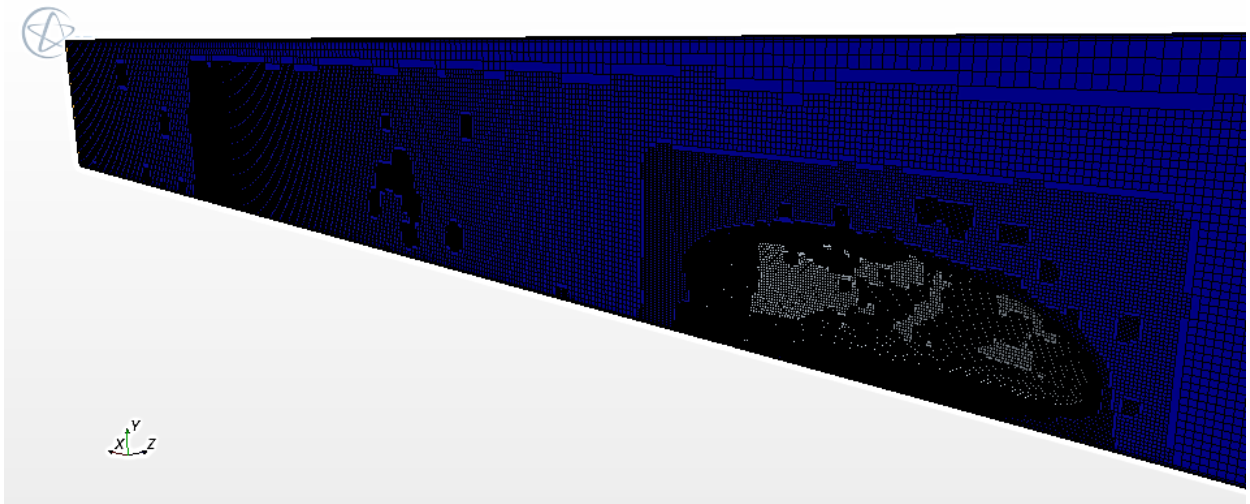


Figure 38 - Mesh refinement for VeloX1 trailing edge flow

In order to capture transition, a prism layer must again be used on the boundary wall of the VeloX1, as in the 2-D validation case with the airfoil.

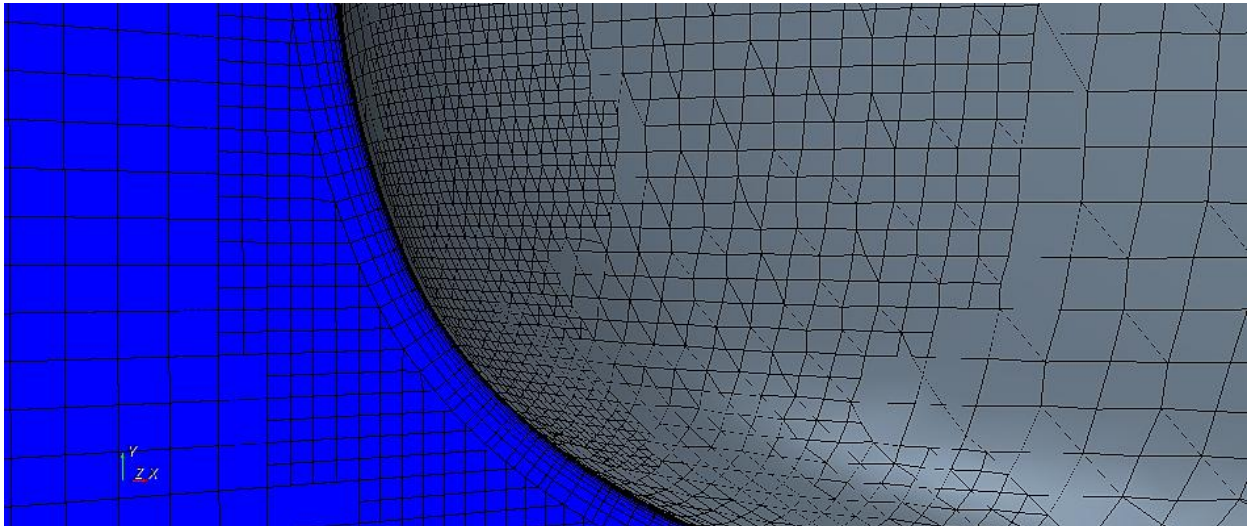


Figure 39 - VeloX1 prism layer on surface mesh

5.1.2 3-D Solver Configuration

Atmospheric conditions given in Table 3, were also used in the 3-D validation.

The wind tunnel tests were conducted at velocities of 28 m/s and therefore the inlet boundary velocity was set to the same value. This resulted in a Reynolds number of almost 5×10^6 . This Reynolds number is based on the length of the bicycle, which is 2.7 m.

5.2 Validation Procedure

To determine whether the 3-D validation is accurate, the transition locations of the simulation will be compared with the transition locations detailed in the literature, as supplied by the HPT. This literature is in the form of a report and gives the transition locations for the wind tunnel tests, as well as a simulation that was conducted using VSAero.

VSAero is based on the assumption that the model being simulated is at rest in a moving airstream, relating to the chosen method to conduct the simulations of the model, which is also simulated in a moving airstream rather than moving at a velocity through the air. The flow field is also assumed to be inviscid, irrotational and incompressible except for regions that are dominated by viscous, rotational flow. Such areas are confined to thin surface boundary layers and wakes. After reviewing the program, Maskew concludes that VSAero produced comparable results to high order methods [58]. These higher order methods include modern day CFD methods.

The HPT also gave the drag area values for the wind tunnel tests, the road tests and the drag area ($C_D A$) as calculated by VSAero. This means that the tests were not conducted using a reference area. The CFD simulations will therefore be run to give only the resulting aerodynamic drag force (D) on the surface of the model:

$$D = \left(\frac{1}{2} \rho V^2 \right) (C_D A) \quad 5.1$$

This will give a value for the drag area, which will then be compared with the experimental values.

The pressure plots on the surface of the vehicle body will also be compared with the experimental results. These plots will also give an indication of the accuracy of the simulation.

5.3 Validation Results

To determine whether the CFD simulation of the VeloX1 is accurate, the simulation results must be compared with the experimental data as described in the preceding section 5.2.

The data supplied by the HPT describes the VeloX1 as having a drag area value, for the wind tunnel test, of 0.031 m² and a value of 0.033 m² for the road tests. These tests were conducted without closing the wheel holes and the ventilation ducts. The wind tunnel tests were repeated with the ducts closed off. This yielded a $C_D A$ value of 0.025 m². The CAD model does not make provision for these ducts or wheel openings, therefore the CFD simulation results must be compared with the second case (when the ducts were closed).

The VeloX1 has a frontal area of 0.363 m². Using this value, it is easy to determine the drag coefficient of the VeloX1 when the ducts are closed off. With the drag area of 0.025 m², the experimental drag coefficient is 0.069.

To determine if the CFD simulation has converged, a force monitor plot is generated and displayed in Figure 40. This is the aerodynamic force (D) that is exerted on the surface of the VeloX1 and Eq. 5.1 can be used to determine the simulated value for $C_D A$. The plot shows that the results oscillate around a final value. This can be eliminated by averaging the values of the last 100 iterations, which will give a fair representation of the mean drag force value.

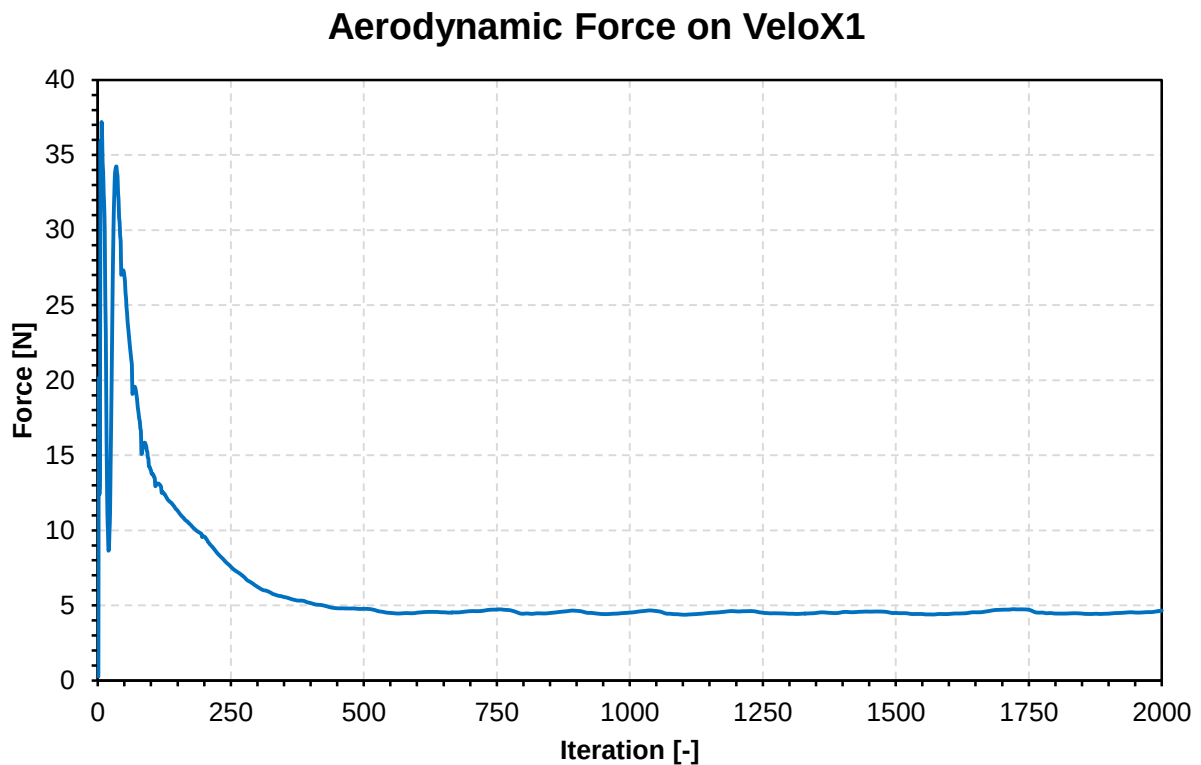


Figure 40 - STAR-CCM+ force monitor plot

The simulated value for half of the aerodynamic force on the VeloX1 is 4.648 N. Because a half symmetrical body was simulated, this value was multiplied by 2 resulting in 9.296 N for the whole body, which translates to a $C_D A$ value of 0.0196 m^2 and a drag coefficient of 0.0533. This is 22.5% less than the experimental value from the wind tunnel tests. It is difficult to simulate the aerodynamic drag precisely, but predicting the transition location is much more important. The difference can be attributed to features in the wind tunnel that are near impossible to incorporate into the CFD simulation. CFD simulations were also run where effort was made to draw the wind tunnel geometrically correct, but this yielded a difference of less than 5% to the rectangular wind tunnel geometry. The difference noted here was that the rectangular wind tunnel had fewer cells. With this small difference in drag force values, it was opted to use the rectangular geometry due to its simplicity and shorter computing time.

Table 4 - Drag Area Results Comparison

Model	$C_D A$	C_D
Wind Tunnel Test	0.025	0.0688
VSAero	0.015	0.0413
STAR-CCM+	0.0196	0.0533

Looking at the simulations conducted using VSAero, the $C_D A$ value is 0.015 m^2 and the drag coefficient is 0.0413. The geometric setup for the wind tunnel of this simulation is not known, but the reasons for this lower prediction will probably be the same as for the above-mentioned. This value is about 22.5% lower than the STAR-CCM+ simulated value.

The transition locations were determined by the HPT using wind tunnel tests, as well as using VSAero. . Laminar flow is experienced on the front half of the VeloX1's body. At about the halfway mark, the laminar boundary layer starts to undergo transition to a turbulent boundary layer. Referring to Figure 41, the point of separation is indicated by an arrow. Transition is induced at the start of the front wheel fairing, as well as at the top of the windscreen. The position of the ventilation duct also causes the flow to separate prematurely.



Figure 41 - VeloX1 transition location with the ventilation ducts taped [57]

The transition location of the model simulated using VSAero, shows quite a few differences to the wind tunnel results. The model in Figure 42 shows that the flow does not separate at the start of the front fairing, neither does the location of the windscreen as the whole model has a smooth surface.

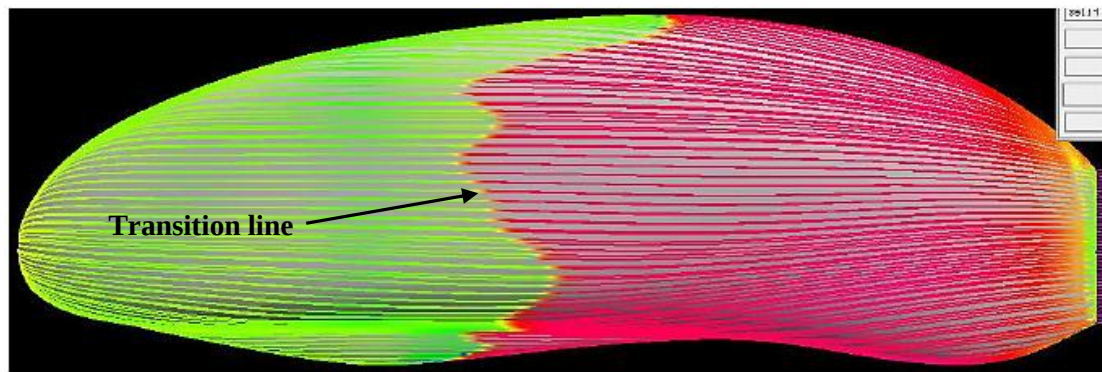


Figure 42 – VeloX1 transition location by VSAero [57]

Referring to Figure 43, transition determined by CFD occurs just before half the length of the model, with laminar boundary layers on the top of the model progressing past half the length of the model. One can see that the CFD-predicted separation is a combination of the wind tunnel results and that of VSAero. The geometry used does not have a windscreen to cause flow separation, observed when referring to Figure 41. However, the laminar flow extends past the halfway mark, indicated by the solid line in both Figure 42 and Figure 43.

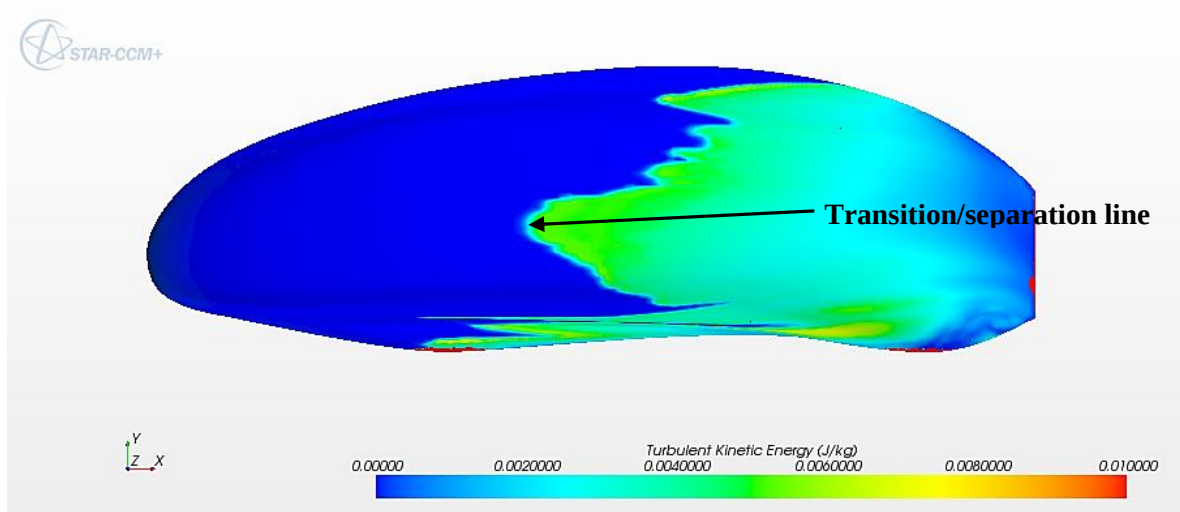


Figure 43 - VeloX1 transition location by STAR-CCM+, turbulent kinetic energy plot

The pressure coefficient on the surface of the VeloX1 was also supplied by the HPT and will be compared with the simulation results. The pressure coefficient calculated by VSAero is illustrated in Figure 44.

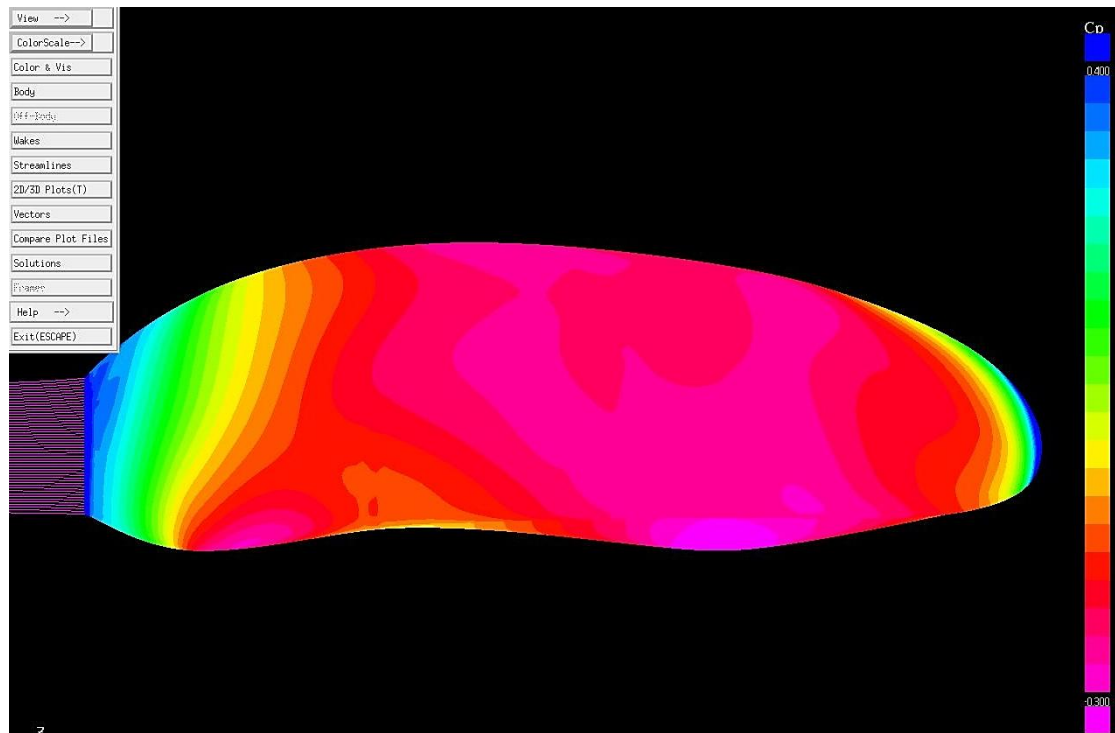


Figure 44 - HPT VeloX1 pressure coefficient by VSAero

Comparing Figure 44 with Figure 45, it can be observed that the pressure on the surface of the VeloX1 is very similar, with differences in the maximum and minimum values. However, the minimum and maximum pressure locations are the same for both simulations, with the lower pressure values being around the centre of the vehicle body, thus near the thickest section. This correlates with the literature, which indicates that flow separation will usually occur when the pressure reaches a minimum and pressure recovery occurs to the rear of the vehicle body.

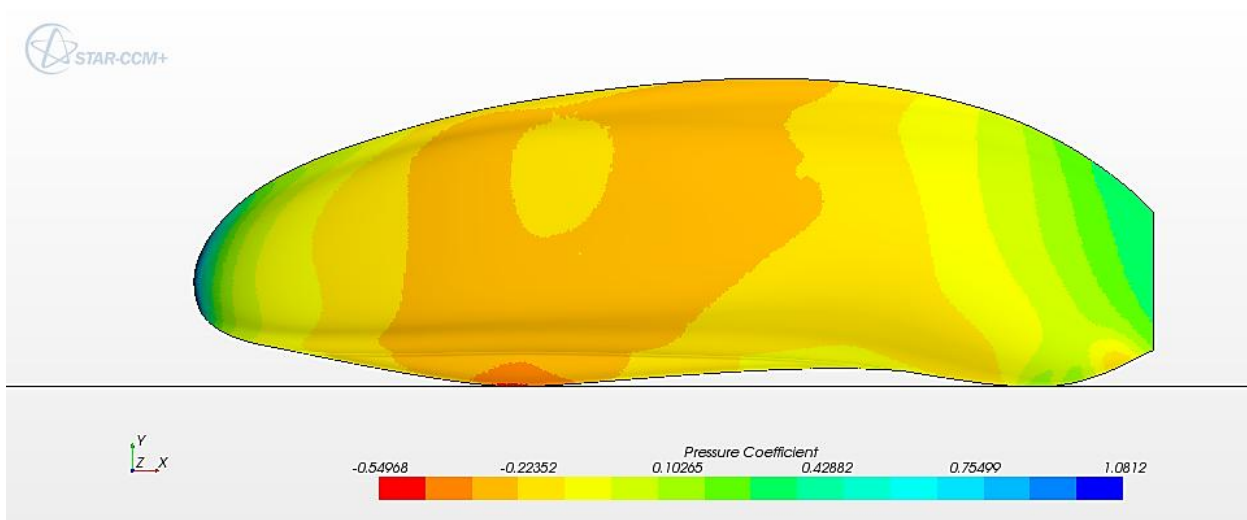


Figure 45 - VeloX1 pressure coefficient by STAR-CCM+

It is also clear, when referring to Figure 45, that total pressure recovery is not achieved, indicating that flow separation has occurred.

5.4 Conclusion

The values calculated by the CFD simulation for the 3-D case are lower than that of the experimental results, 22.5% lower in the case of the STAR-CCM+ simulation and about 50% lower for the VSAero simulations. This can be attributed to various reasons, including the fact that the actual model has a windscreen, wheel holes and ventilation ducts. All of these factors contribute to a higher drag value. Other factors to consider include surface roughness and manufacturing irregularities. The transition plots of the different models compare relatively well. There are differences in the extent to which the models experience laminar flow, but these differences can be attributed to differences between the actual model and the CAD models (the CAD model being smoother). The pressure distribution plots also compare well. It can therefore be assumed that the transition models can be used to predict the transition of the 3-D models.

Chapter 6: Conceptual Design

During the conceptual design phase, different ideas for the design were explored and references were made to existing designs of popular solar vehicles (Refer to Section 2.6). These steps are important during this phase, leading up to the final design of the vehicle. Many existing designs have had the driver pod in the middle of the vehicle, with the wheels covered by fairings. However, by analysing these designs in terms of the design parameters discussed, it is clear that the frontal area of the design is significantly increased.

It was suggested to move the driver to the side and integrating the driver pod with the fairing, resulting in a catamaran design. This not only resulted in a design with a smaller frontal area, but also one with a smaller wetted surface. Theoretically this means a lower overall drag coefficient. This also decreases the junctions between the fairings and the body and should further aid in decreasing the drag coefficient of the vehicle, which is a design parameter. These parameters can be further reduced by using shorter and narrower fairings on the wheels opposite to the driver position of the vehicle.

Three basic designs were proposed for the next generation solar-powered vehicle based on the above ideas. These designs were proposed to the team for comparison and discussion in order to determine which design will potentially have the best efficiency. Review of existing solar-powered vehicles revealed that the proposed concept designs were similar to these existing designs. This indicates that the methodology used to achieve these concept designs is also used by the leading solar car teams around the world.

6.1 Concepts

The airfoil used for the main body of the vehicle is a modification to the S5 airfoil used in the wing design of Jonker Sailplanes. This airfoil was obtained from Jonker Sailplanes for use in the design of the solar-powered vehicle and has also been used in the 2012 vehicle. This also applies to the airfoil used for the wheel fairings during the conceptual phase, which is the symmetrical airfoil used for the tail wing of the same sailplanes. By using the same airfoil for all three conceptual designs, the design with the least amount of aerodynamic drag can be chosen, knowing that the airfoil will have the same properties.

6.1.1 Concept based on Symmetry

The first concept proposed would be similar to the 2012 solar powered vehicle, with the driver pod in the middle of the vehicle. Opposed to the previous vehicle design, the driver would be seated to the front of the vehicle, thereby aiding in the aerodynamic stability of the vehicle. Furthermore, each wheel would be covered individually by a fairing with the suspension, steering and braking systems enclosed within the fairings. This reduces the frontal area of the vehicle slightly, compared to the 2012 solar powered vehicle. Refer to Figure 46 – First concept design (symmetrical).

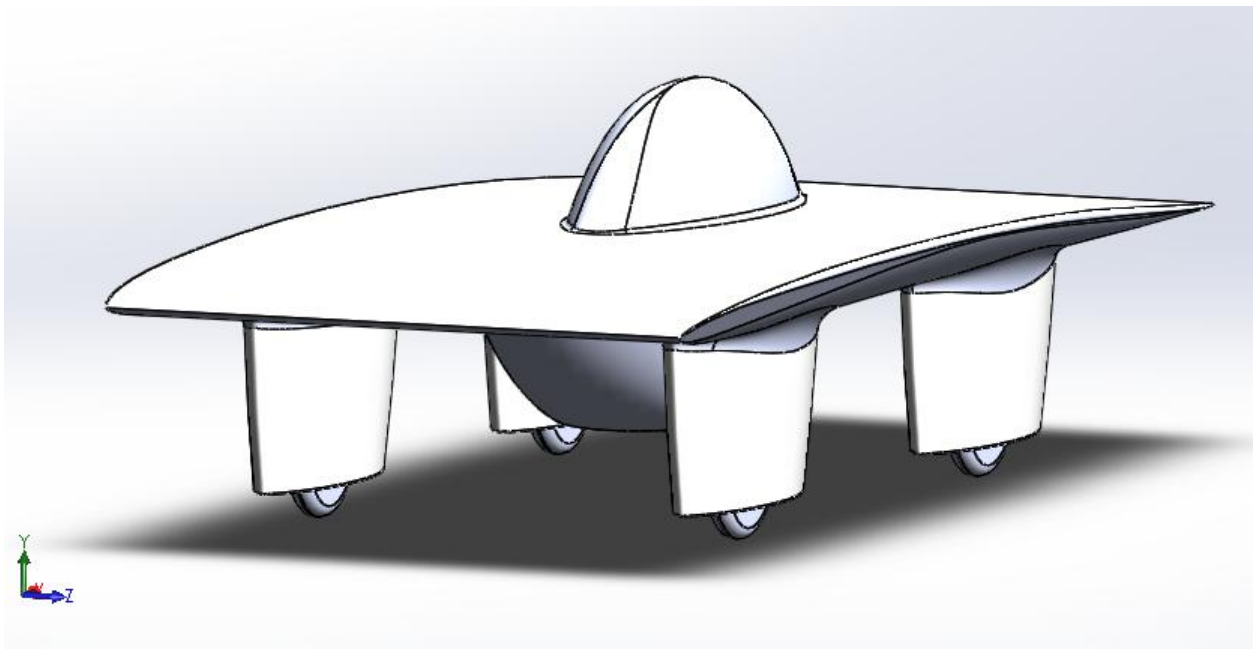


Figure 46 – First concept design (symmetrical)

As the driver is seated in the centre of the vehicle, this design results in a larger frontal area, as well as a larger wetted surface. In theory this already indicates that this design will have a larger drag coefficient when referring to Eq. 2.2. This equation states that when the velocity of the air flowing past the object or vehicle is kept constant, the frontal area and the drag force will be directly proportional. If the frontal area is increased, then the drag force will also increase proportionally.

6.1.2 Asymmetrical Concept with Long Fairings

It was discussed in the previous sub-section that having the driver seated in the centre of the vehicle will have a negative impact on the design because the frontal area of the vehicle will be too large and therefore

the drag of the vehicle will also be substantially higher. The second concept design is therefore modified, moving the driver to the right hand side of the vehicle, between the front and rear wheel. This means that the driver pod and the right hand wheel fairings are integrated into a single structure, thereby reducing the frontal area of the vehicle substantially. Once again referring to Eq. 2.2, the drag force of the vehicle will be reduced as the frontal area of the vehicle is reduced.

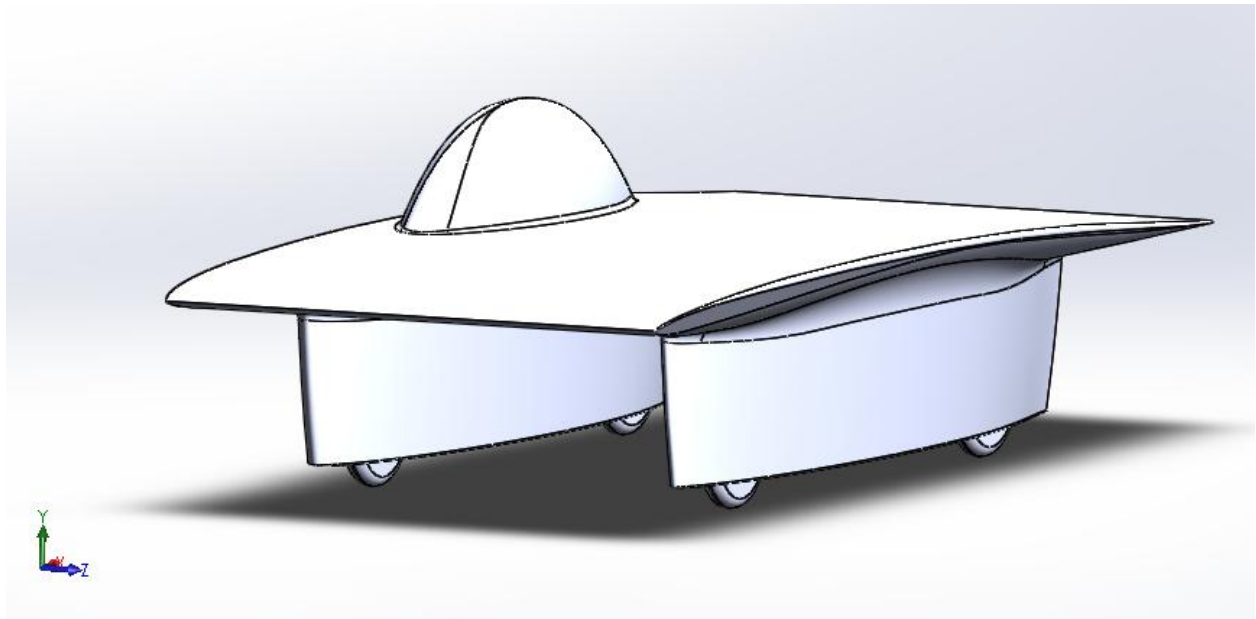


Figure 47 – Two long fairings with driver to the right hand side of the vehicle

This is the catamaran effect mentioned during the brainstorming process. The wheels on the left hand side of the vehicle would also be covered by a single, long fairing (as depicted in Figure 47), but one that is slightly narrower than the right hand side fairing (Figure 48). This basic design is also used by Nuon Solar Team, reviewed during the literature survey.

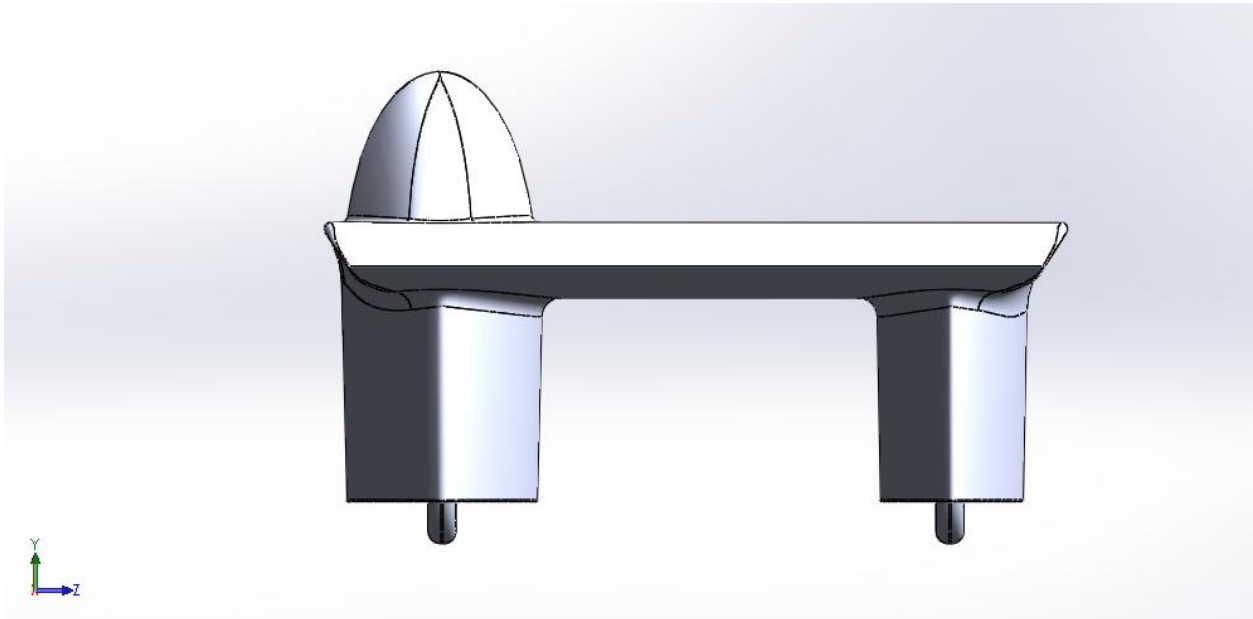


Figure 48 – Front view of the second concept

This would allow components to be installed inside the fairing in a compartment between the wheels on the left hand side. By using this space between the wheels, the main body can be narrower and thus the frontal area can be reduced even further. This means it may have a clear advantage over the first concept as the frontal area would be significantly reduced. It should, however, be noted that by utilising the space between the wheels on the left hand side of the vehicle, the centre of gravity of the vehicle will shift to the rear, even though the driver is still seated forward of the neutral point of the vehicle. This might have implications for the aerodynamic stability of the vehicle. Most of the weight should be towards the nose section of the vehicle, moving the centre of gravity as far forward from the neutral point as possible.

6.1.3 Asymmetric Design with Short Fairings

The third concept idea is based on the same principal as the preceding concept, with the driver seated in a single long fairing on the right hand side of the vehicle. The difference between the two concepts is that the third concept sees a change in the left hand fairing design. The single long fairing is replaced by two shorter fairings as in the symmetric design proposed first (Figure 49).

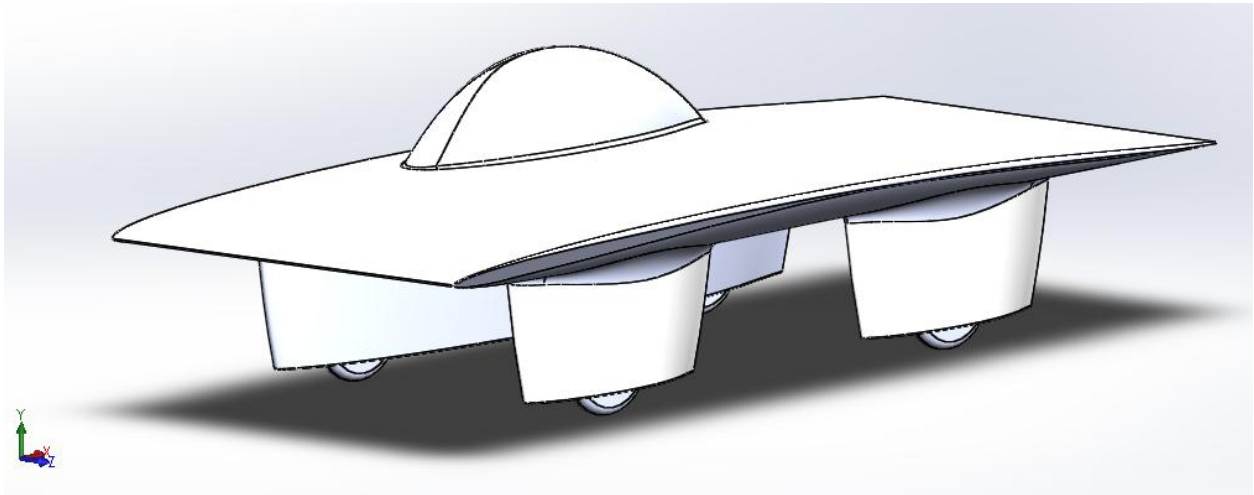


Figure 49 – Asymmetric design with three fairings

This change means that the components will have to be installed inside the main body of the vehicle as there is not enough space in the shorter fairings. It will then be possible to install the components to the front section of the vehicle and thereby moving the centre of gravity further forward from the neutral point of the vehicle. This will increase the aerodynamic stability of the vehicle, meaning it will be less likely to experience lift on the nose of the vehicle when driving. It also means that the front wheels will experience better traction and therefore the vehicle will have better road handling capabilities at higher speeds.

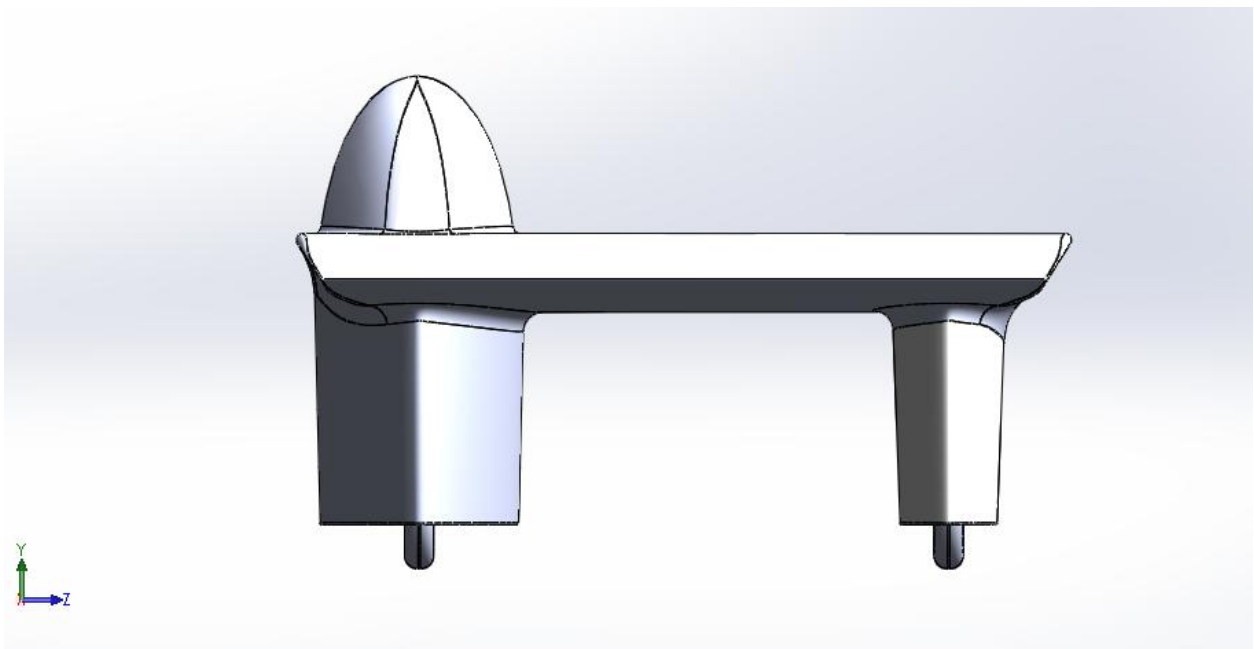


Figure 50 – Front view of the third design, showing the effect that the shorter fairings has on the frontal area

By using shorter fairings, the maximum width of these fairings will be much narrower (Refer to Figure 50) than when a single longer fairing is used. This means that the frontal area, as well as the wetted surface of the third design, will be even less than in the second concept, which should result in a further reduction of the aerodynamic drag force.

Theoretically it therefore means that the third design should be the most aerodynamic and it will therefore make sense to use this design for optimisation. This design is also similar to the design used by Tokai University.

6.2 Comparison Setup

During the validation process described in the preceding chapter, Chapter 5: 3-D Validation: HPT VeloX1, a setup and procedure was described which should provide a suitable platform on which to conduct the comparison of these concept designs. Comparing the concepts is an essential step in the design process of the vehicle. By doing an aerodynamic study about the drag force of the concept designs, it is possible to compare the different designs and choose the design with the most favourable aerodynamic properties. It should be noted that for the purpose of this dissertation, to accurately determine the aerodynamic drag of the vehicle, the transition location should be determined. The turbulent kinetic plots will therefore be used to show transition locations and compare it with one another. The drag area of the models, along with the drag coefficients, are also good indications of which concept design can be used in the detail design phase. The chosen concept will be refined to further reduce the aerodynamic drag of the vehicle.

6.3 Comparison Procedure

To attain a value for the aerodynamic drag coefficient, a force monitor was used in the STAR CCM+ simulation to measure the drag force of the vehicle body. To speed up the comparison process, a server with 64 processors were used. Equation 5. 1 is again used, as in the 3-D validation case in Section 5.2. This will give a value for the drag force (D), so that the drag area ($C_D A$) can be determined. The frontal area of the vehicle body (A) is also known and this will provide the value for the aerodynamic drag coefficient (C_D). The same settings were used as described in sections 5.1.1 and 5.1.2 of the 3-D validation procedure.

6.4 Comparison Results

Comparing the different concepts, following the process as described, yielded the results summarised in Table 5, where C1, C2 and C3 represents the three concepts discussed (respectively).

Table 5 - Conceptual design comparison

	C1	C2	C3
F	57.9	46.8	43.9
CdA	0.121	0.098	0.091
A	0.864	0.812	0.766
Cd	0.140	0.120	0.119

Referring to Table 5, it is quite clear that the least aerodynamic design would be the first concept (C1) being a symmetric design. C1 has a high drag coefficient of 0.14, largely due to the larger frontal area as well as a larger wetted surface. This means that there is more air in contact with the surface of the vehicle, thus a higher shear force. The increased number of appendages increases the interference drag between the body parts. The higher the shear force and the interference drag on the surface of the vehicle, the higher the drag force will be and essentially the drag coefficient.

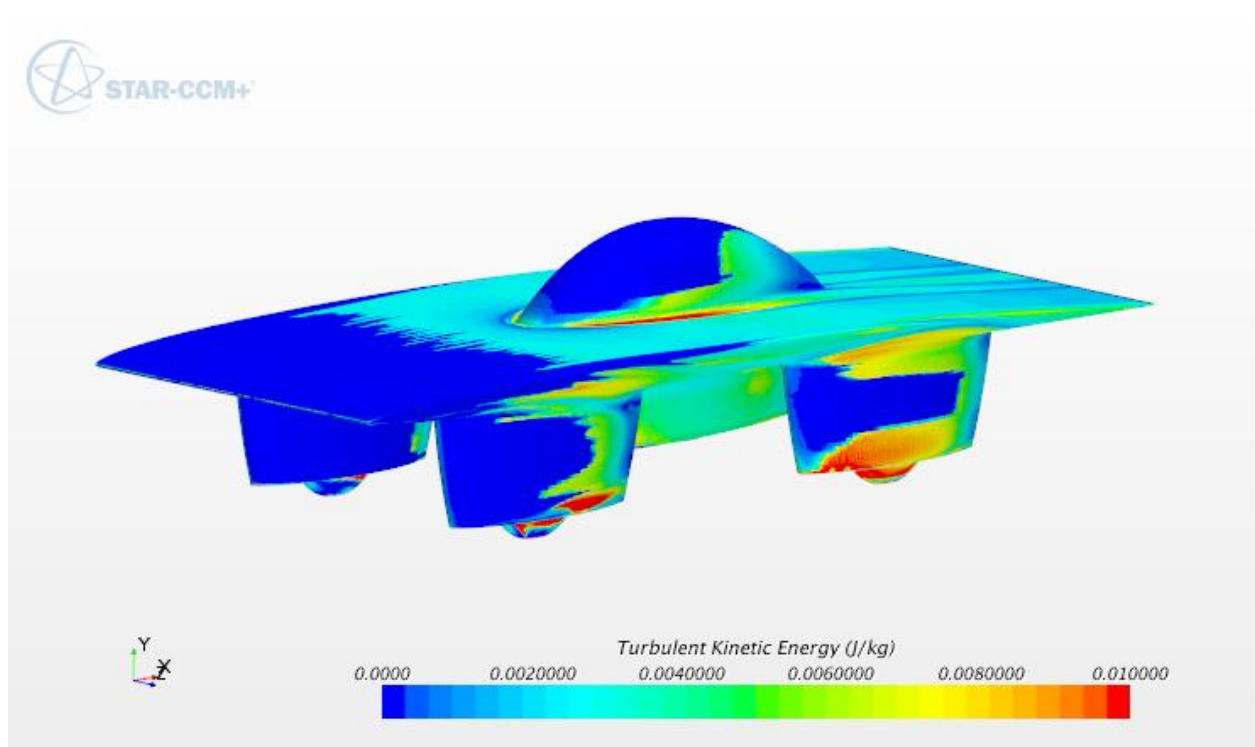


Figure 51 - Turbulent kinetic energy of the first concept

Referring to Figure 51, it is observed that C1 experiences substantial laminar flow on the front fairings. There is still laminar flow on the rear fairings, but increased turbulent flow is observed. When the vehicle is viewed from a lower angle (Figure 52), it is clear why the drag force of the vehicle is so much higher in than the other concepts.

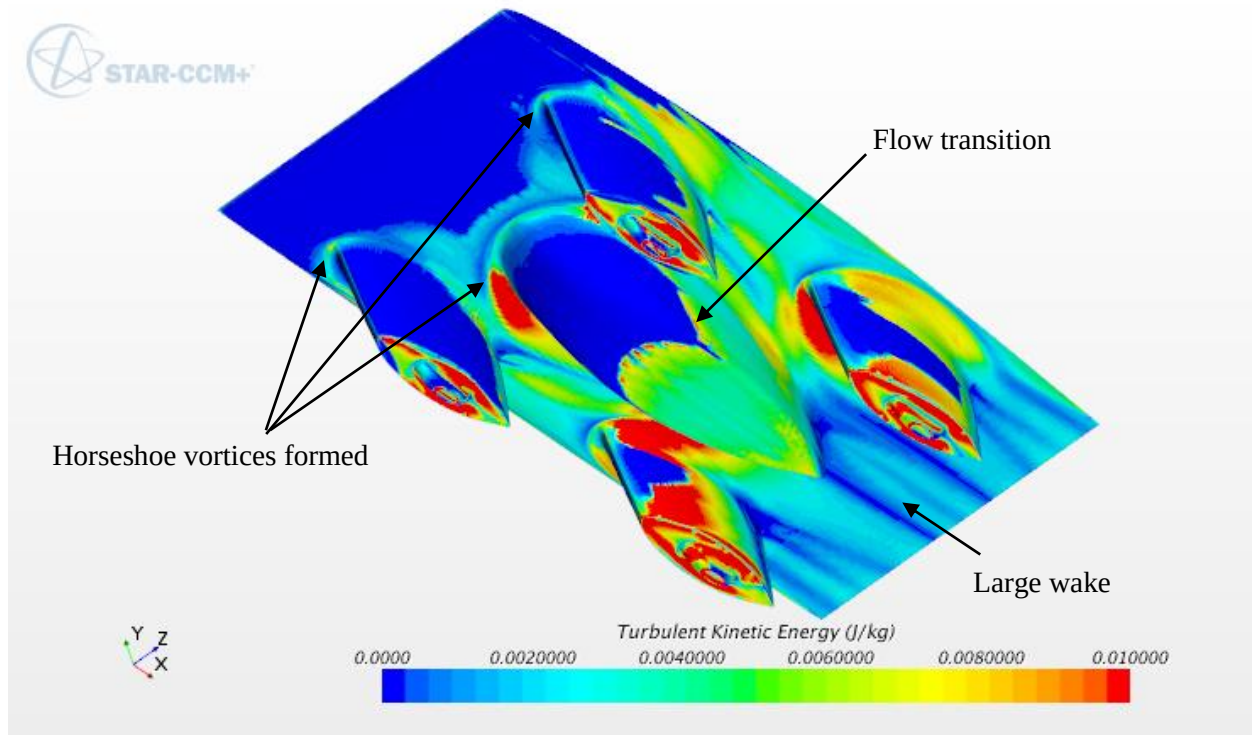


Figure 52 - Bottom view of the symmetric concept vehicle

The driver pod increases the wetted surface substantially and produces a very large turbulent wake due to large vortices that formed at the pod junction. The rear fairings also experience increased turbulent flow on the inner sides next to the driver pod, as this narrower section increases the velocity of the air and thereby increases turbulence. The pod also shows quick transition of laminar flow.

The second concept design (C2) shows a substantial reduction of almost 20% in the drag force. This is because of the reduction in the frontal area of the vehicle. Other factors contributing to this large reduction is the smaller wetted surface as well as fewer appendages, reducing the number of junctions. Fewer junctions mean less interference drag. The drag coefficient of this model is 0.12, which is about 16% less than the first concept and this is because of a reduction in the frontal area of about 6%.

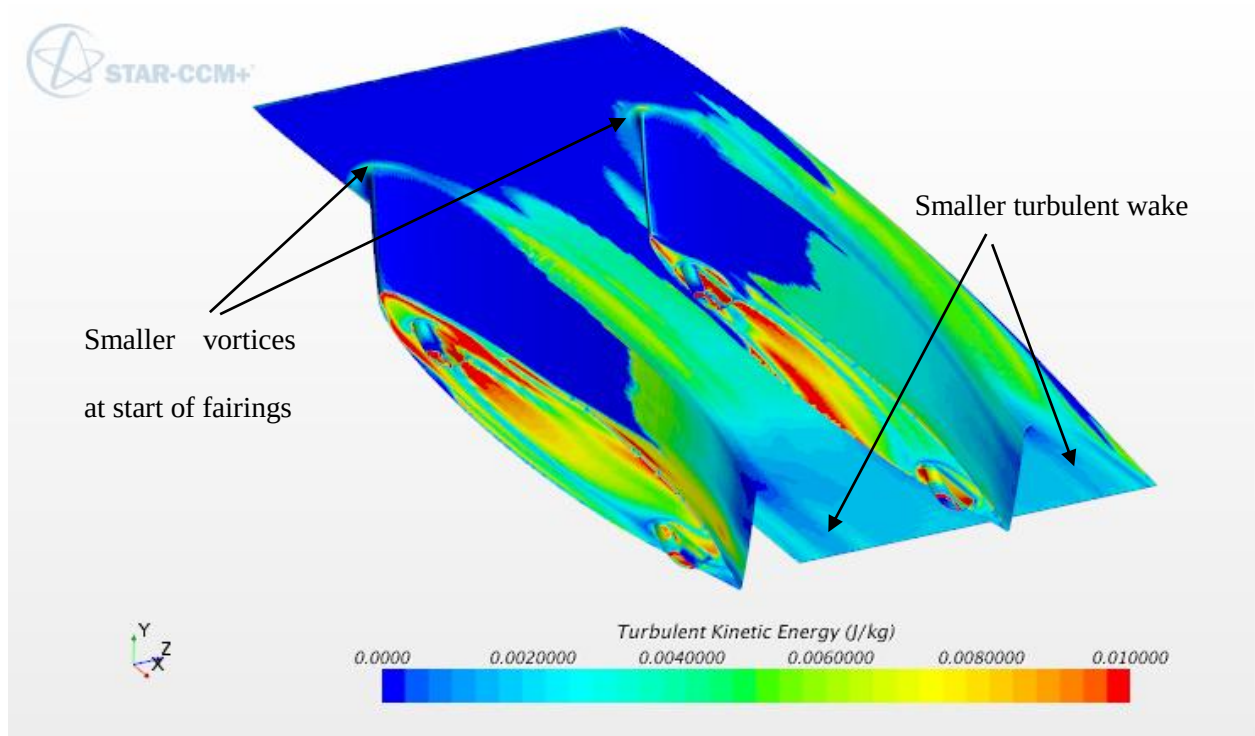


Figure 53 - Turbulent kinetic energy of the belly section of the second concept

In Figure 53 it can be observed that the longer fairing experiences laminar flow to about two thirds of the fairing length on the outside and to about half of the fairing length on the inner side, helping to reduce the drag of the vehicle further. The effect of the horseshoe vortices formed by the fairings are much smaller than in the previous concept. This means smaller wakes will be created behind the fairings, all contributing to the lower drag force experienced with this concept.

As discussed, the third concept design (C3) is expected to be the most aerodynamic as the wetted surface and the frontal area is further reduced from C2. C3 has a frontal area which is about 5.7% smaller than C2 but this relates to only a 0.7% reduction in the drag coefficient to a value of 0.119.

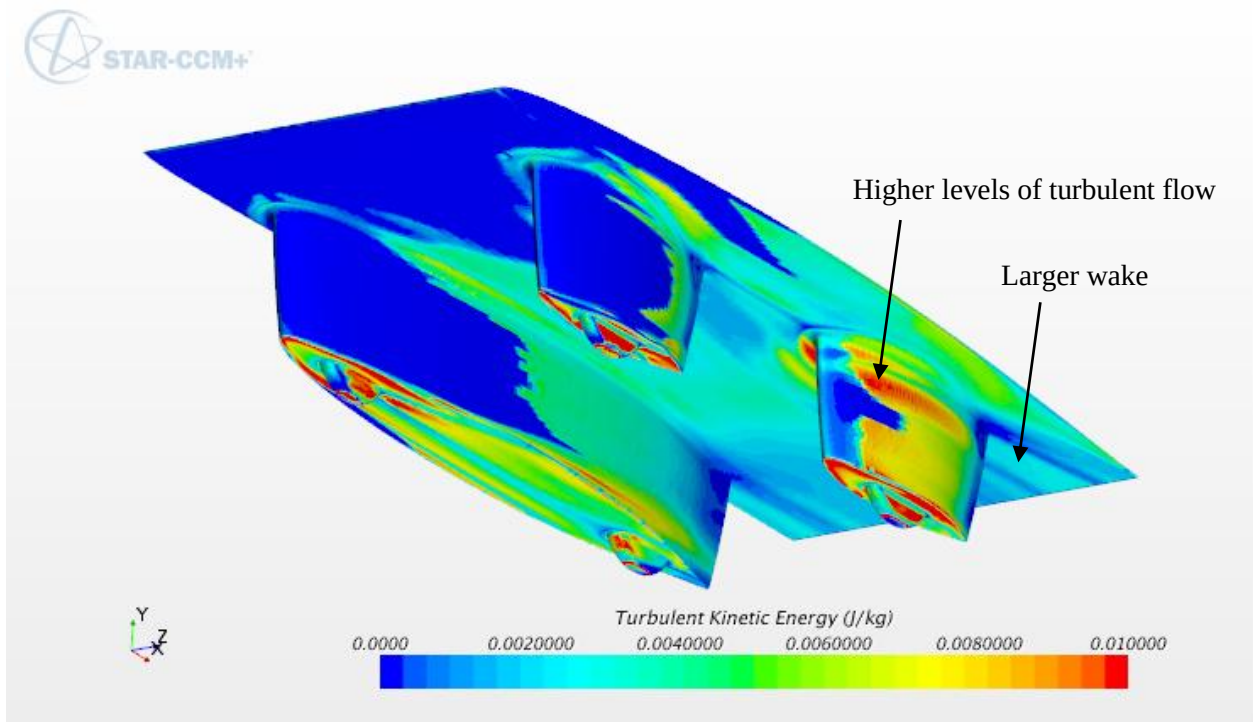


Figure 54 - Turbulent kinetic energy of the belly section of the third concept

This small reduction is attributed to the high levels of turbulent energy experienced on the rear fairing and also a larger wake behind this same fairing. This means that, even though the frontal area of the vehicle is reduced, the drag area of the vehicle is larger and this means larger areas of turbulent flow. The vortices created at the start of the fairings, do not differ much from C2.

6.5 Conclusion

Looking at the results, it can be argued that the small difference between C2 and C3 means that either concept can be modified and improved on in the detail design phase. This is true but because C3 has a much lower frontal area, it means that if the interference drag and turbulence on the rear fairing is improved, the drag can potentially be reduced more than when the same procedures are applied to C2. This increases the advantages of the third concept design. Keeping in mind the descriptions of the concepts regarding the placement of internal components, it was argued that C3 holds even greater advantages in that the aerodynamic stability potential of this concept is the greatest. This, coupled with the fact that it did have the lowest drag of all the concepts tested, means that the third concept holds the greatest potential for improvement and was selected for the detail design phase.

Chapter 7: Detail Design

After the baseline had been determined, the different features of the design which were discussed, had to be optimised in order to reduce the drag force of the vehicle. These features included variations in the airfoil profiles used for the fairings, the canopy and the body of the vehicle. Individual simulations of the different components could determine the more aerodynamic shapes to be integrated with the final vehicle body. This is done while constantly considering the competition regulations.

7.1 Component Analysis

7.1.1 Analysis Procedure

In order to compare the different components with each other, either 2-D simulations or 3-D simulations were used. As the design of the different components was based on airfoil profiles, the first step in the comparison was to evaluate different airfoil profiles in 2-D using XFLR5. The airfoil profiles were chosen such that the transition location was as far to the rear as possible. This theoretically implies that the laminar flow on the 3-D profile will be extended further.

For the 3-D simulation the same simulation setup as discussed in Chapter 4 was applied to the components in STAR-CCM+. The mesh and solver configurations were therefore already validated and it was assumed that the transition locations on the individual components would give a fair representation of the laminar flow for the vehicle. The components were compared with each other by evaluating the drag coefficients, as well as studying the transition locations on the turbulent kinetic energy plots. On these plots, the transition location is where the colour in the figure abruptly changes, from a dark shade (dark blue) to a lighter shade (yellow or light blue).

The results of these comparisons are described in the following sub-sections.

7.1.2 Main Vehicle Body

The design of the main body of the vehicle was based on a modified version of the JS1 sailplane wing airfoil (Figure 55). This is testament that it was an excellent profile to be used to enhance the aerodynamic properties of the vehicle.

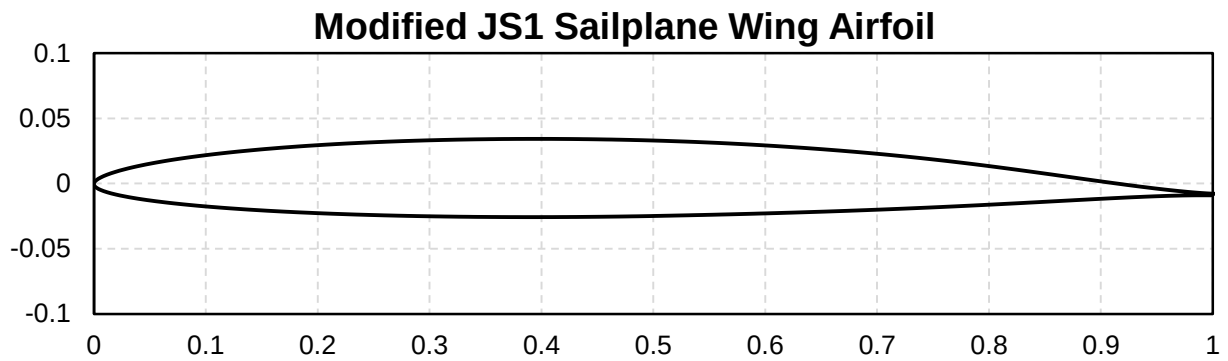


Figure 55 – Modified JS1 sailplane wing profile used for main body

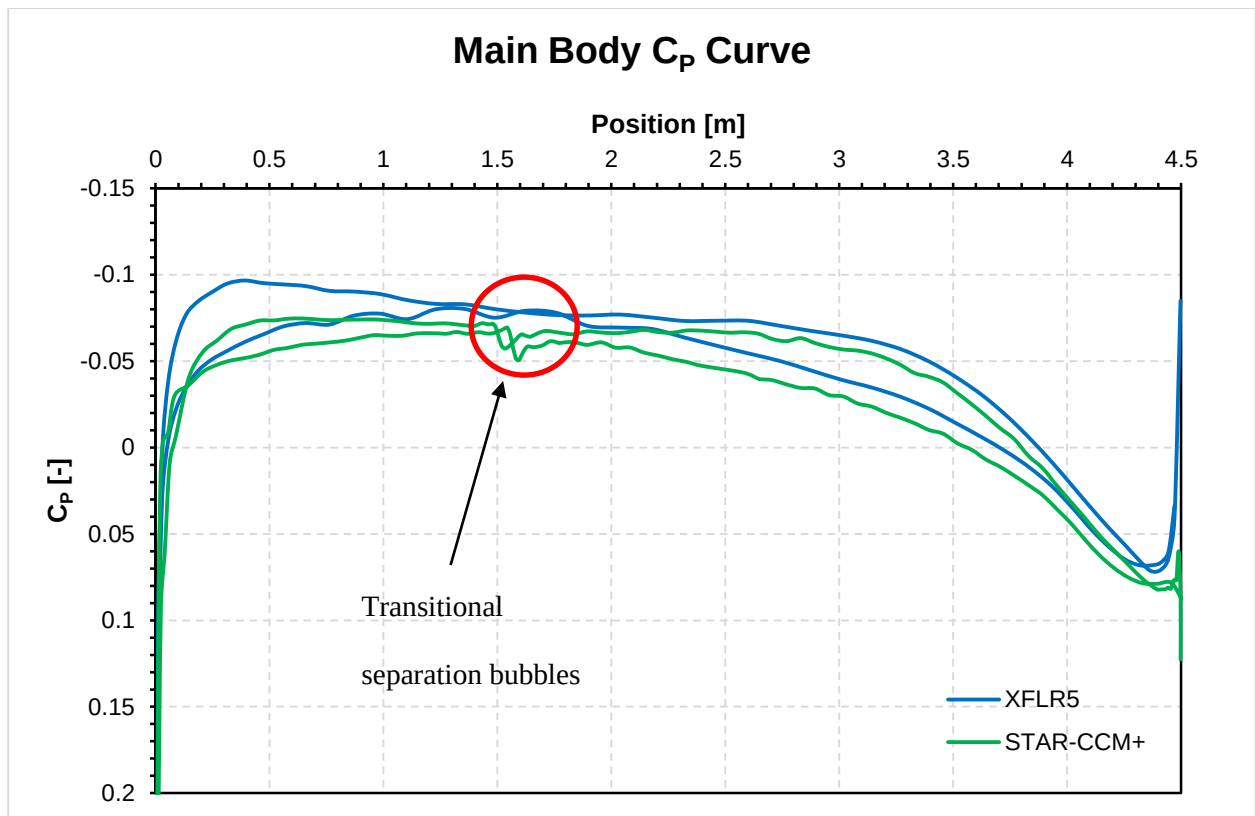


Figure 56 - Pressure coefficient curves of main body profile

Referring to the above figure (Figure 56), the XFLR5 3-D results at a 0° angle of attack shows that the body experiences transitional separation bubbles at about 35% of the chord length on the upper surface. The laminar flow extends a bit further on the lower surface, to about 50% chord length. These separation bubbles cause the laminar flow to transition. The transition locations in the STAR-CCM+ simulation is calculated to be about 35% chord for the upper and lower surfaces. This compares well with the XFLR5 results.

Referring to the STAR-CCM+ 3-D pressure coefficient plot of the main body in Figure 56, the same pressure gradient is perceived, with slightly higher pressure coefficients. The minimum pressure coefficient values in the two cases differ by about 20%. The minimum value in the case of the STAR-CCM+ results is about -0.08 compared with almost -0.1 in the XFLR5 results.

The peak in the pressure coefficient value at the trailing edge of the 3-D case, is a result of the trailing edge not having a knife edge, but a thickness of 5 mm. The solution, therefore, does not completely converge and the spikes in the above figure is observed. Regarding the stability of the airfoil, the pitching moment coefficient (C_m) is -0.006, calculated by XFLR5. This differs greatly from the value calculated in STAR-CCM+, which has a value of -0.018 for C_m . This difference might be attributed to calculation methods used by the two codes, but the more probable reason might be the geometric differences. The body used in STAR-CCM+ has rounded sides, while the body used in XFLR5 has flat sides. The vortices created on the body sides might alter the moment forces on the body differently. As discussed in the literature, the pitching moment must be negative in order to counteract the effect of the lift force. This means that the airfoil will be statically stable.

This airfoil was made into a 3-D part (Figure 57), with large rounded edges to improve the ease of manufacture, as well as providing a large flat surface where internal components of the vehicle, such as batteries, could be installed.

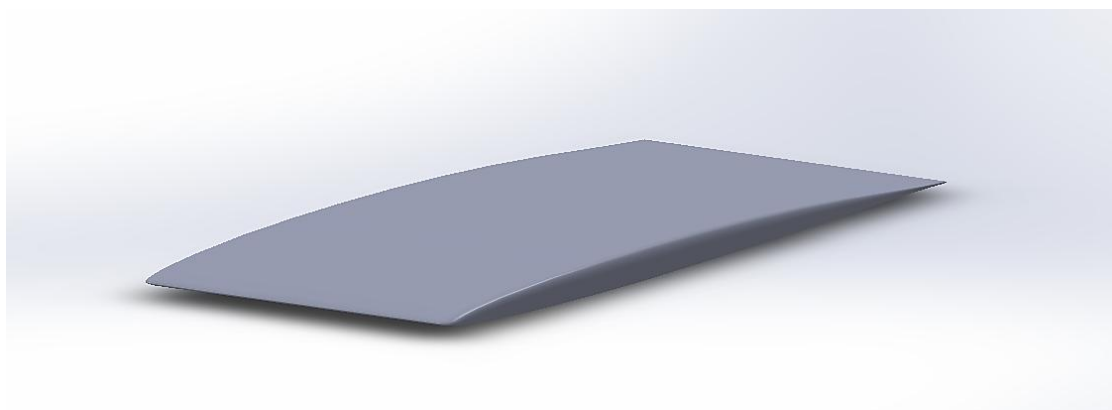


Figure 57 - Main body of the vehicle

Note the rounded leading-edge of the main body in Figure 58. This has a large radius which not only improves the manufacture of the moulds and the actual parts, but helps to improve the aerodynamics by

avoiding sharp edges, as was the case of the 2012 vehicle. If sharp edges were present on the leading edge, it would cause the flow to separate from the surface. The rounded surface has the effect of easing the flow over the body of the vehicle. This premature separation would result in a higher drag force and therefore negatively affect the aerodynamics. This means that the laminar flow would extend past the leading-edge (Figure 59). In the case of sharp corners on the leading-edge of the body, flow would have transitioned and minimal laminar flow would have been possible on the sides of the main body.

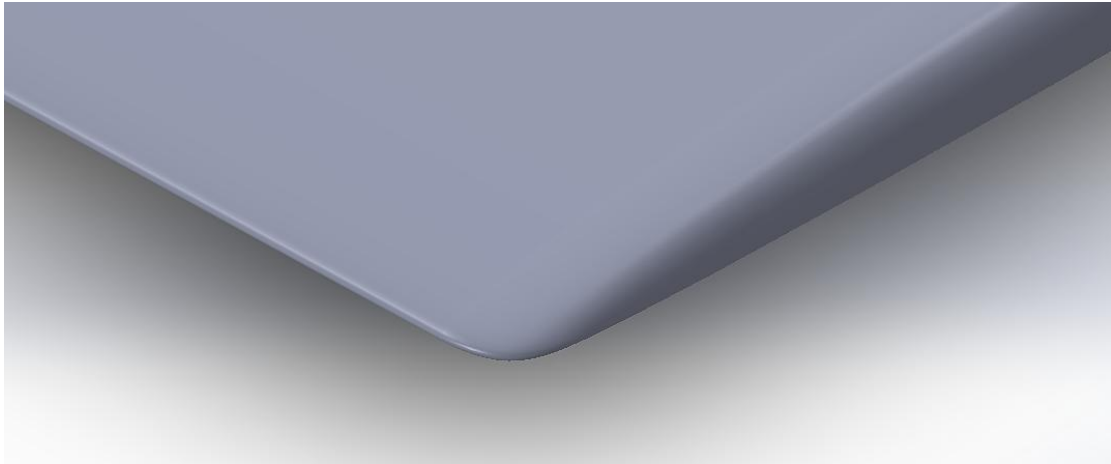


Figure 58 - Rounded leading-edge of the main body

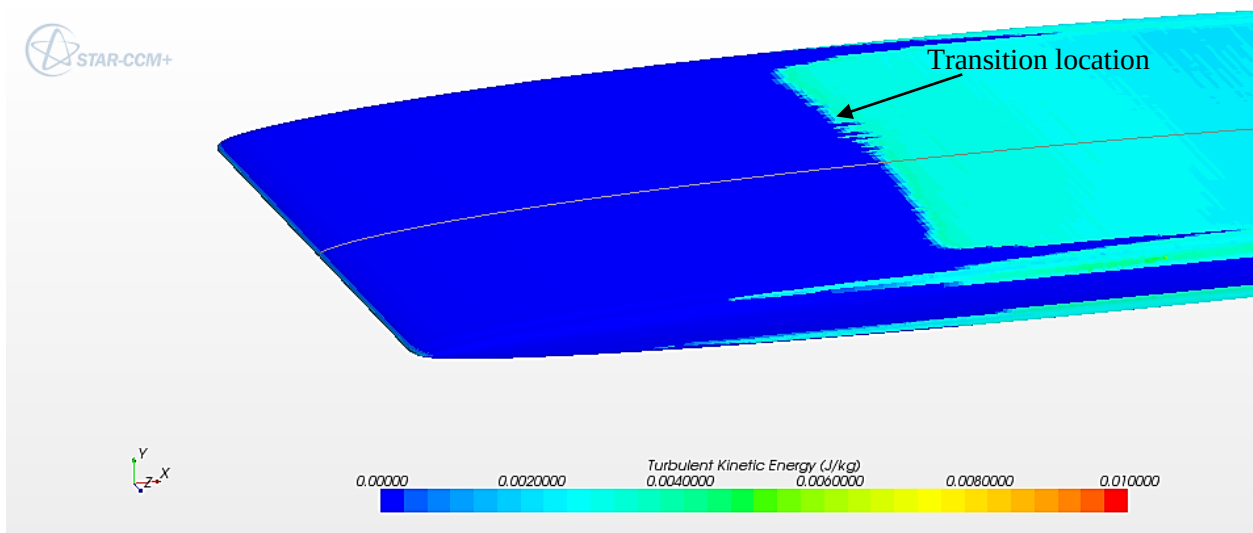


Figure 59 - Laminar flow on leading-edge of main body

7.1.3 Fairing Profile Selection

It is easier to select an airfoil from a selection of profiles that is known to have laminar flow and has already been proven in a 3-D case. In this case the airfoil was a modified version of the tail wing of the JS1

sailplane, namely the RUD20 airfoil (Figure 60). This sailplane has been proven to be the best during multiple world championships, and the laminar flow properties are favourable for the application of wheel fairings.

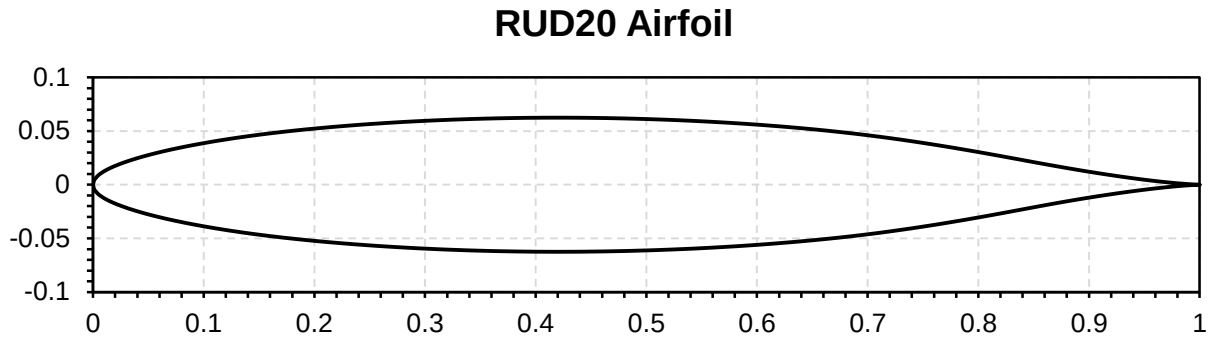


Figure 60 - RUD20 airfoil

For the RUD20 airfoil, the point of highest pressure is near 50% chord length, which is favourable. This indicates that the laminar flow will also extend along the length of the profile, with the transition location at approximately 45.8% of the chord length. The RUD20 airfoil has a drag coefficient value (C_d) of 0.005.

This airfoil was compared with other aerofoils exhibiting the same characteristics. These aerofoils had to be symmetric, so that the wheels of the vehicle would be centred in the fairing. The trailing section could not draw too sharply into a narrow trailing edge, because this would limit the space available in the fairing for the wheels, suspension, etc.

The YS900 airfoil, shown in Figure 61, was compared with the RUD20 airfoil because its maximum thickness was further back at 51.91%. YS900 also had a stronger pressure recovery which would make it better suited when two fairings were in tandem and the rear fairing was in the slipstream of the front fairing.

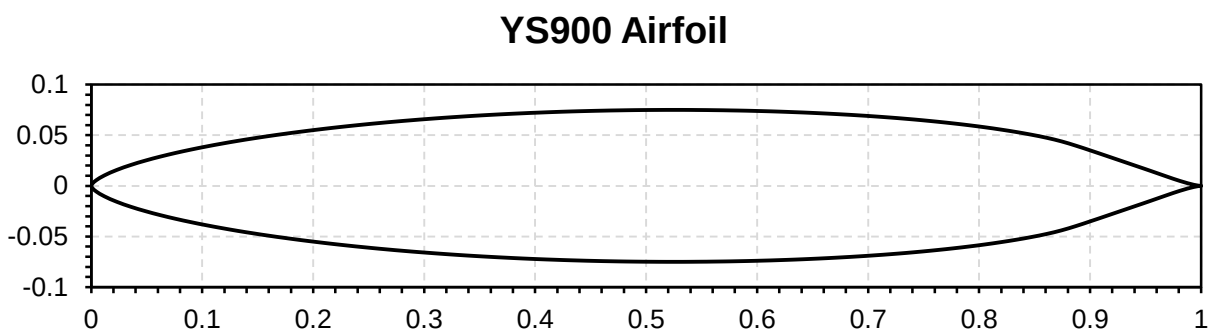


Figure 61 - YS900 airfoil

The transition location of the YS900 airfoil is much further back than the RUD20 airfoil, at 88.5% of the chord length. This is extremely favourable and it also has a C_d value of only 0.003. These characteristics made the YS900 airfoil better suited to the fairings design.

3-D CAD parts were made using these profiles, taking into account the points discussed in section 2.4, namely the inclination angle of less than 10° . These 3-D parts were integrated with the rough 3-D concept model of the vehicle selected in the previous chapter and the model of the vehicle was re-evaluated in STAR-CCM+ and compared to each other. The RUD20 fairing had a drag force value of 73.32 N. The YS900 fairing, as expected, had a lower drag force value of 72.3 N due to the extended laminar flow, albeit only marginally.

Another airfoil with a favourable profile is the NACA 16015 airfoil (Figure 62). This airfoil is closer to the RUD20 airfoil although the trailing section does not draw into thin point which is better for the fairing design.

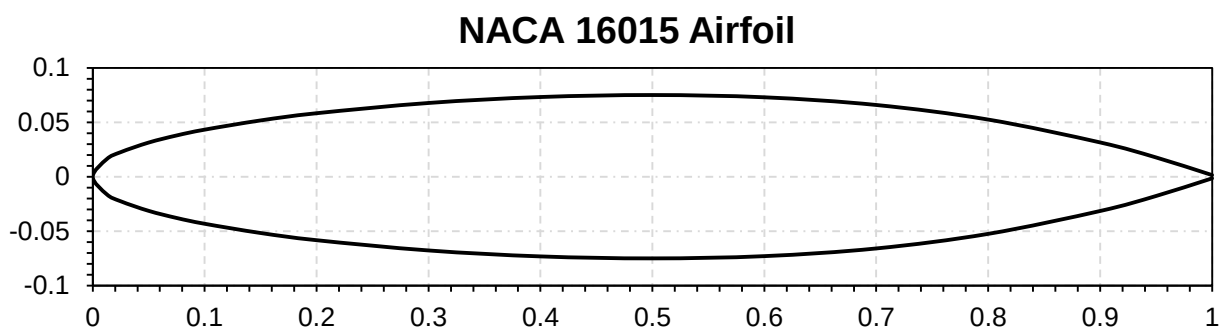


Figure 62 - NACA 16015 airfoil

The thickest section of the wheel is at 50% chord length. As the wheel will usually be centred in the fairing, it makes sense from a manufacturing standpoint that the fairing should be thickest at 50% chord length. However, the transition characteristic of this airfoil was not favourable.

To thicken the airfoil at the rear for reasons stated, the NACA 16015 was integrated with the YS900 airfoil and shown in Figure 63. This resulted in an airfoil with a C_d value of 0.003 and a transition location of about 75% of the chord length. The pressure recovery was not as drastic as that of the YS900 airfoil, but stronger than the NACA 16015.

50:50 NACA 16015 and YS900 Combination Airfoil

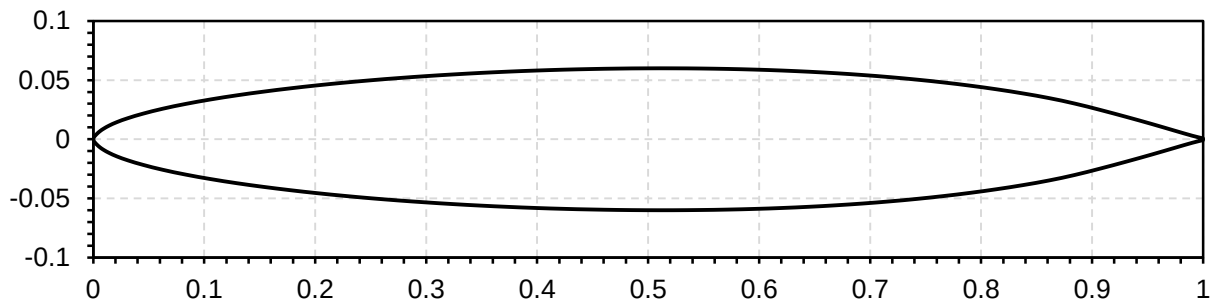


Figure 63 - NACA 16015 and YS900 airfoils combination

NACA 16015 and YS900 Combination Airfoil C_p Curve

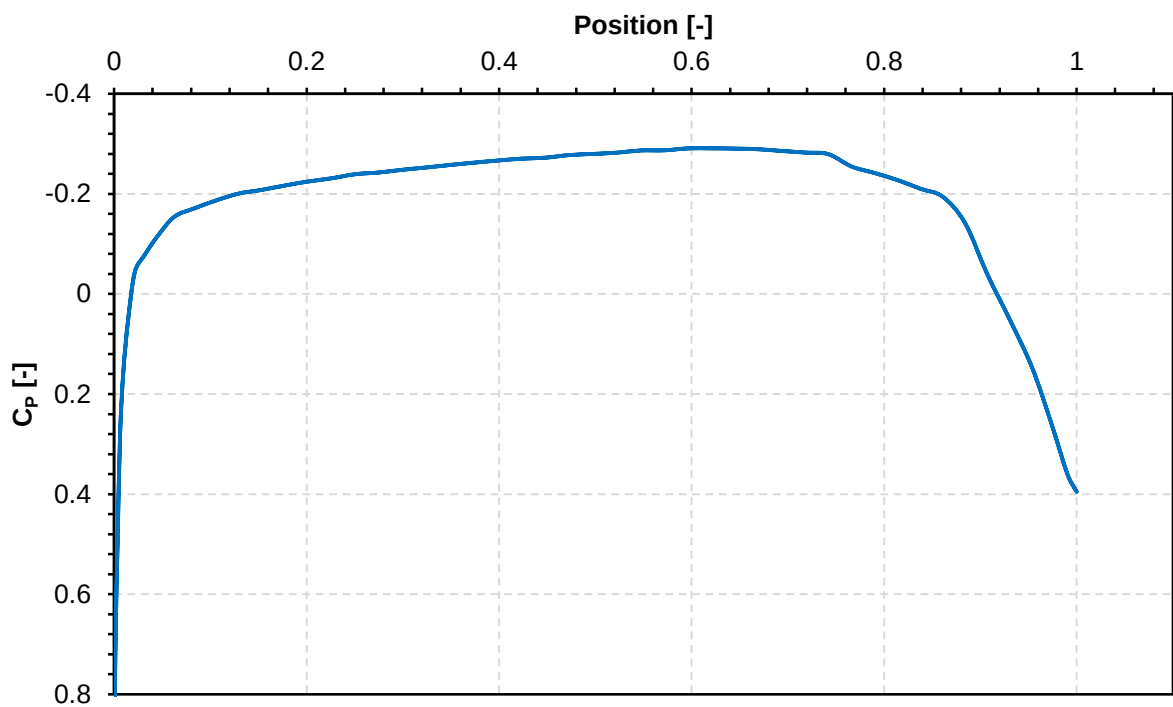


Figure 64 - Pressure distribution along the chord length of the NACA 16015 YS900 airfoil

The NACA 16015 YS900 airfoil exhibited poorer transition in the 2-D case, but when the 3-D CAD fairing was made and evaluated using CFD, the overall drag force value for the fairing was 71 N. This was better than any of the other fairings evaluated due to the favourable pressure gradient up to about 70% chord length (Figure 64). The drag force value for this fairing was reduced even further by reducing the width of the fairings to the minimum width, thereby reducing the frontal area, which was one of the design parameters. With this modification the drag force value of the vehicle was 64 N.

7.1.4 Fairing Design

As mentioned in the previous section, the rear wheel fairing is in the slipstream of the front fairing. The front fairing would ideally have a constant pressure gain with a flatter profile. Refer to Figure 64. The rear fairing should have a much steeper gradient with a very sharp pressure recovery. Theoretically this should enhance the laminar flow on the rear fairing.

3-D CAD configurations of the fairings were made and evaluated using CFD to determine if this was the case.

Configuration 1

The first configuration had the front fairing as the NACA 16015-YS900 combination fairing. However, due to the YS900 fairing having a steeper pressure recovery, it was selected as the rear fairing. This configuration is depicted in Figure 65.

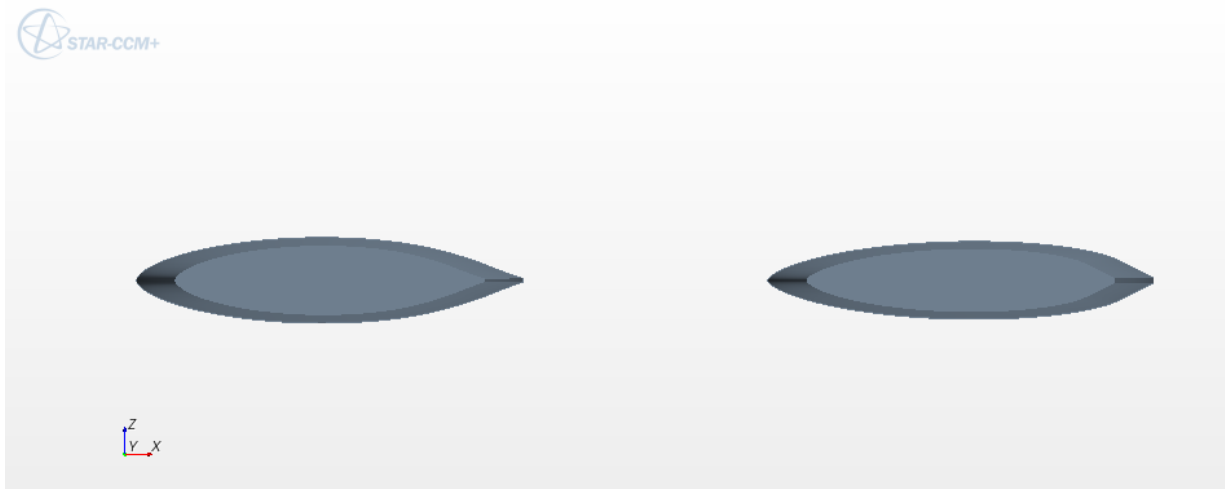


Figure 65 – Fairing Configuration 1

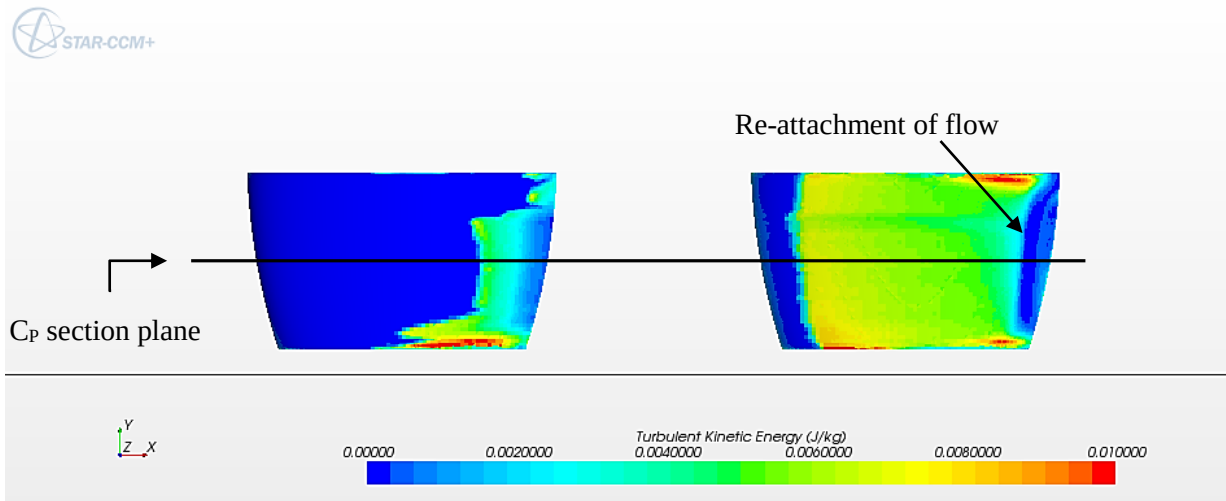


Figure 66 - Transition locations of flow on fairing Configuration 1

This configuration had a drag force of 18.24 N. Even with the sharper leading-edge and sharp pressure recovery of the YS900 airfoil in the rear, the flow transitioned relatively quickly. Figure 66 shows the turbulent kinetic energy plots and the transition locations on the fairings. The reason for the poor laminar flow on the rear fairing was because the fairing is already in a turbulent flow region.

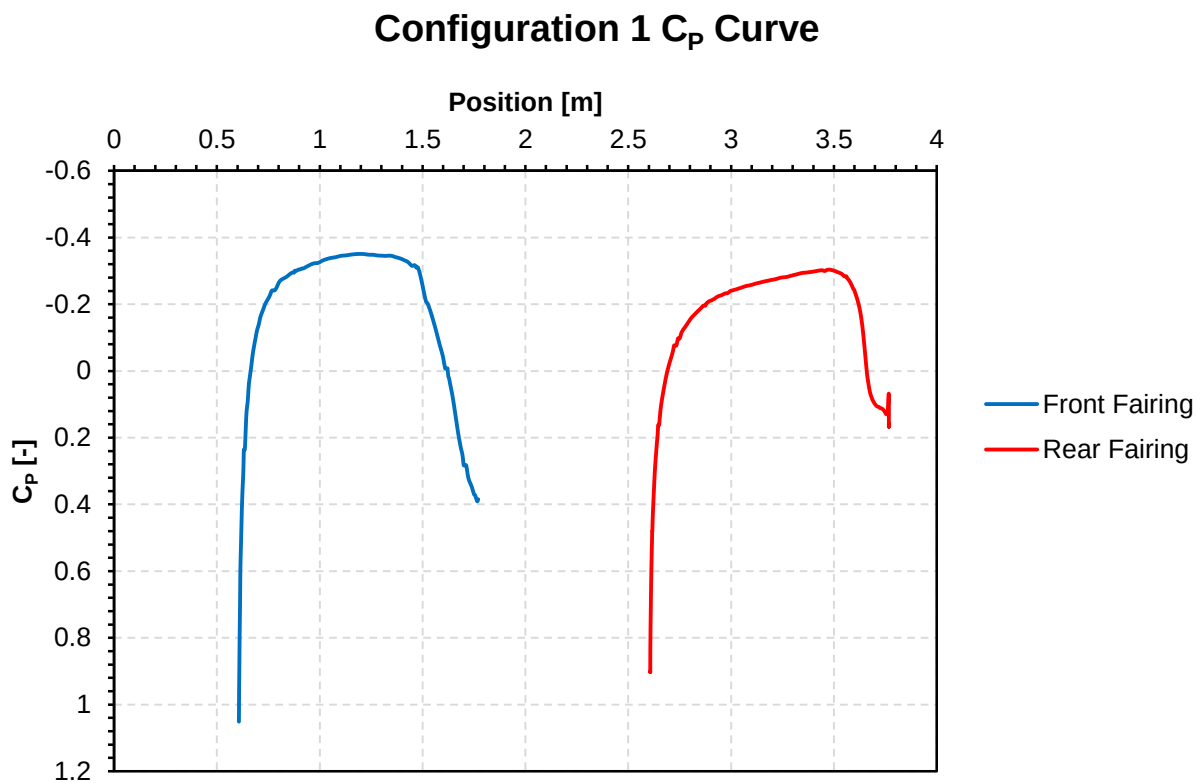


Figure 67- Pressure coefficient plot of Configuration 1

It is clear that, referring to the turbulent kinetic energy plots in Figure 66, the flow re-attaches in the pressure recovery region of the rear fairing. This is indicated by the darker coloured sections on the trailing edge. Even though this is the case, looking at the pressure coefficient plots in Figure 67, the pressure recovered is not all that satisfactory. Therefore, a larger wake will be present behind the rear fairing. The section plane from which the pressure coefficient data is extracted, is indicated by the horizontal line in the above figure.

Configuration 2

To test if a much sharper leading-edge and an even stronger pressure recovery for the rear fairing would further enhance the laminar flow on the rear fairing, the following configuration was made, while keeping the front fairing the same as in Configuration 1. The front fairing was kept the same because it showed good laminar flow in configuration 1, so there was no reason to substitute it at this time.

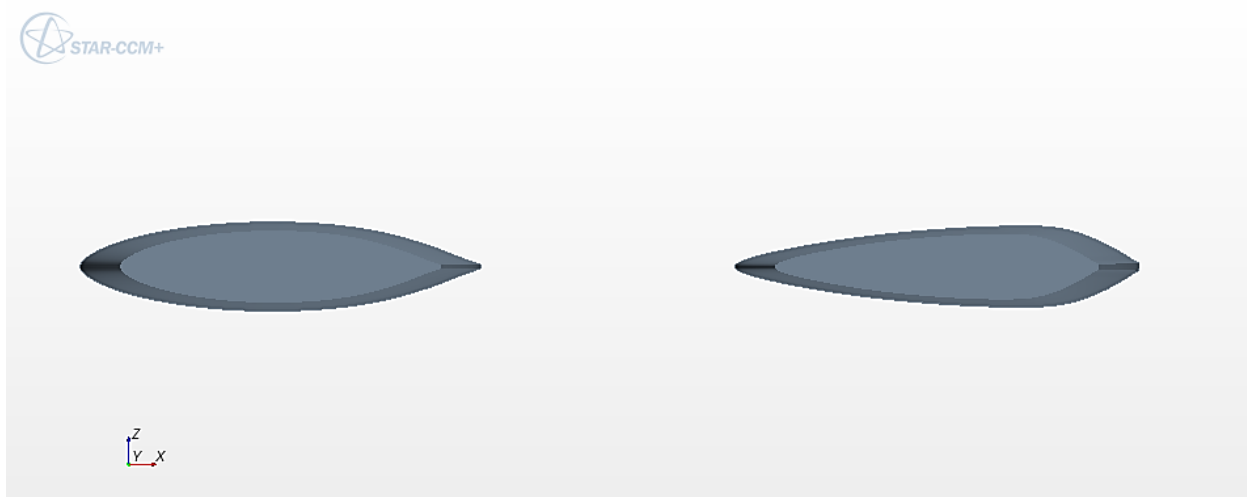


Figure 68 – Fairing Configuration 2

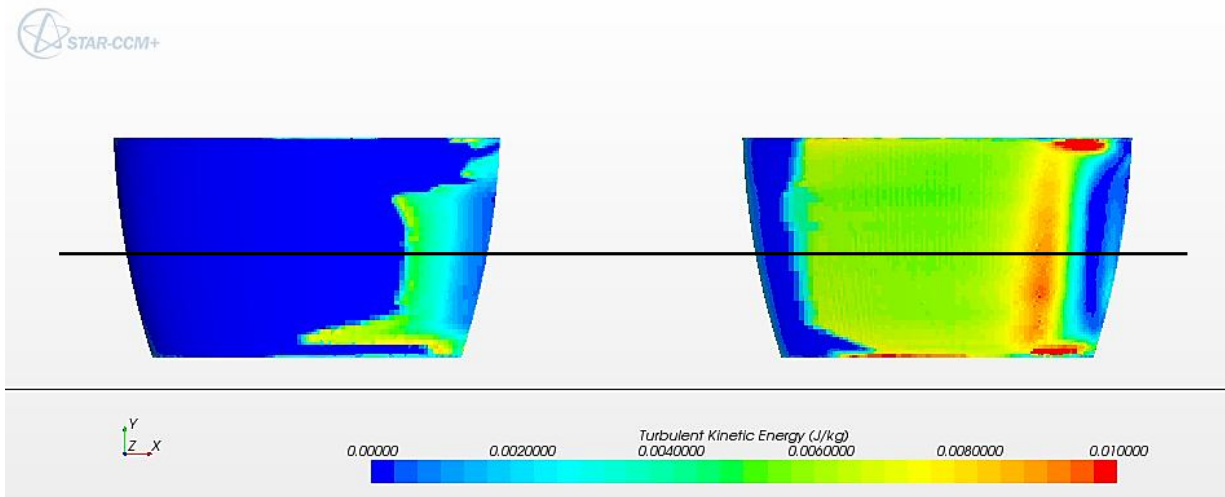


Figure 69 - Transition locations of flow on fairing Configuration 2

This configuration (Figure 68), however, had a higher drag value at 19.44 N due to the higher levels of turbulent kinetic energy on the rear fairing (visible on the turbulent kinetic plot in Figure 69 as red). This was an indication that the sharper leading-edge did not promote much laminar flow, possibly again due to the already turbulent flow. The rear fairing was also quite close to the front fairing, meaning that the flow did not have enough time to re-laminarise before reaching the rear fairing. This configuration is not favourable; not only due to the higher drag value, but also because of the limited space in the rear fairing for components such as the suspension and the braking mechanism on the rear wheel.

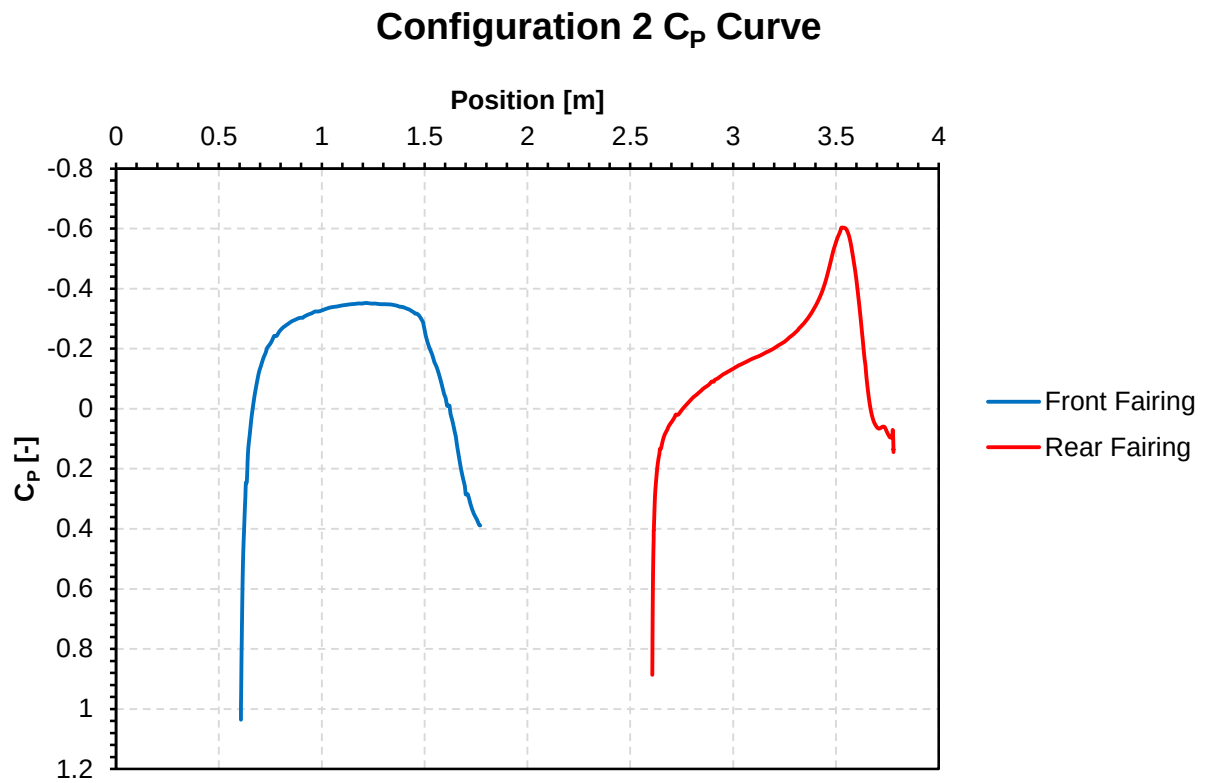


Figure 70 - Pressure coefficient plots of configuration 2

Once again one can refer to the pressure coefficient plots in Figure 70. The rear fairing shows a very steep positive pressure gradient due to the sharper leading-edge. This would be favourable in laminar flow regions. It also has a very steep pressure recovery region, but the pressure recovered is less than the previous configuration. This means it will leave an even larger wake, which could be the reason for the higher drag force. This makes it less favourable, adding to the reasons already mentioned.

Configuration 3

The third configuration had the same design for the front and the rear fairings, namely the NACA 16015-YS900 combination fairings (Figure 71). These fairings had a drag force of 18.45 N, which, even though it was slightly more than the first configuration discussed, was chosen as the configuration for the vehicle.

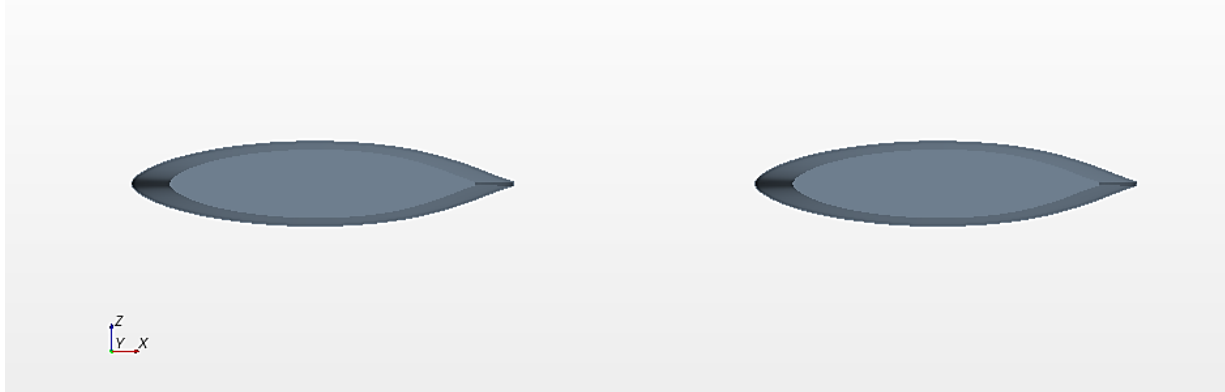


Figure 71 – Fairing Configuration 3

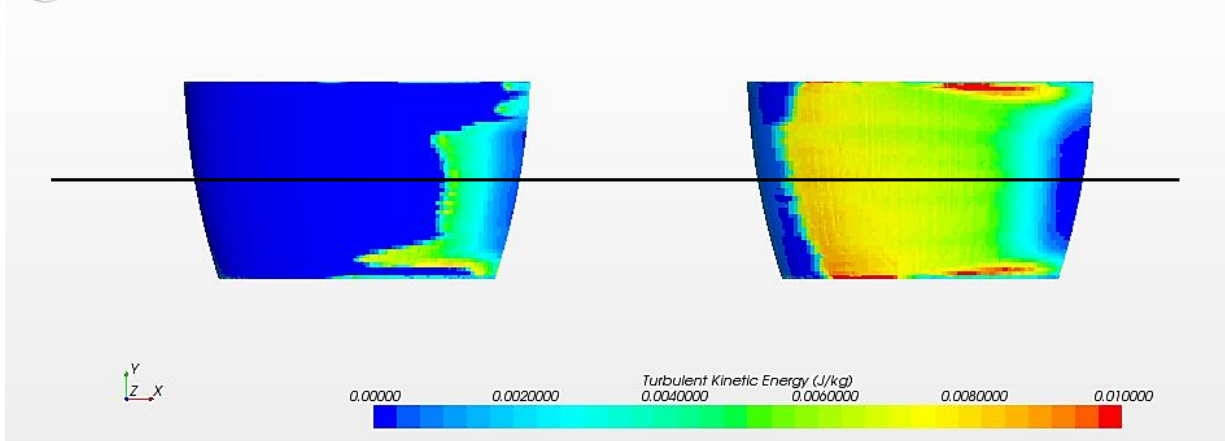


Figure 72 - Transition locations of flow on fairing Configuration 3

The reason for this is that the difference is marginal. Configuration 3 has a drag value of only 1.15% more than Configuration 1. Furthermore, Configuration 3 has more advantages in that it simplifies the manufacturing process. Instead of having to manufacture two moulds for the two different fairings, only one mould has to be manufactured which could be used twice. Of the three configurations, the third configuration has the smallest laminar flow regions on the rear fairing. The turbulent flow, however, also has the lowest turbulent kinetic energy levels as can be observed in Figure 72. This is possibly why it only shows a slight increase in the drag value.

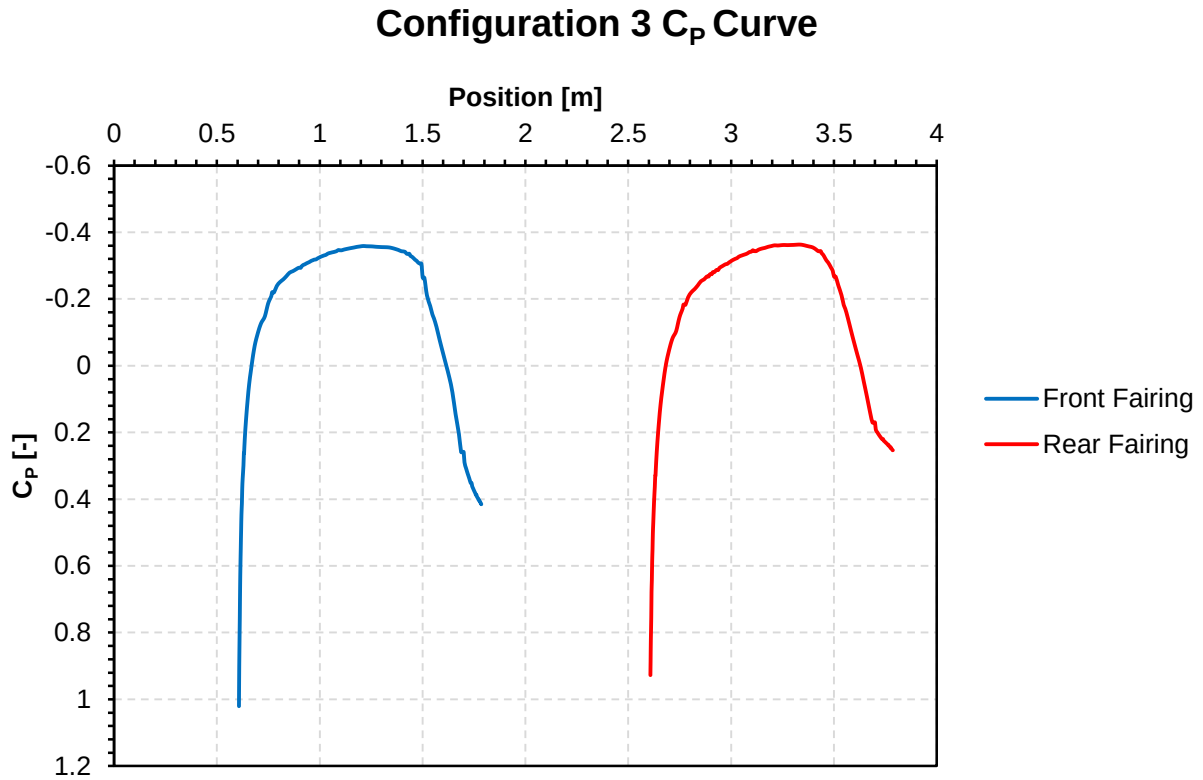


Figure 73 - Pressure coefficient plots of configuration 3

Referring to the pressure coefficient plots in the above figure (Figure 73), an assumption can be made that the pressure recovered is the greatest of the three configurations. This also aids in reducing the overall drag of the fairings and means that this configuration will have the smallest wake behind the rear fairing.

The frontal area of Configuration 1 and Configuration 2 is the same as the front fairing is the widest. There will also be more space in the rear fairings of the third configuration for the suspension system and the wheel and its braking mechanism. This third configuration was therefore selected for the preliminary detail design and will be improved during the various revisions to follow.

7.1.5 Canopy Design

Three different canopy configurations were evaluated and compared. The different canopy configurations were based on different symmetric airfoil profiles, as was done with the selection process of the fairings. The canopy should allow enough space for the driver to have unrestricted movement. However, the sections not utilized by the driver should not be too wide. This is to provide more space for the solar arrays.

According to the competition regulations the driver should be able to move 45° forward from a vertical sitting position and 25° to the rear, without touching the canopy.

Canopy 1

The first canopy design was based on the design of the RUD20 airfoil. The aerodynamic properties of this airfoil were discussed in the previous section and should therefore have favourable characteristics for the design of the canopy.

When referring to the turbulent kinetic energy plot in Figure 74, it is clear that this canopy design shows favourable laminar flow which extends to about 50% of the canopy length before transitioning to turbulent flow. The laminar section at the bottom of the profile is disregarded because it does not have an influence on the final selection criteria. This section will be cut from the canopy when integration with the vehicle takes place. This canopy has an aerodynamic drag force of 10.83 N

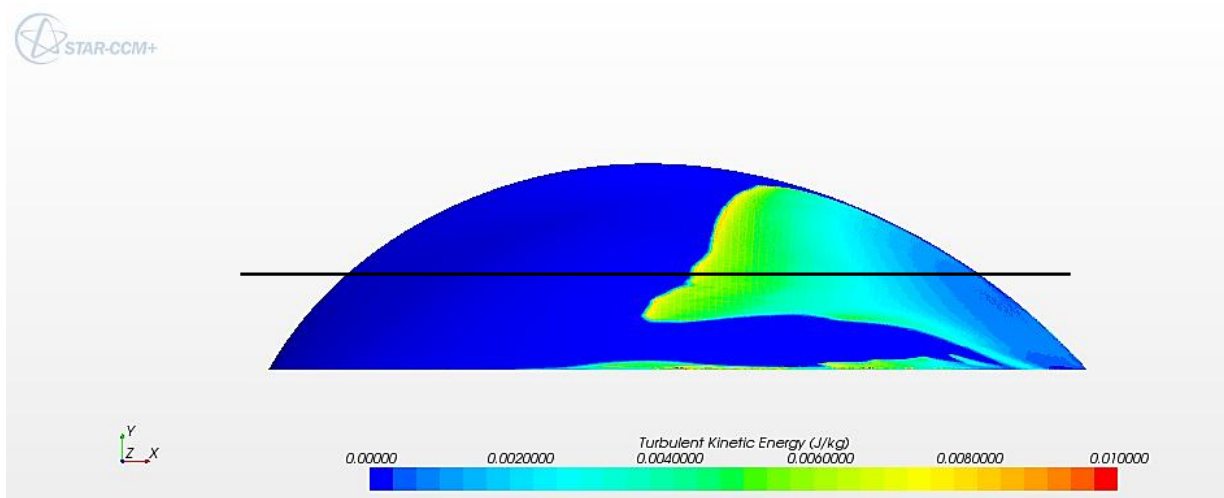


Figure 74 - Canopy 1 turbulent kinetic energy plot

The pressure coefficient plots were analysed at about 50% of the final height of the canopy. This section plane is indicated by the solid line in Figure 74. The same is done in the other canopy images. Analysing the pressure coefficient plot of canopy 1 in Figure 75, the transition location is visible at the 2.5 m mark. Note this is not the length of the canopy, but the position relative to the length of the entire vehicle, which

is 4.5 m. The canopy only has a favourable pressure gradient for about 40%. This is because the widest section of the canopy is to the front, and the trailing section becomes narrower.

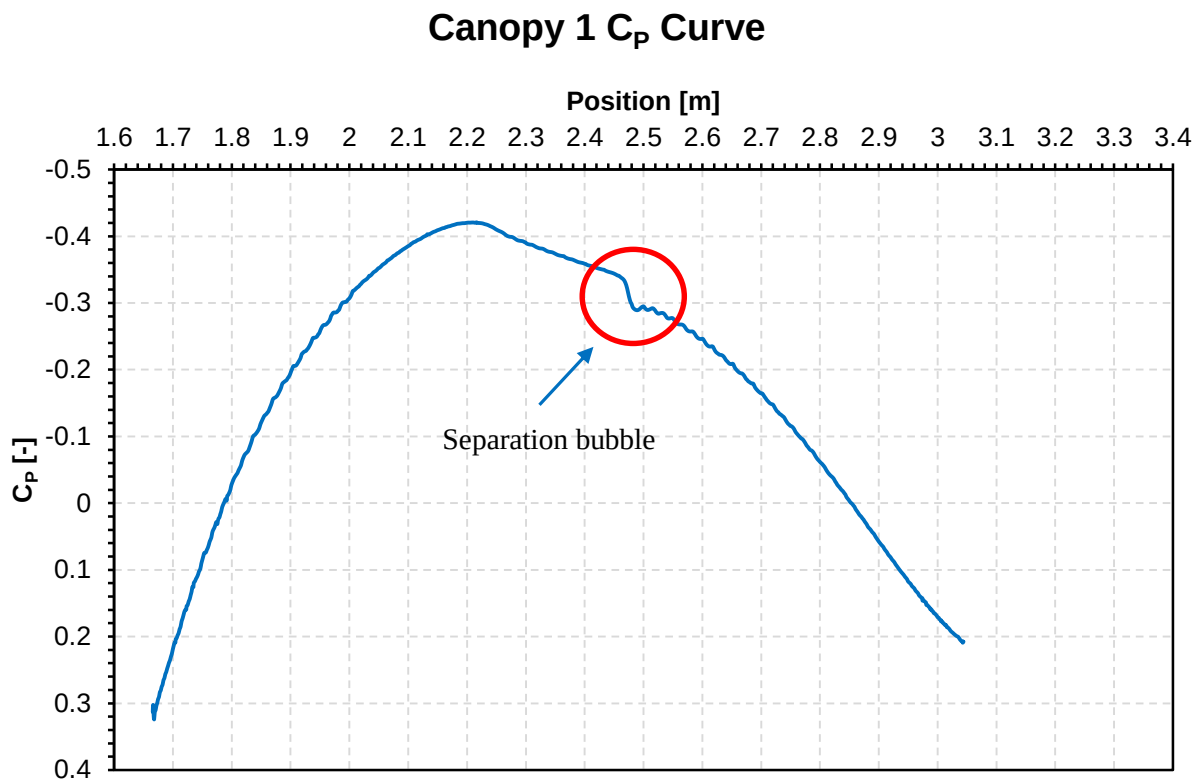


Figure 75 - Pressure coefficient plot of canopy 1

Regarding the space requirements for the driver, this canopy does meet the requirements. However, the trailing section of the canopy recedes quite rapidly. This does not allow ample space for a roll bar that will provide sufficient protection to the driver in the case of vehicle roll-over. The roll bar is an essential requirement for the safety of the driver. For the reasons mentioned, this canopy will be modified to allow more head room for the driver as well as more space for the roll bar integration in the trailing section.

Canopy 2

The second canopy design has a higher profile in the trailing section to allow for easier integration of the roll bar. The canopy still allows the driver the required head space in the x-direction (forward and backward) but the sideward movement of the driver is somewhat limited. This means a smaller frontal area and a lower drag, but will limit the comfort of the driver. When the driver is fully kitted with a helmet, the side to side movement will be limited which might cause driver fatigue over long distances.

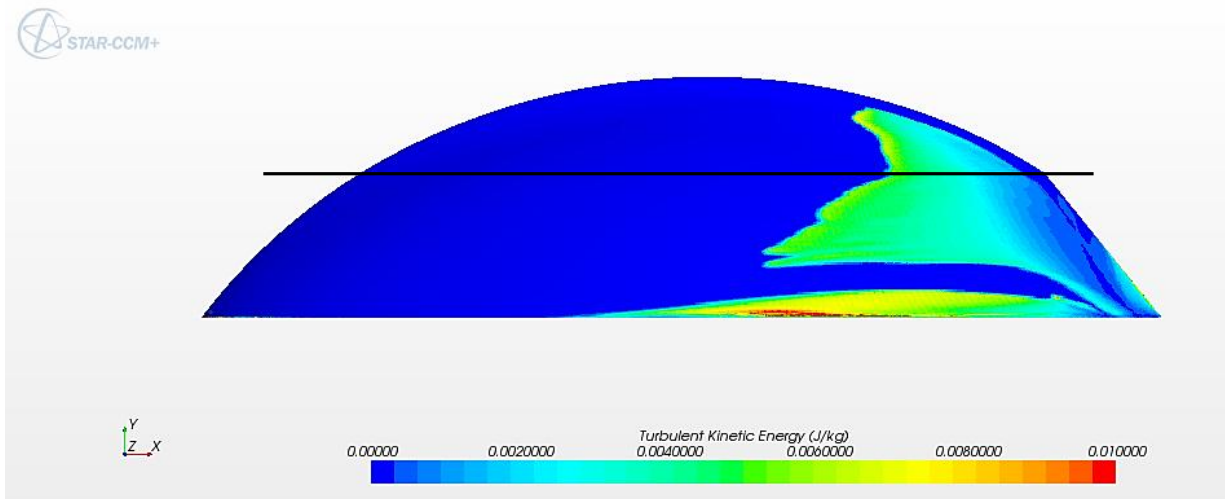


Figure 76 - Canopy 2 turbulent kinetic energy plot

This design has better laminar flow than Canopy 1, extending beyond the halfway mark of the canopy to between 60% and 70% of the chord length (Figure 76). This design also has a larger wetted surface so one would assume it should have a higher drag force, but due to the laminar properties this design has a drag force value of 9.15 N. This is a 15.51% improvement on the previous design.

The pressure coefficient plots of the canopy (Figure 77) were once again analysed at about 50% of the height of the canopy. Canopy 2 has an improved pressure gradient over Canopy 1 and experiences this positive gradient to about 60% before a negative gradient is experienced.

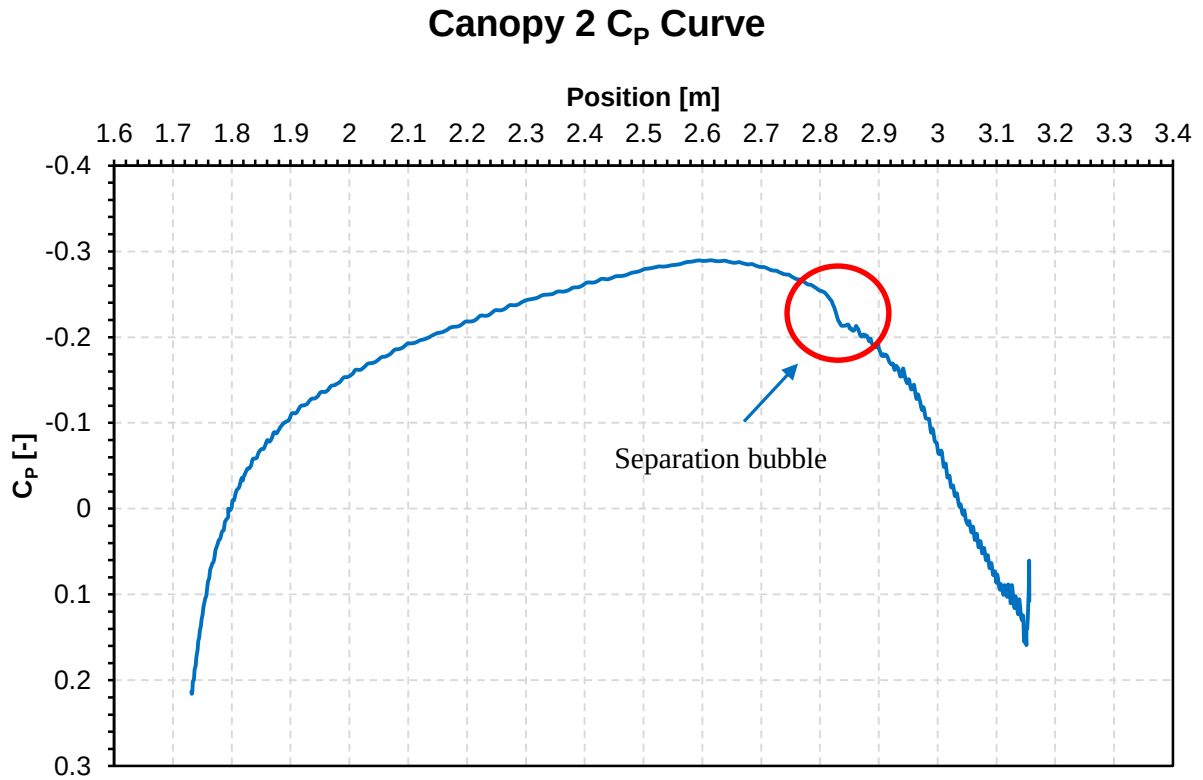


Figure 77 - Pressure coefficient plot of canopy 2

The above plot shows that the flow transitions at about 75% chord length. This is a great improvement over the previous canopy design. The canopy has a sharper pressure recovery region than Canopy 1 but total pressure recovery is not achieved. To allow for more head room for the driver and a narrower trailing section, this canopy design is modified further.

Canopy 3

Modifications to Canopy 2 resulted in a canopy with a narrower trailing section but still wide enough to allow the driver ample head room as well as the integration of a roll bar. This also meant that Canopy 3 would have a very sharp pressure recovery region and an adverse pressure gradient. The second canopy had sharp edges on the rear section but these have been replaced with thinner, rounded trailing edges.

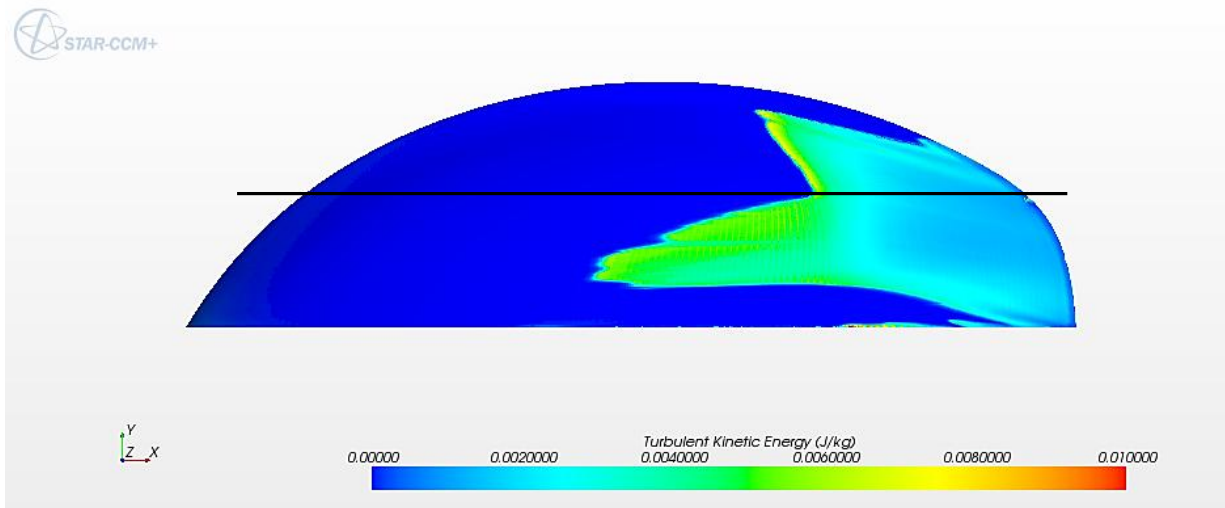


Figure 78 - Canopy 3 turbulent kinetic energy plot

Canopy 3 does not have laminar flow that extends as far as the second design. This can be seen when comparing the turbulent kinetic energy plots with one another (Figure 76 and Figure 78). This is partly due to the larger frontal area, which is necessary for the comfort of the driver. The canopy is wider towards the top of the canopy. This design has a drag force value of 9.44 N which is only a 3.07% increase over Canopy 2, but still 12.83% less than the first design.

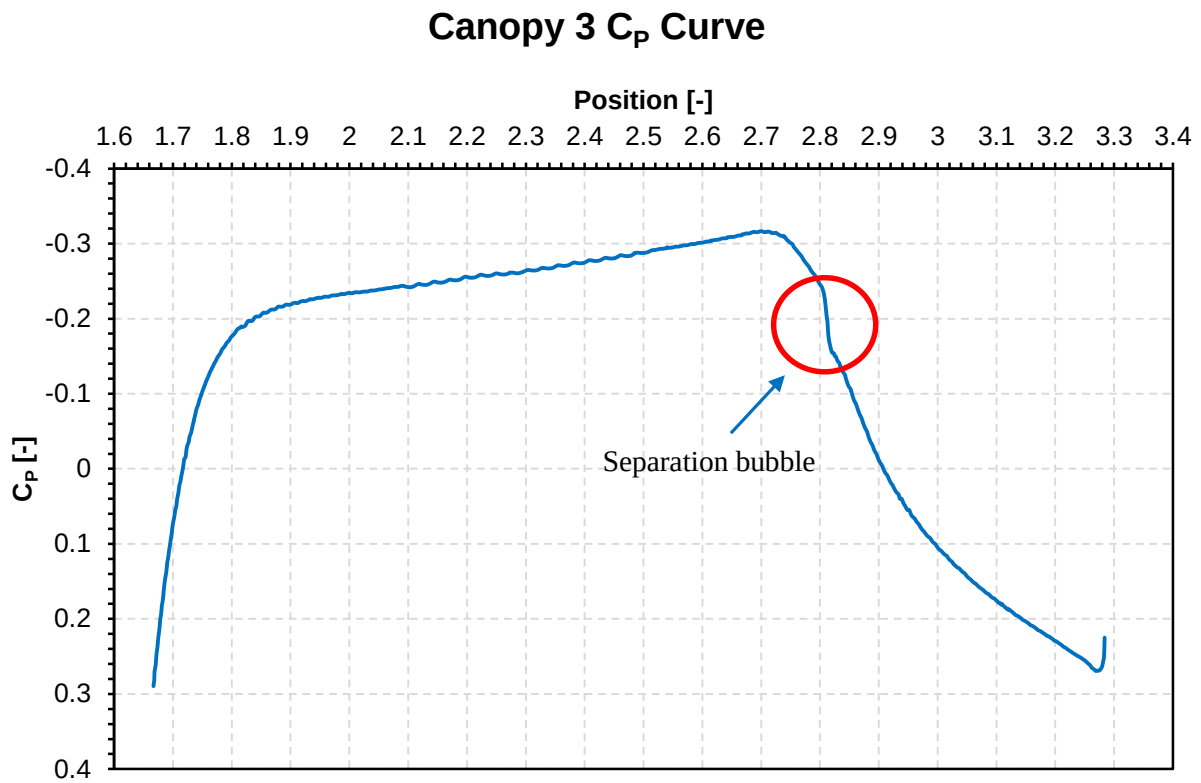


Figure 79 - Pressure coefficient plot of Canopy 3

Referring to Figure 79, a very sharp positive pressure gradient is experienced, transitioning to a gentle gradient. The gradient continues to improve to about 65% chord length. The laminar flow transitions at about 70% chord length after which a steep pressure recovery is experienced, as expected. Almost full pressure recovery is experienced, which is an improvement over the previous canopy. This means that Canopy 3 will have a smaller wake.

The larger frontal area has the effect of increasing the drag force, but as a better pressure recovery is experienced, the drag increase from the previous design is not as great. The design has to be a trade-off between aerodynamics and ergonomics, as driver comfort also plays a large role. Ergonomically, Canopy 3 is the better, design although it has a slightly higher drag force. Canopy 3 is therefore selected to be integrated into the preliminary detail design of the solar-powered vehicle.

7.2 CFD Analysis of Detail Design

Effort was made to minimise the design parameters, especially the frontal area of the vehicle. The CAD part was made using SolidWorks and then evaluated using STAR-CCM+, to determine the design with the most favourable aerodynamic properties in terms of drag coefficient. Adjustments could be made to each iteration, based on the drag coefficients and the transition locations determined when studying the turbulent kinetic energy values on the surface of the model. These adjustments included variations in the fillet size of the different junctions, the trailing edge thickness of the body and the appendages, as well as reduced interference.

The detail design was completed by running simulations using STAR-CCM+ with the $k-\omega$ SST turbulence model and the $\gamma-Re_\theta$ transition model. In order to determine the number of cells where the accuracy converged, namely the mesh independence, the simulation was run at different base size settings. The base cell setting was started at 2500 mm and then reduced with each revision to determine the mesh independence (Figure 80). It was assumed that the simulation reaches mesh independence at roughly 14 million trimmed cells. It was clear that there was no significant reduction in the drag force between 14 million and 17 million cells. The simulations were conducted at a base cells size of 250 mm which related to about 14 million cells. The reason was that the computational power required was still manageable and more accurate than with 8 million cells. Reducing the base size further would not result in a significant increase in the accuracy of the simulation, but the increased computational power would have prolonged the simulation significantly.

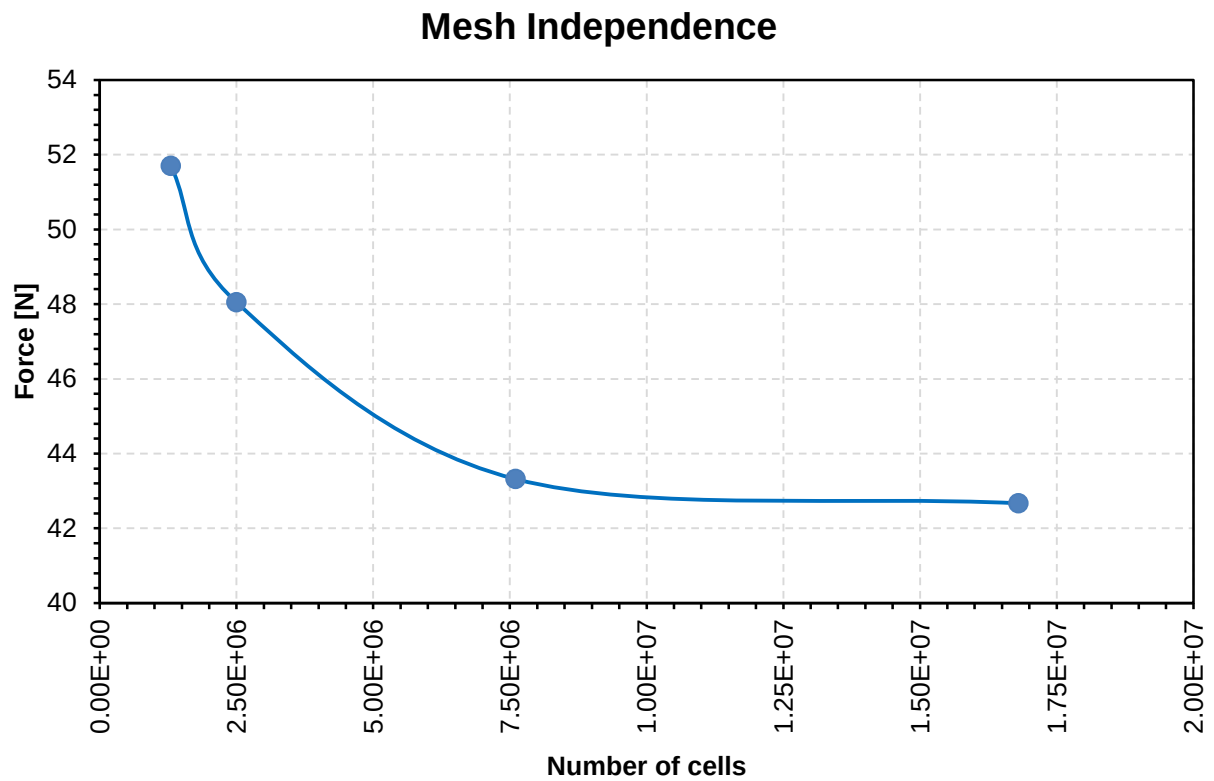


Figure 80 – Mesh independence

The vehicle was simulated in a space of 32 m long, 15 m wide and 7 m high. This relates to 2 vehicle lengths ahead of the model, 5 vehicle lengths behind the model and a tunnel which was more than 6 times as wide and 5 times as high. This relates to the studies mentioned in the literature and ensures that the flow ahead of the model is laminar. The trailing edge flow can also be properly captured as there is enough cells behind the vehicle.

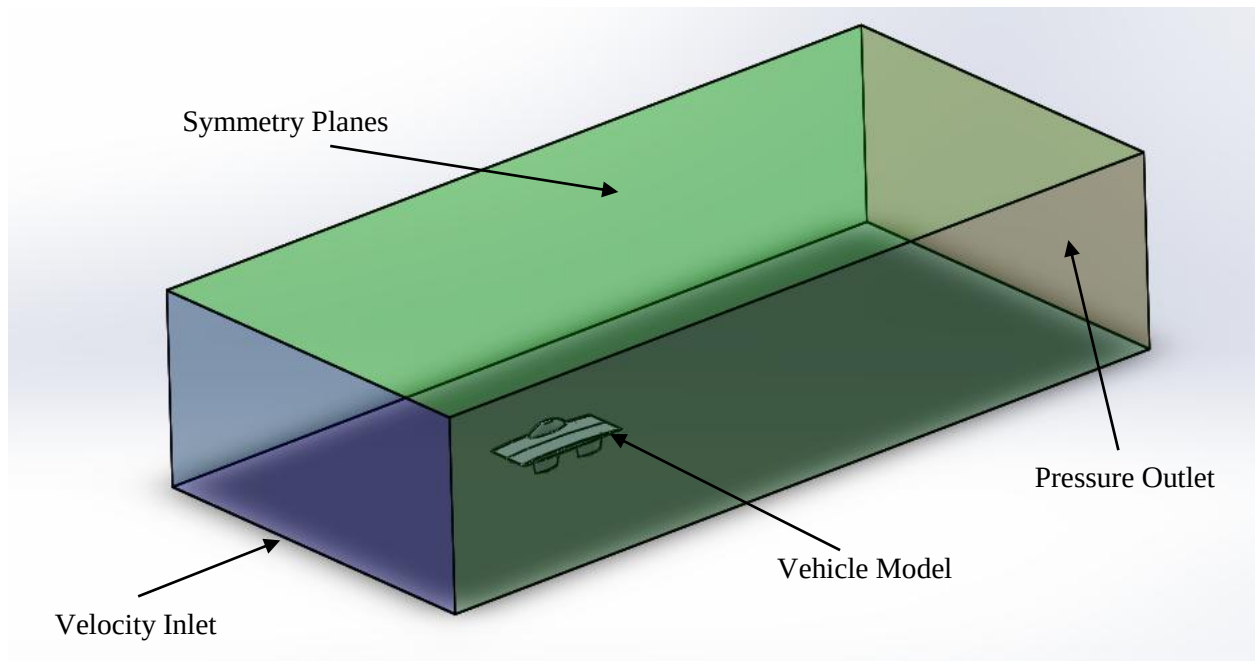


Figure 81 - Detail design simulation boundary definitions

As discussed in the validation and verification chapters earlier, the inlet boundary of the simulated space was treated as a velocity inlet set to a specific velocity. The simulations were set up as in the 2-D validation case, where the model is kept stationary relative to the ground. This means that the air is simply blown past the model. These results, discussed later, did not have the expected result. The simulations were therefore re-done, to simulate the vehicle moving relative to the ground. This velocity was set to 28 m/s as in the 3-D validation case. This relates to a Reynolds number of about 8.3×10^6 for a vehicle model with a length of 4.5 m. The outlet boundary was treated as a pressure outlet. The walls of the block were treated as symmetry planes, therefore it was as though the vehicle was travelling in an open space on a level road. The floor of the simulation domain was treated as a vector at the same tangential velocity as the inlet boundary, in other words 28 m/s. This is to simulate the vehicle moving at a speed of about 100 km/h along the tunnel and not simply standing still with the air blowing past it, at that same speed.

For a smooth transition from the vehicle body to the open space, the simulations were run using the “Template Growth Rate” and the “Boundary Growth Rate” settings, set to “slow”. This ensures that the cells are created evenly around the vehicle’s surface. As indicated, all simulations were conducted with a base cell size of 250 mm, with refinements of 50 mm on the vehicle surface. Feature lines on the vehicle

are important for quality cells. The boundary layer was captured in 15 layers with a stretching factor of 1.5. The thickness of the boundary layer was set to 10 mm.

When analysing the results, the value for the drag force had the tendency of oscillating. These oscillations are as a result of the flow of fluid over the body of the vehicle detaching from the surface of the vehicle, thus becoming turbulent. These oscillations in the final drag force calculation can be very small, but it is necessary to average the values of the last 100 iterations. It can be further reduced by using the polyhedral meshing method, regarded to be more accurate, instead of the trimming method. A typical polyhedral cell has 14 faces as opposed to the 6 faces of a hexahedral (trimmed) cell. This allows for a more accurate solution. However, the number of cells created by the polyhedral mesher was too large when compared with that of the trimmer and would increase the computing time substantially. The following sub-sections describe the constraints that were adjusted with each simulation in order to reduce the drag coefficient of the vehicle.

7.2.1. Model Based on Near Optimum Values

The first model which was based on near optimum values, according to the theory discussed, was based on the same design as the baseline chosen during the conceptual design phase. The features of the model was based on the theoretical optimum values as discussed in the literature, in order to reduce the design time of the vehicle. The junction fillets in this case were 45 mm between the body and the fairings and 20 mm between the body and the canopy. Fairing trailing edges had 7 mm fillets and the body trailing edge had a thickness of 5 mm. The canopy had an L/h ratio of about 4. The trailing edge thickness of the parts would not be varied as it complicated the simulation significantly in terms of obtaining a high quality mesh and achieving convergence.

Applying the trimming method in STAR-CCM+ to the CAD model, the result was a simulation model with a very fine mesh made up of about 13.5M cells. The boundary layer of the vehicle was captured by 15 prism layers, with a thickness of 10 mm. The drag force calculated was 40.7 N. With A in Eq. 2.2 equal to 0.789 m^2 , the C_D value is calculated as 0.107.

The turbulent kinetic energy plots in Figure 82 shows where the laminar flow transitions to turbulent flow on the body of the vehicle. This is where the colour scheme suddenly changes, almost representing a solid line.

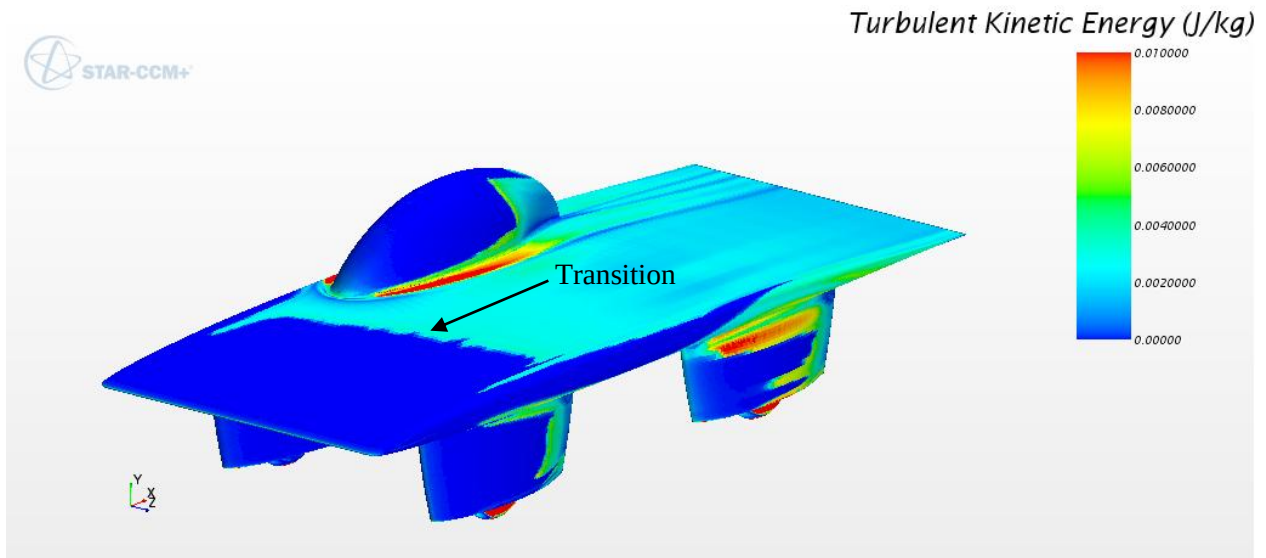


Figure 82 - Laminar flow on upper part of the baseline model

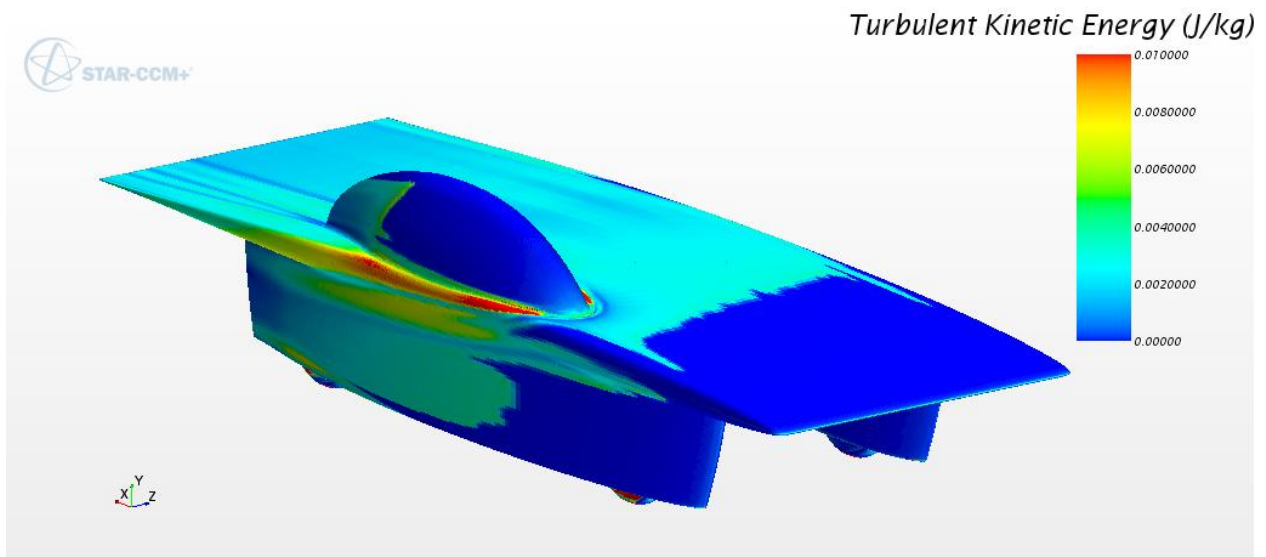


Figure 83 - Laminar flow on right side of the baseline model

Reviewing the turbulent kinetic energy plots of this baseline model (Figure 82 and Figure 83), it can be estimated that the laminar flow on the upper part of the body extends to about 35% chord length. This correlates with the individual analysis of the main body of the vehicle. This is not ideal, but the transition position can be attributed to the thickness of the profile. A profile with a smaller frontal area might extend

the laminar flow, but this reduces space for internal components. The body profile is already at a minimum thickness. The rounded edges of the leading-edge of the body do not cause transition.

The canopy shows favourable laminar flow, as already discussed in a previous section. The wake behind the canopy, however, is quite large, but not much wider than the canopy thickness. This is expected. The canopy can, however, be improved to reduce this wake and improve laminar flow. This might also aid in the reduction of the high levels of turbulent flow on the body adjacent to the canopy, caused by the vortex that is formed at the head of the canopy.

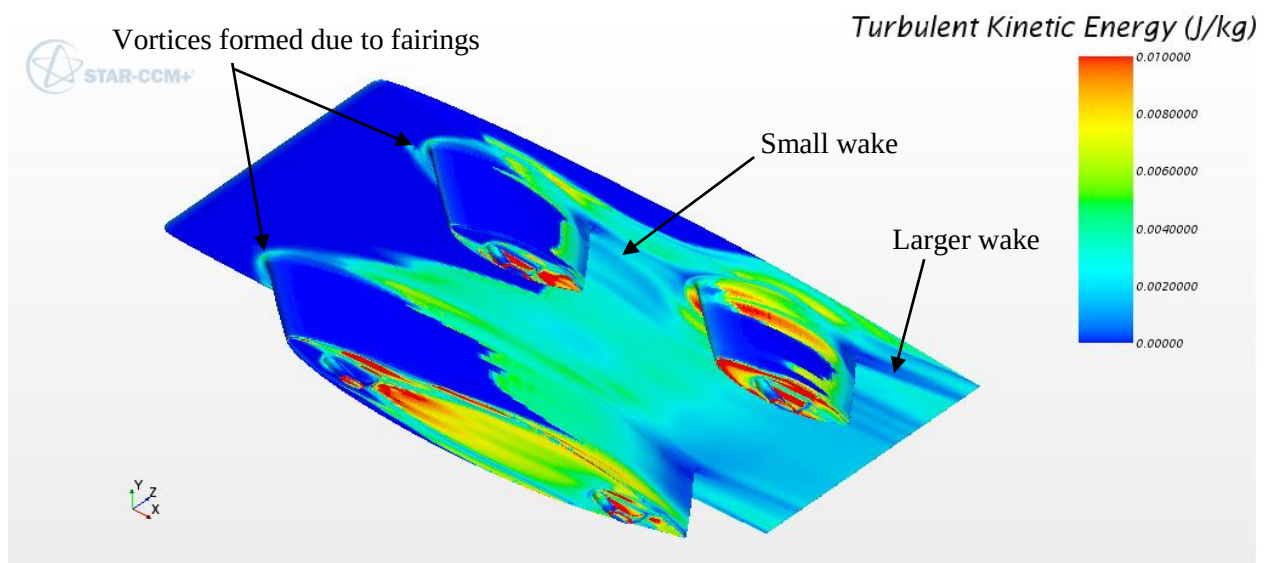


Figure 84 - Laminar flow on lower part of the first vehicle model

Referring to the literature, it is stated that the optimum fairing junction fillet radius should be between 4-6% of the chord length of said appendage. 45 mm is a good initiation point for the first revision. Referring to Figure 84, it can be observed that the fairing junction fillets cause the flow to transition due to adverse pressure gradient caused by the junction. This causes the laminar flow ahead of the fairing to transition. The turbulent flow is then forced into a vortex along the side of the fairing. The transition of laminar flow causes turbulent flow on the rest of the lower body section, with the farthest laminar flow experienced to about 40% of the body.

Laminar flow on the front fairing is favourable, extending to about 80% of the chord length. This was expected as the simulations of the fairing configurations showed this. The flow separates and then partially re-attaches on the trailing edge of the fairings. Due to the re-attachment of flow and the strong pressure

recovery present during the individual simulation of the fairing configuration, the wake behind this fairing is small. This still means that the rear fairing is caught in the turbulent wake. This turbulent wake causes a larger vortex to be formed along the junction of the fairing and the body. Laminar flow, however, is still experienced across about 50% of the surface of the rear fairing, but not as much as on the first fairing. This is attributed to the vehicle moving at speed, ensuring that the flow can re-laminarise by the time it reaches the rear fairing. Due to the larger vortex along the junction, a larger wake is experienced behind the rear fairing. Laminar flow on the right hand fairing extends to about 40% chord length and partial reattachment is also observed. The wake behind this fairly thick fairing is satisfactory. The sharper edges at the bottom of the fairings also cause transition on the floor section of the fairings, on the inner side.

Systematic adjustments to be made to the current model are as follows:

- Adjust fairing junction fillets to see effect on transition of laminar flow and drag coefficient of the vehicle. Fillet radii should theoretically be 4-6% of the chord length of the fairing.
- Adjust the canopy junction fillet radius.
- Left hand front fairing should be adjusted to allow rear fairing to experience more laminar flow.
- Increase fillet radius on the bottom edges of the fairings.

7.2.2. Effect of Increasing Fillet Radii on Fairings

The model that is described next remains the same with only the fairing junction fillets changing from 45 mm to 60 mm. This is to test the first point mentioned in the previous sub-section. The theoretical optimum for the junction fillet radius is described as being 4-6% of the fairing length. The fairings being 1000 mm long meant that the junction fillet would be between 40 mm and 60 mm. This revision will thus test if enlarging the radius will allow the flow to progress smoother around the fairings. At this moment all fairing radii are kept constant at 60 mm, regardless that the length of the right hand fairing is 3000 mm.

This resulted in the simulation model comprising 13.4M trimmed cells. The vehicle has A as 0.79 m^2 , slightly larger than the previous model, due to the larger junction fillets. D is equal to 41.1 N , relating to a C_D value of 0.108 . Comparing these results with that of the baseline model, there is a 1% and 0.1% increase in the drag force and frontal area values, respectively. The value for C_D slightly increases by 0.9%.

In the figure below (Figure 85) it is clear that the laminar flow does not change significantly by changing the fillet radius to 60 mm.

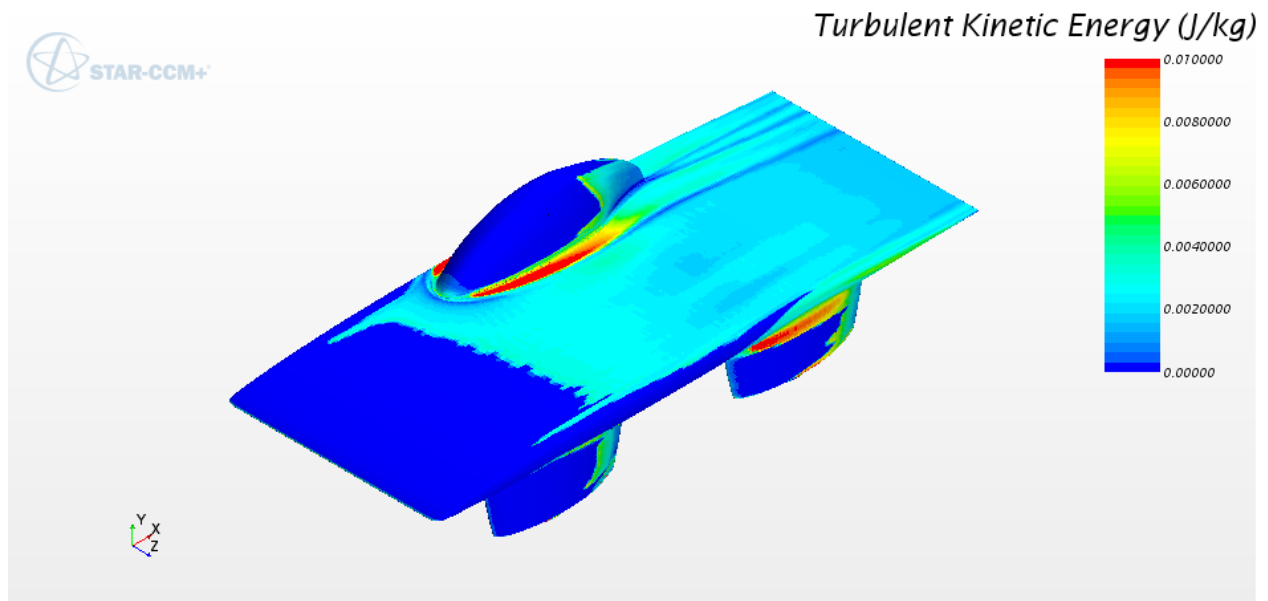


Figure 85 - Laminar flow on upper surface of the model after changing the fillet radii to 60 mm

In Figure 86 the reason for the increase in the drag force is observed. The larger junction radius causes the flow to transition earlier than in the previous section and therefore the horseshoe vortex is formed quicker. This means that the laminar flow on the belly section of the vehicle does not extend quite as far. These occurrences has the effect of increasing the drag force. The fillet radius is still within the optimum range, so even though the fillet causes a larger wetted surface, the vortices created do not increase the drag significantly as the increase is only about 1%.

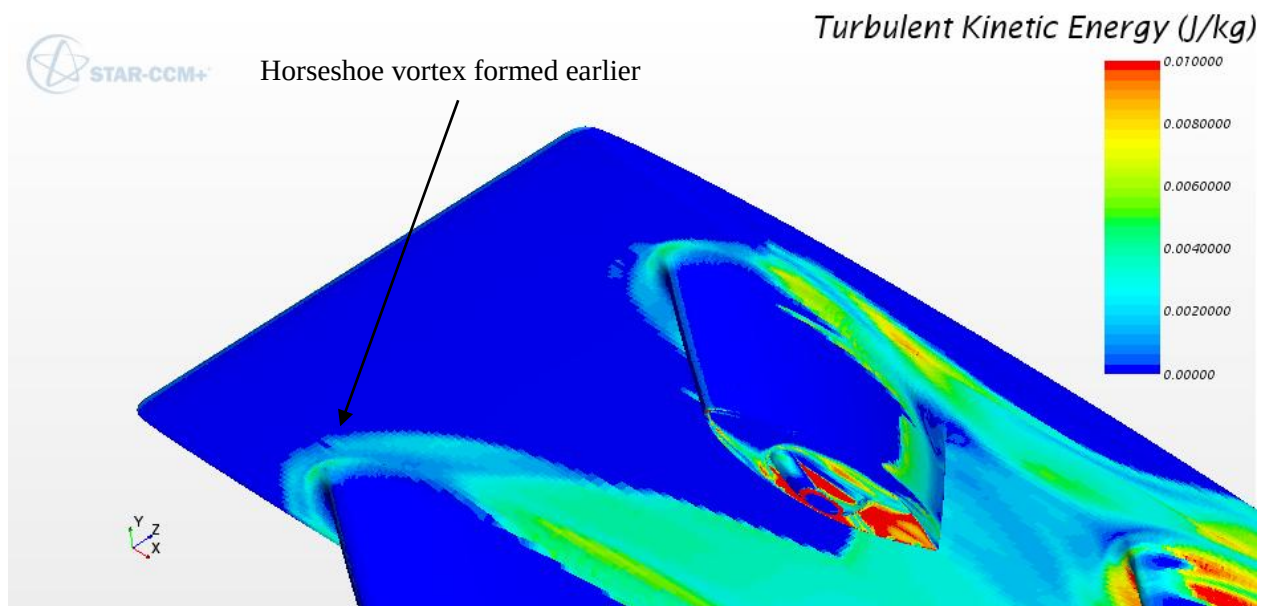


Figure 86 - Transition at fairing leading-edges of the revised model

The increase in the drag force and the drag coefficient is small and it can therefore be assumed that the 4-6% fillet size theory still holds true. The fact remains that the increase in the radius increased the drag force. It can be assumed that if this radius is increased further, especially on the shorter fairings, the increase in wetted surface and frontal area will cause a much larger increase in the drag force.

The differences between the first two models of the vehicle are so little that either model can be used for further optimisation. The following model will therefore serve as a test to see the effect on the drag coefficient by reducing the radius (outside the theoretical optimum).

7.2.3. Effect of Reducing Fillet Radii to Below Theoretical Optimum

The third model revision of the vehicle, tests the fairing junction fillet radius outside the theoretical optimum at 30 mm. This is 3% of the chord length of the fairing. For the theory tested in the previous two revisions to hold true, this model should show an increase in the drag force value, even though there will be a slight reduction in the frontal area of the vehicle.

The trimmed simulation model results in 13.5M cells, with A equal to 0.789 m². This model had the same drag force value as the first model. With the drag force equal to 40.4N, this is a reduction of 1.7% in the drag force value from the model with the 60 mm fillet radius. The reduction, compared to the first model with a 45 mm fillet, is negligible. The frontal area of this model is 0.2% smaller than that of the 60 mm

fillet radius model. The drag force measured on this model relates to a C_D value of 0.107, which is a reduction of about 1.5%, compared to the second model revision.

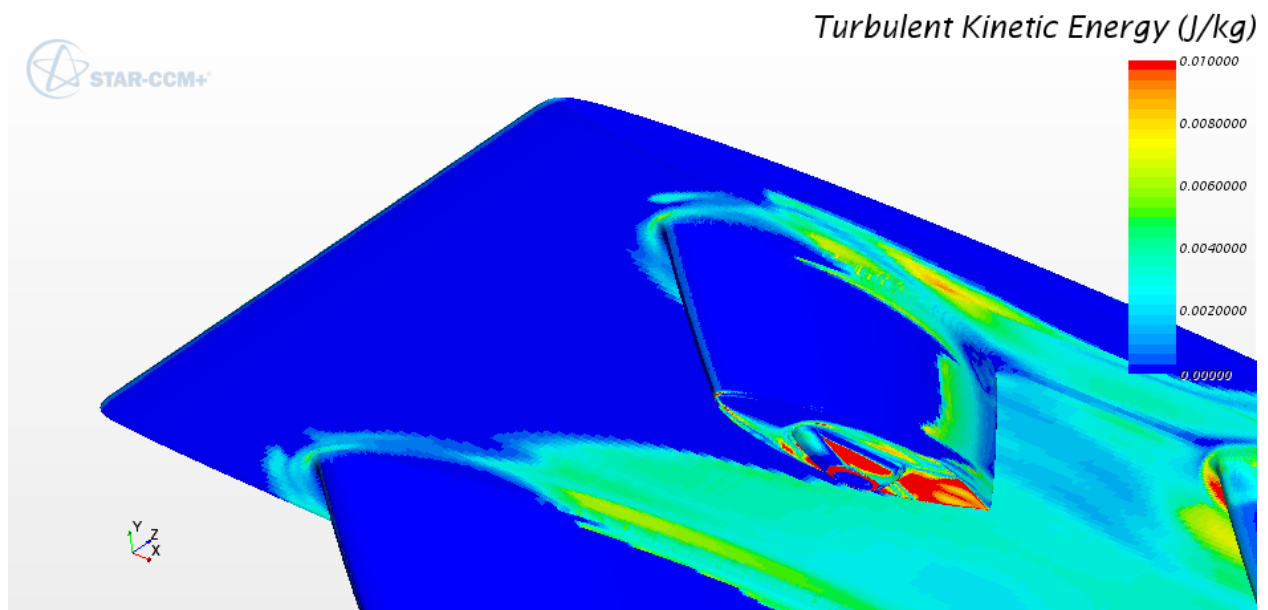


Figure 87 - Transition at fairing leading-edges of the 30 mm fillet radii model

One can see that this model has increased laminar flow at the leading-edge of the fairing junctions with the body, when referring to Figure 87. It can also be observed that the effect of the vortex is not as great as with the previous model, where a 60mm radius was implemented. The smaller radius has the effect of reducing the effect of the vortex and thereby the total drag force of the vehicle. This relates to research done by Maughmer *et al.* [41], where it was found that the smaller the fillet radius, the lower the drag force on the junction. The reason for this is that the smaller fillet radius does not extend as far ahead of the fairing as the larger fillet and the transition of flow is slightly delayed. The drag force is not increased greatly, rather remaining the same as the 45mm fillet radius. It can therefore be assumed that the theoretical 4-6% ratio for the fillet size is not a rule of thumb and a smaller fillet radius might be advantageous.

This same theory can be applied to the junction fillets of the canopy. The same procedure will be followed as with the fairings. The fairing fillets will be reverted to the 45 mm size as it has a slightly larger frontal area, but the same drag coefficient than the smaller 30 mm fillets. The canopy fillets will be changed from the current 20 mm size to 45 mm fillets, to further test the theory.

7.2.4. Applying the Fillet Radius Theory to Canopy Junctions

Having tested the fairing junctions, the same method can be applied to the canopy junction with the body. This is a more sensitive case as increasing the canopy junction radius, decreases space available for the solar array. It is for this reason that the initial junction radius is chosen at 20 mm. The canopy junction radius is set to 45 mm.

Adjusting the fillet radius from 20 mm to 45 mm resulted in a trimmed simulation model with 13.4M cells. A is equal to 0.789 m^2 , showing that the larger fillet radius of the canopy junction had no significant effect and the change is negligible (about 0.1%). The model was, however, still simulated in order to see the effect that the larger radius will have on the overall drag force, as it is usually accepted that a smaller junction will cause a horseshoe vortex, with a larger effect on the total drag force of the vehicle. The drag force simulated was 40.5 N, about the same as the previous model, as well as the baseline model. At this stage it can be assumed that the fillet radius will apply to the canopy junction fillet size as well. The drag force and the frontal area results in a C_D value of 0.107, as in the other similar models.

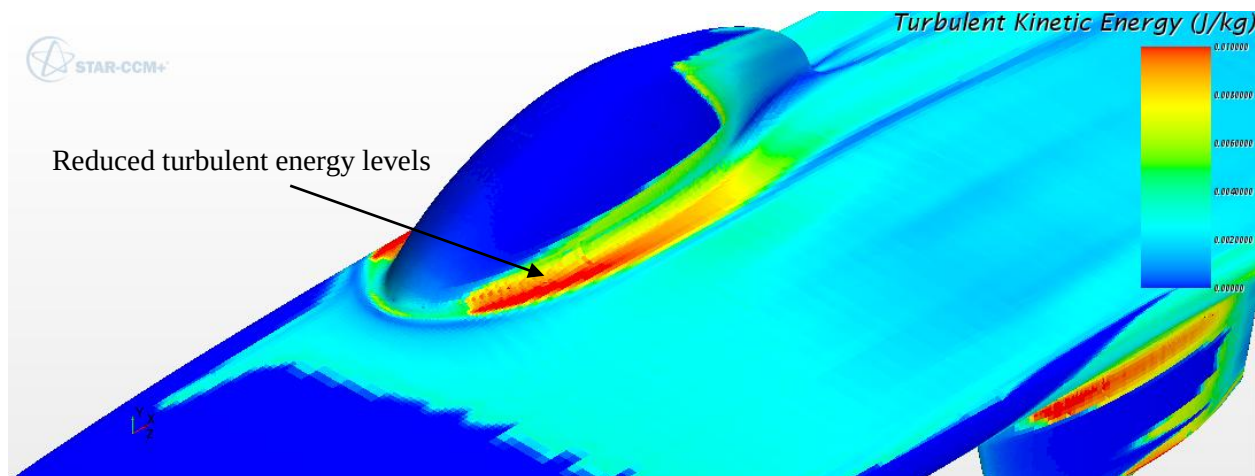


Figure 88 - Canopy wake of the present model

When comparing Figure 88 with Figure 85, it shows that the increased canopy junction fillet radius has the effect of slightly reducing the turbulent kinetic energy levels on the body adjacent to the canopy. This might be attributed to the large radius, thus the effect of the vortex on the body is reduced, albeit only slightly, with no significant change on the drag force.

7.2.5. Effect of a Large Canopy Junction Fillet

As described above, the canopy fillet radius is varied. The fifth model has a canopy fillet radius of 60 mm and keeps the fairing fillets the same as in the previous revision. When the 60 mm fillet radius of the fairing junction was tested in the second model, an increase of about 1% was experienced compared to the baseline model with a 45 mm fillet. It can now be expected that the same trend will hold if the canopy junction fillet is increased from 45 mm to 60 mm.

The simulation results in a vehicle with 12.6M trimmed cells. The value for A is 0.79 m^2 , which is 0.1% greater than the previous model and about 0.15% greater than the first or baseline model. D is 41.1 N, 1.5% greater than the model with the 45 mm canopy fillet and 1% in excess of the baseline model. This translates to a C_D value of 0.108, which is about 1.4 % greater than the previous revision and 0.8% greater than the baseline model.

Taking this into consideration, it can be assumed that the same theory applies to the canopy junction fillets as described for the fairing junctions, as the same trend of about 1% drag increase is experienced. It means that when the vehicle is moving at a speed of about 100 km/h, the size of the junction fillets does not have such a large effect on the overall drag of the vehicle, provided that it is within the theoretical optimum tested here. The canopy junctions might also not have such a large effect on the drag force of the vehicle because the canopy is already in a turbulent flow region. If the flow was laminar at the start of the canopy, the vortex might have had a larger effect on the total drag force.

Looking at the revisions, it can be assumed that the best fillet radius size for the fairings, as well as for the canopy is 45 mm and will be used in models to follow. It should be noted that this may not be the optimum, but due to time constraints, more revisions are not conducted for the fillet radii. It just illustrates the method that can be used to achieve a near optimum value. It can, however, be confirmed that the optimum radius size is between 4-6% chord length.

7.2.6. Effect of Diverting Flow around the Rear Fairing

Having established the effects of fillet radii for this vehicle design, one can move to the third point as outlined at the end of the first revision of this vehicle design. This is to adjust the front left hand fairing so

that the trailing flow will be channelled around the rear left hand fairing. This is achieved by using an asymmetrical fairing; in this case, the same airfoil profile used to design the main body section. The 2-D pressure coefficient for this profile was analysed and a favourable pressure gradient was present. The fairing should therefore experience good laminar flow, reducing the overall drag force of the vehicle. The fairing should allow the flow to pass on the inner side of the rear fairing. The fairing fillets are kept at 45 mm.

The simulation results in a vehicle with 13.4M trimmed cells. The value for A is 0.794 m^2 , which is roughly 0.5% greater than the previous model (canopy fillet adjustment) and 0.6% greater than the baseline model. This larger increase is due to the new fairing being wider at the top, where it is integrated with the body. This change in the geometry sees the drag force reducing to 39.5N, 3.8% less than the previous model and 2.9% less than the baseline. This translates to a C_D value of 0.104.

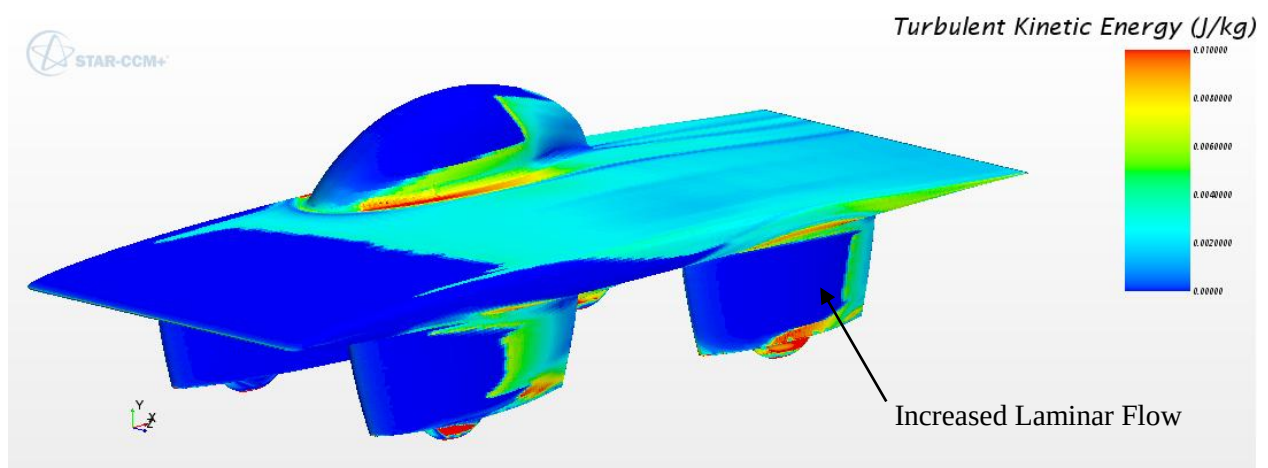


Figure 89 - Laminar flow on the sixth model

In Figure 89 the improved laminar flow on the rear left hand fairing is clear. The turbulent flow on the lower section of the rear fairing near the ground, slightly increases the drag force, but not as much as when the rear fairing is completely in the turbulent wake of the front fairing. This increased laminar flow on the rear fairing, ensures that even with an increase in the frontal area of the vehicle, there is actually a decrease experienced in the drag coefficient of this model. There is more turbulent flow adjacent the fairing on the inner side of the body, but this side of the fairings also sees a significant increase in the laminar flow.

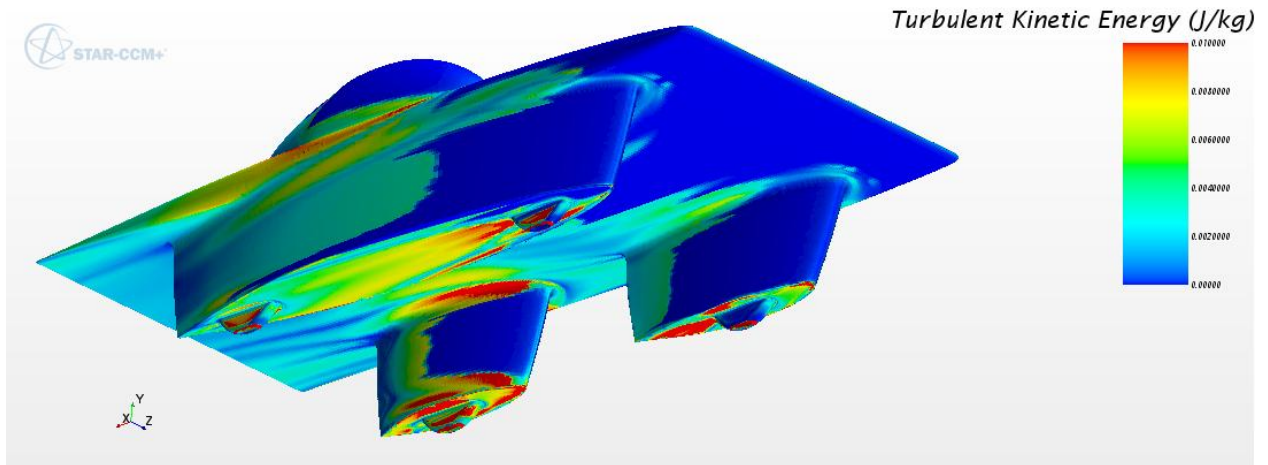


Figure 90 - Laminar flow on inside of fairings

There are still high levels of turbulent flow on the lower part of the rear fairing (Figure 90), which might be improved if the fairing does not have such sharp edges at the bottom. This will be improved on in a later model revision. The wake behind the fairings is also clear in Figure 91.

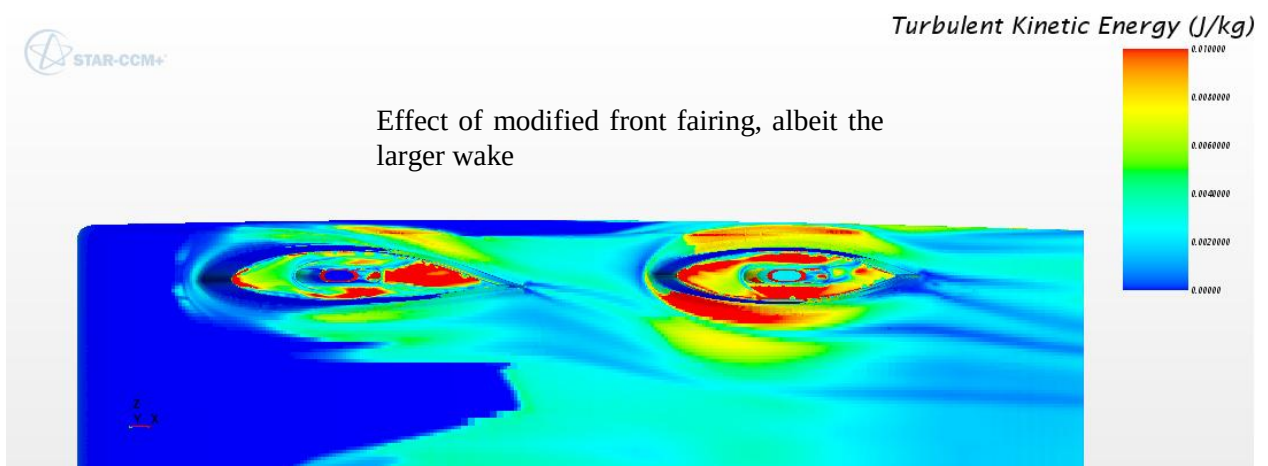


Figure 91 - Fairing wake due to the modified front fairing

This fairing configuration, different from the configurations discussed in the previous section, actually creates a larger wake than two identical fairings, as can be seen when referring to Figure 91. However, the increased laminar flow on the rear fairing ensures that the larger wake does not result in an increase in the drag force. The increased turbulent flow around the rear fairing, as mentioned, is visible in red.

It can be assumed that the advantages of the configuration discussed here, outweighs that of the two identical fairings. Even though it complicates the manufacturing process, the increased laminar flow may increase the efficiency of the vehicle in race circumstances.

7.2.7. Applying Fillet Radius Theory to the Driver Cockpit Fairing

Having established that the two different fairings give a positive change, the following step is to once again change the theoretical optimum for the fairing fillet radius. It was mentioned previously that the length of the long fairing, which includes the driver cockpit, has been ignored until a later revision. This fairing has a chord length of 3000 mm, and if the same theory is to be applied, the fairing fillet radius should be set to between 120 mm (4% of chord) and 180 mm (6%). For the shorter fairings a value of 4.5% was used and therefore the long fairing fillet radius will be set to 135 mm. Once again it is assumed that the theoretical optimum value will hold true and a reduction in the drag coefficient will be experienced.

The simulation results in a vehicle with 12.4M trimmed cells. The value for A is 0.797m^2 , which is roughly 0.4% greater than the previous model without the 135mm fairing fillets and 1% greater than the baseline model. This increase is due to the 135 mm fillet radius used on the long fairing. This change in the geometry sees the drag force increase to 40.0 N, 1.2% greater than the fairing modification, but 1.7% less than the baseline model. The drag force translates to a C_D value of 0.104, which is equivalent to the previous model. The model experienced an increase in the frontal area, and this together with only a slight increase in the drag force (0.5N), compared to the previous model, results in the drag coefficient remaining the same.

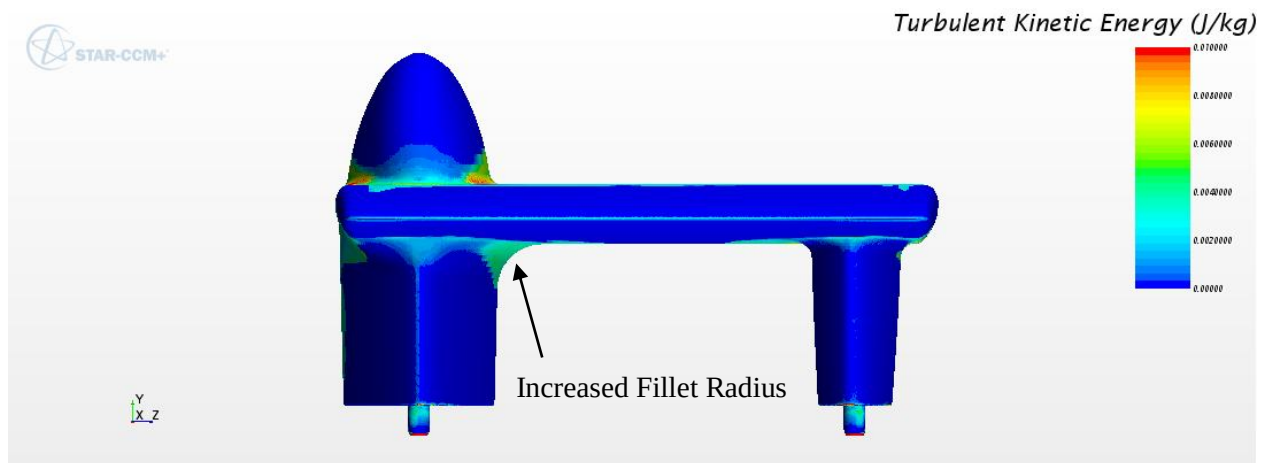


Figure 92 – Increase in frontal area due to larger fillets on the right hand fairing (left side in above image)

Figure 92 shows the larger fillet of the right hand fairing with the associated increase in the frontal area of the vehicle. As mentioned this increase does not result in an increase in the drag coefficient of the vehicle. Once again it can be ascribed to the reduction of the effect of the vortex created at the start of the fairing

and it can be assumed that for this model the theoretical optimum for the fairing junction size once again holds true. This theoretical optimum should be investigated with all future vehicle designs, as it is not a rule that it will reduce the drag force. It is largely dependent on the effects of the vortex created by the junction, which in some cases might be reduced by a larger fairing radius.

7.2.8. Effect of Removing Sharp Floor Edges on Fairings

The next step to improve on the drag coefficient is the final point mentioned at the end of the baseline model section. This is to increase the radius of the bottom edges of the fairings to promote a smooth transition of flow from the leading-edge of the fairing to the bottom floor section.

This results in a simulation with 12.7M trimmed cells. The value for A is 0.801 m², which is roughly 0.5% greater than the previous model and 1.5% greater than baseline model. This larger increase is due to the new fairing being slightly wider due to the changed geometric nature of the fairing design. This change in the geometry sees the drag force reduced to 39.1 N, 2.3% less than the second to last model (effect of modifying the front fairing) and 4.0% less than the baseline model. This translates to a C_D value of 0.102, which is a reduction of 2.8% lower the previous model with the larger fairing. This drag coefficient is also a 5.8% reduction from the baseline model. This satisfies the objective to achieve a reduction of at least 5% in the drag coefficient of the vehicle.

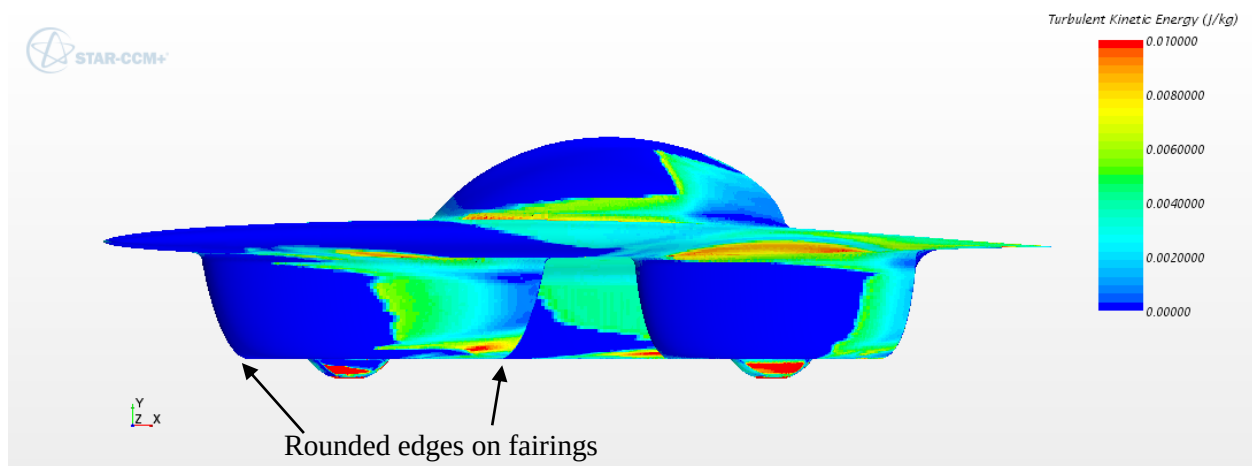


Figure 93 - Rounded lower edges of fairings

Increasing the radius of the lower edge of the fairings can be observed in the above figure, Figure 93. This change also sees an increase in the overall length of the fairings, albeit slightly. The fairing can be scaled

down in the design phase in order to maintain the same chord length as in previous models, but this will also reduce the length of the lower section of the fairing. This, in turn, means that the width of the fairing will either have to remain the same (thus scaling the fairing in the x direction only and thereby elongating it), which will alter the airfoil properties, possibly reducing the laminar flow, or scaling the fairing uniformly meaning that the width of the fairing is reduced and thereby the space available for the wheels and suspension. The fairing is thus elongated.

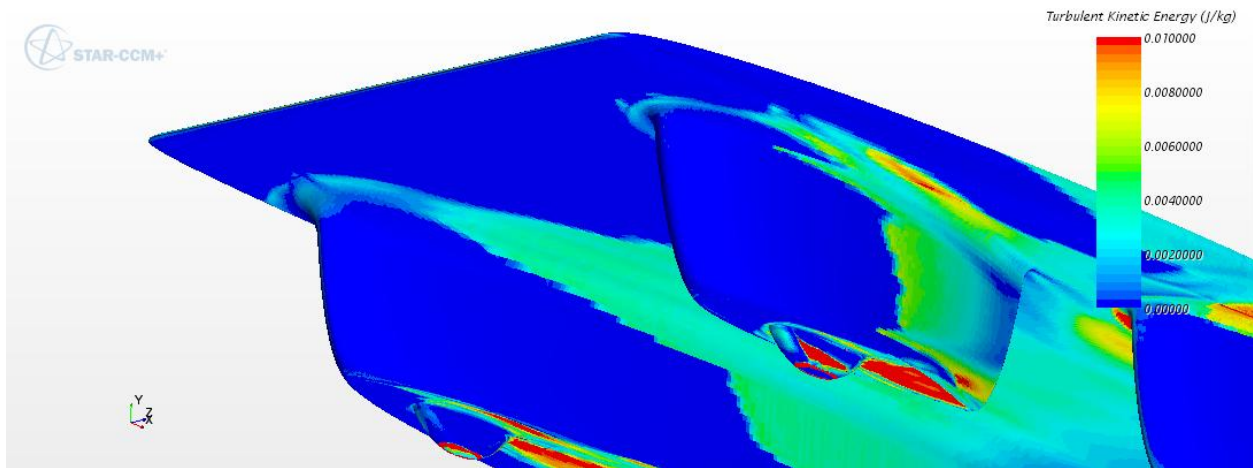


Figure 94 - Improved laminar flow on lower edge of fairings

The increase in the radius of the lower edge does achieve what is expected, namely to increase the smooth transition of laminar flow from the leading-edge of the fairing to the lower section, without causing transition to turbulent flow prematurely (Figure 94).

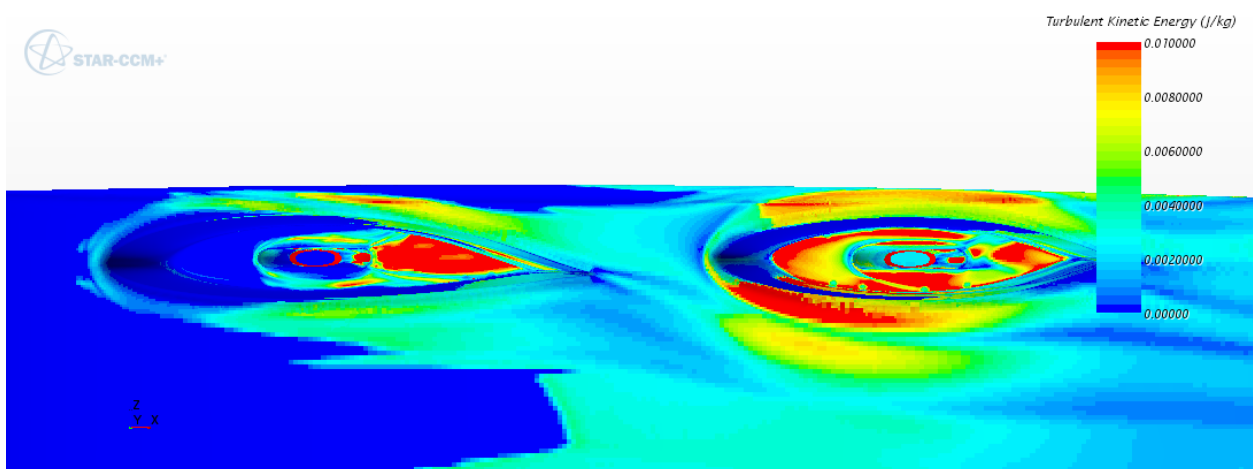


Figure 95 - Improved laminar flow on rear left fairing

The same flow properties is not experienced on the rear fairing, probably because of the turbulent wake caused by the front wheel (Figure 95). Even though this is the case, the drag force of the vehicle is not increased, and the drag coefficient is further improved.

7.3 Detail Design Results

It was mentioned that the simulations were done with the model stationary relative to the ground. These results are summarised in Table 6. The result of the last model increased, and as it was expected to decrease, it was opted to redo the simulations with the vehicle moving relative to the ground.

Table 6 - Drag force results of the stationary vehicle

	Model 1	Model 2	Model 3	Model 4	Model 5	Model 6	Model 7	Model 8
F	43.0	43.0	40.4	42.1	43.8	42.3	41.4	41.9
CdA	0.089	0.090	0.091	0.088	0.091	0.088	0.086	0.087
A	0.789	0.790	0.789	0.789	0.790	0.794	0.797	0.801
Cd	0.113	0.113	0.115	0.111	0.116	0.111	0.108	0.109

The step where the fillets were optimised for both the long and short fairings (Model 7, Table 6) had the lowest drag coefficient value of 0.108, which is 4.8% lower than the very first model simulated and before any changes to the features of the vehicle were made. The last model of the vehicle saw an increase of 0.5% compared to this model, which was unexpected.

The results of the variation in the design of the solar-powered vehicle, when moving relative to the ground, is summarised in Table 7. It can clearly be seen in Figure 96 how the drag coefficient varies with the changes to the design, as described in the preceding section of the dissertation.

Table 7 - Drag force results of the moving vehicle

	Model 1	Model 2	Model 3	Model 4	Model 5	Model 6	Model 7	Model 8
F	40.7	41.1	40.4	40.5	41.1	39.5	40.0	39.1
CdA	0.085	0.086	0.084	0.084	0.086	0.082	0.083	0.081
A	0.789	0.790	0.789	0.789	0.790	0.794	0.797	0.801
Cd	0.107	0.108	0.107	0.107	0.108	0.104	0.104	0.102

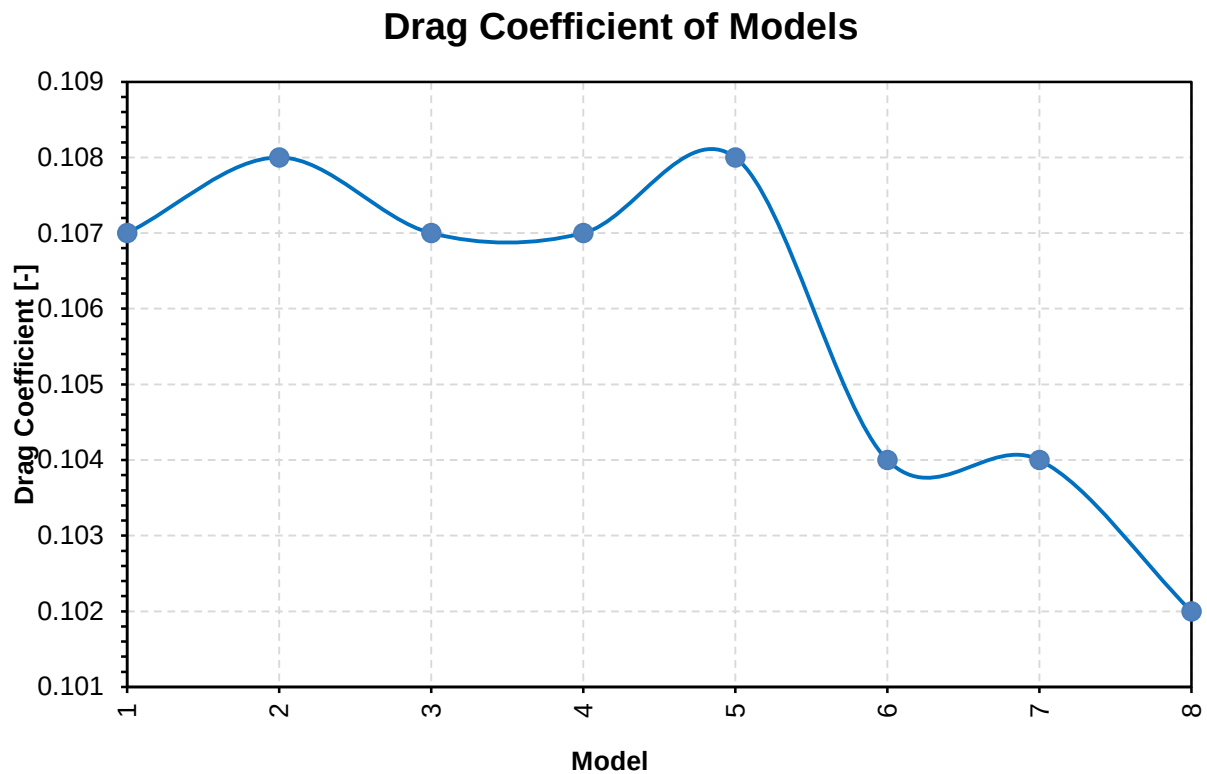


Figure 96 - Drag coefficients of the vehicle model when moving relative to the ground

The last model, with the rounded fairing edges and the optimised fillets for both the long and short fairings (Section 7.2.8), had the lowest drag coefficient value of 0.102, which is 5.8% lower than the very first model simulated and before any changes to the features of the vehicle was made. Reviewing the above results, it should be noted that the vehicle should always be simulated when moving relative to the ground, instead of being stationary.

7.4 Conclusion

These iterations and modifications to the vehicle show that it is possible to improve the vehicle design by following the methodology described. It was stated in the objectives that a reduction of 5% in the drag coefficient is deemed significant, provided that a near optimum base model is used. The results showed that the base model was improved by almost 6%. It should be noted that this was achieved when the vehicle was simulated to be moving relative to the ground rather than remaining stationary. This is a more realistic representation of reality and causes a significant reduction in the drag force calculated. It can therefore be assumed that this is a suitable methodology, although many more iterations and modifications may be

required to significantly reduce the drag of the vehicle. By applying this methodology to future designs, this is possible. Reducing the effect of vortices created by the appendage fillets, by keeping the fillets within a theoretical optimum and in some cases reducing the fillet size below this theoretical optimum value, is the most significant contribution in the reduction of the drag coefficient of the vehicle. Adding to this, it should be ensured that all the appendages experience laminar flow, meaning that none of the appendages should be in the turbulent slip-stream of another appendage. The methodology also proves that CFD is indeed a valuable tool in the design process and should be continually implemented and improved upon. It significantly reduces the design time required and the need for the use of wind tunnels. These results can be further validated by the use of a wind tunnel to test the as-built design of the vehicle and comparing this with the CFD results.

Chapter 8: Conclusion and Recommendations

The main objective of this study is to optimise the aerodynamics of a solar-powered vehicle using CFD. This had to be conducted within the Sasol Solar Challenge regulations. In doing this, the method is described that can be used to analyse and design future vehicles and it is proven that CFD is a viable tool with which to conduct the work. Lastly, the transition models used in the CFD code had to be validated. The conclusions that are made from the study are summarised under the headings below.

8.1. 2-D Validation

The NACA 0018 airfoil was used for the 2-D validation, as well as a trimmed mesh setting. The airfoil was tested at different angles of attack, which were achieved by using an overset mesh function within STAR-CCM+. This correlates with the experimental tests conducted by Timmer which was also conducted at a range of angles of attack. At lower angles of attack, the CFD results compared well (within 5%) with these experimental results, as well as with the results obtained from the XFOIL-based XFLR5. At higher angles of attack, these values differed by more than 10%. CFD simulations were conducted using the γ -Re θ transition model coupled with the k - ω SST turbulence model. Lift coefficients were compared with coefficients of drag, as well as lift coefficients at different angles of attack. It can be concluded that using the transition models for a 2-D simulation achieves satisfactory results.

8.2. 3-D Validation

The 3-D validation was conducted on the HPT's VeloX1 human-powered bicycle. The CFD simulation results were compared with the wind tunnel results conducted on the bicycle by the HPT from the Delft University. Furthermore, results conducted by the same institution using VSAero, were also compared with the CFD results.

As with the 2-D validation, the simulation was conducted using the γ -Re θ transition model coupled with the k - ω SST turbulence model. The transition locations determined by CFD showed similarities to the wind tunnel results, extending to about halfway the body length, but differed in other respects. The value for the drag area (supplied by the Delft University) was larger than that calculated by the CFD. Reasons for this may be attributed to differences in the geometry, such as surface roughness and irregularities due to

manufacturing. The actual model also has ventilation ducts, wheel holes and a windscreen, all contributing factors to higher drag values.

As transition position is a better way to determine the accuracy of the simulations, rather than using the drag area or drag force value, it was assumed that the CFD will predict the laminar flow accurately enough to be used for the purposes of the detail design of the solar-powered vehicle. It would seem that the transition model have been satisfactorily validated for the 3-D simulations of the vehicle.

8.3. Detail Design

This iteration-based methodology whereby the initial design parameters are modified, proved to work well when multiple revisions of the vehicle is used. Analysing each revision and then modifying the design according to the results and the available literature, enabled one to determine the positive or negative changes to the design. This methodology can thus be assumed to be effective in the design of solar-powered vehicles.

The results indicated that CFD is indeed a valuable tool to be used in the design process and is an essential part of the detail design phase as it reduces the need for wind tunnels. It also reduces the design time required as each iteration of the design can be evaluated within a few hours if the resources are available. The results also indicated the importance of using the correct simulation settings. If the vehicle is simulated to be stationary relative to the ground, the calculated drag force will be significantly higher than when the vehicle is simulated to actually move relative to the ground.

8.4. Recommendations for Future Work

Future work can be divided into the different components as analysed in this study as well as the overall design. The main body of the vehicle should be further optimised to extend the laminar flow past the 50% mark, as opposed to the current 35% mark. As indicated in the revisions, the front fairing should be designed such that the rear fairing experiences more laminar flow. The front design should, however, be further analysed as increased laminar flow is possible. A turbulent airfoil for the rear fairing is also an alternative to the current design. The canopy size can be decreased, thus optimised for size and shape, which would further reduce the overall drag of the vehicle.

With the integration of the components, effort should be made to reduce all junctions. This means, aligning the outside edges of the fairing with that of the outside edge of the main body. In this way the junctions are reduced considerably. The leading fillet of the fairing should also be modified to have a gentler slope (in excess of the current 45°) as described in the literature. The trailing edges of the fairings should be sharper. As indicated in the last model, the bottom of the fairings should be rounded. Wheel wells can also be implemented to channel the air that is sucked into the body through the wheel holes to the outside of the body. The overall manufacturing process of the car should be improved to ensure that body panels have smaller seam gaps. It should also be noted that any change in the surface geometry can cause transition of flow. Stickers used for branding should thus be avoided. Any branding on the vehicle should be painted onto the surface of the body to ensure a smooth finish.

Taking into account the above, all of these factors can be optimized simultaneously. This will eliminate the trial and error methodology of adjusting a single variable at a time, as described in this dissertation. This means that a mathematical algorithm can be developed, which will use a number of input variables and vary these parameters simultaneously in order to determine the best possible combination. These parameters are then applied to the model which can be simulated to determine the drag coefficient of the vehicle.

Experimental tests should be developed in order to validate the CFD code with the actual as-built design of the vehicle. It was assumed that the CFD code was accurate to such a degree that it could be used for these simulations. This assumption is based on the 3-D validation process conducted in this dissertation.

Bibliography

- [1] D. W. Kurtz, *Aerodynamic design of electric and hybrid vehicles: a guidebook*, Pasadena: Jet Propulsion Laboratory, 1980.
- [2] CD-adapco, Dynamics:Simulating Systems Issue 33, Melville, NY: CD-adapco, 2013.
- [3] S. i. Sport, “Sasol Solar Challenge,” 2014. [Online]. Available: http://www.solarchallenge.org.za/?page_id=468. [Accessed 17 September 2014].
- [4] Wikipedia, “Tour de Sol,” 2012. [Online]. Available: http://en.wikipedia.org/wiki/Tour_de_Sol. [Accessed 28 February 2013].
- [5] G. Tamai, *The Leading Edge: Aerodynamic Design of Ultra-streamlined Land Vehicles*, 1st ed., Cambridge, USA: Bentley Publishers, 1999.
- [6] H. Smith, *The illustrated guide to aerodynamics*, Blue Ridge Summit, PA: Tab Books, 1992.
- [7] A. Watts, “Semi-truck Boat Tail Improves Fuel Efficiency 7.5%,” 5 November 2009. [Online]. Available: <http://wattsupwiththat.com/2009/11/05/semi-truck-boat-tail-improves-fuel-efficiency-7-5/>. [Accessed 8 April 2013].
- [8] “Aerospaceweb.org,” 1997-2012. [Online]. Available: <http://www.aerospaceweb.org/question/aerodynamics/q0215.shtml>. [Accessed 8 April 2013].
- [9] A. Shapiro, *Shape and Flow: The Fluid Dynamics of Drag*, New York: Anchor Books, 1961.
- [10] R. Beardmore, “Roymech,” 28 January 2013. [Online]. Available: http://www.roymech.co.uk/Related/Fluids/Fluids_Drag.html. [Accessed 09 March 2014].
- [11] S. R. Ahmed, B. Bayer, J. Callister, N. Deußen, H.-J. Emmelmann, H. Flegl, A. George, A. Gilhaus, H. Götz, H. Großmann, R. Hoffmann, W.-H. Hucho, D. Hummel, W. Kohl, G. Mayr, G. Necati, R.

Piatek, D. Schlenz, W. Sebbeße, N. Singer and P. Steinberg, Aerodynamics of road vehicles, 4th ed., W. Hucho, Ed., Warrendale, Pa.: Society of Automotive Engineers, 1998.

[12] VATSIM, “VATSIM,” 2014. [Online]. Available:

http://www.vatnz.net/cms/index.php?option=com_content&task=view&id=181&Itemid=173.

[Accessed 9 March 2014].

[13] A. Filippone, Advanced Aircraft Flight Performance, New York: Cambridge University Press, 2012.

[14] “Cortana Corporation,” [Online]. Available: http://www.cortana.com/Drag_Description.htm.

[Accessed 9 April 2013].

[15] R. B. Langtry, “A Correlation-based Transition Model using Local Variables for Unstructured Parallelized CFD Codes,” Institut für Thermische Strömungsmaschinen und Maschinenlaboratorium: Universität Stuttgart, Stuttgart, 2006.

[16] B. R. Munson, D. F. Young, T. H. Okiishi and W. W. Huebsch, Fundamentals of Fluid Mechanics, John Wiley & Sons (Asia) Pte Ltd, 2010.

[17] “The Free Dictionary,” 2013. [Online]. Available:

<http://encyclopedia2.thefreedictionary.com/Separation+of+Flow>. [Accessed 11 April 2013].

[18] H. Schlichting and K. Gersten, Boundary-layer Theory, Berlin: Springer-Verlag, 2000.

[19] F. Menter, R. Langtry and S. Volker, “Transition Modelling for General Purpose CFD Codes,” *Flow Turbulence Combust*, vol. 77, pp. 277-303, 18 August 2006.

[20] M. Germano, U. Piomelli, P. Moin and W. H. Cabot, “A dynamic subgrid-scale eddy viscosity model,” *Physics of Fluids*, vol. 3, no. 7, pp. 1760-1765, July 1991.

[21] K. Walters and D. Cokljat, “A Three-Equation Eddy-Viscosity Model for Reynolds-Averaged Navier-Stokes Simulations of Transitional Flow,” *Journal of Fluids Engineering*, vol. 130, pp. 121401.1-101401.14, December 2008.

- [22] F. R. Menter, T. Esch and S. Kubacki, "Transition Modelling Based on Local Variables," *Enginnering Turbulence Modelling and Experiments*, vol. 5, pp. 555-564, 16-18 September 2002.
- [23] R. B. Langtry and F. R. Menter, "Transition modeling for General CFD Applications in Aeronautics," in *AIAA Aerospace Sciences Meeting and Exhibit*, Reno, Nevada, USA, 10-13 January 2005.
- [24] F. R. Menter, T. Esch and S. Kubacki, "Transition modelling based on local variables," in *5th International Symposium on Turbulence Modeling and Measurements*, Mallorca, Spain, 2002.
- [25] F. R. Menter, R. B. Langtry and S. R. Likki, "A Correlation-Based Transition Model Using Local Variables - Part 1: Model Formulation," *Journal of Turbomachinery*, vol. 128, no. 3, pp. 413-422, 2006.
- [26] R. B. Langtry, F. R. Menter, S. R. Likki, Y. B. Suzen, P. G. Haung and S. Volker, "A Correlation-Based Transition Model Using Local Variables - Part 2: Test Cases and Industrial Applications," *Journal of Turbomachinery*, vol. 128, no. 3, pp. 423-434, July 2006.
- [27] S. Mansor, N. A. R. Nik Mohd and C. W. Chung, "Validation of CFD Modeling and Simulation of a Simplified Automotive Model," *Applied Mechanics and Materials*, vol. 735, pp. 319-325, 2015.
- [28] V. K. Yakkundi and S. S. Mantha, "Computational Analysis of "Drag & Lift" of a car With K- ϵ std, and K- ω SST (Shear Stress Transport) Turbulence Models & Effect of Mesh Refinement," *Curie*, vol. 3, no. 3 and 4, pp. 117-129, 2010-2011.
- [29] D. R. Carroll, *The Winning Solar Car*, Warrendale: SAE International, 2003.
- [30] G. Buresti, G. V. Iungo and G. Lombardi, "Methods for the drag reduction of bluff bodies and their application to heavy road-vehicles," *DDIA*, vol. 6, p. 4, 2007.
- [31] B. W. McCormick, "Static Stability and Control," in *Aerodynamics, Aeronautics, and Flight Mechanics*, John Wiley & Sons, Inc., 1995, pp. 473-483.

- [32] “Longitudinal Static Stability,” Stanford, 04 January 2005. [Online]. Available: <http://adg.stanford.edu/aa241/stability/staticstability.html>. [Accessed 21 October 2014].
- [33] “Chapter 7: Fixed-Wing Aerodynamics and Performance,” Train Army, 27 12 2012. [Online]. Available: <https://rdl.train.army.mil/catalog-ws/view/100.ATSC/124A2C90-72CD-4B05-AB8B-ABC8718F9760-1274574464617/3-04.203/chap7.htm>. [Accessed 21 October 2014].
- [34] K. E. Kenyon, “Curvature Boundary Layer,” *Physics Essays*, vol. 16, no. 1, pp. 74-85, 2003.
- [35] S. Ruitenbergh, “S&D Tech Blog,” 14 February 2015. [Online]. Available: http://sdtechblog.weebly.com/uploads/6/5/2/5/6525158/7411858_orig.jpg. [Accessed 26 February 2015].
- [36] L. M. M. Boermans, F. Nicolosi and K. Kubrynski, “Aerodynamic Design of High-performance Sailplane Wing-fuselage Combinations,” in *21st ICAS Conference*, Melbourne, 13-18 September 1998.
- [37] R. L. Simpson, “Junction Flows,” *Annual Review of Fluid Mechanics*, vol. 33, pp. 415-443, 2001.
- [38] A. Ahmed and M. J. Khan, “Effect of sweep on wing-body juncture flows,” in *33rd American Institute of Aeronautics and Astronautics Aerospace Sciences Meeting*, Reno, 1995.
- [39] S. F. Hoerner, *Fluid-Dynamic Drag*, Vancouver: Published by author, 1965.
- [40] S. F. Hoerner, *Fluid-dynamic drag : practical information on aerodynamic drag and hydrodynamic resistance*, Bakersfield, California: Hoerner Fluid Dynamics, 1992.
- [41] M. Maughmer, D. Hallman, R. Ruszkowski, G. Chappel and I. Waitz, “Experimental Investigation of Wing/Fuselage Integration Geometries,” *Journal of Aircraft*, vol. 26, no. 8, 1989.
- [42] “World Solar Challenge,” [Online]. Available: http://www.worldsolarchallenge.org/files/136_solar_team_twente.large.jpg. [Accessed 2015 February 2015].

- [43] “Nuon Solar Team,” 2012. [Online]. Available: <http://www.nuonsolarteam.nl/?lang=en>. [Accessed 20 March 2012].
- [44] N. S. Team, 8 August 2013. [Online]. Available: <http://www.flickr.com/photos/nuonsolarteam/9814081074/>. [Accessed 23 January 2014].
- [45] J. Haringman, “solarracing.org,” 2013. [Online]. Available: <http://www.solarracing.org/2013/08/29/tokai-university-reveals-their-car-challenger/>. [Accessed 11 March 2014].
- [46] T. Abe and P. Starr, “Evolving an Engineering Design Methodology,” in *International Conference of Engineering Education*, Budapest, 2008.
- [47] D. G. Ullman, *The Mechanical Design Process*, New York: McGraw-Hill, 1992.
- [48] B. E. Kissinger, “Aerodynamic optimization of a solar race car body shape,” UMI, Ann Arbor, 1997.
- [49] AAIA, “Guide for the verification and validation of computational fluid dynamic simulations,” *American Institute of Aeronautics and Astronautics*, Vols. AIAA-G-077-1998, 1998.
- [50] J. R. Grace and F. Taghipour, “Verification and validation of CFD models and dynamic similarity,” *Elsevier*, vol. 139, no. Powder Technology, pp. 99-110, 2004.
- [51] “Profil 2,” 2015. [Online]. Available: <http://www.profil2.com/eng/default.asp>. [Accessed 2015 February 23].
- [52] W. A. Timmer, “Two-dimensional low-Reynolds number wind tunnel results for airfoil NACA 0018,” *Wind Engineering*, vol. 32, no. 6, pp. 525-537, 2008.
- [53] CD-adapco, “The world's most comprehensive engineering simulation inside a single integrated package,” 2013. [Online]. Available: <http://www.cd-adapco.com/products/star-ccm-plus>. [Accessed 14 5 2013].

- [54] K. Maley, "Best Practices: Volume Meshing [PowerPoint]," CD-adapco, 2012.
- [55] N. E. Ahmad, E. Abo-Serie and A. Gaylard, "Mesh Optimization for Ground Vehicle Aerodynamics," vol. 2, no. 1, 2010.
- [56] U. o. Strathclyde, "Sustainable Engineering," [Online]. Available:
http://www.esru.strath.ac.uk/EandE/Web_sites/09-10/MCT/html/Technical/rfoil.html. [Accessed 2015 February 23].
- [57] K. Epema, Interviewee, [Interview]. 26 June 2013.
- [58] B. Maskew, "Program VSAERO Theory Document," National Aeronautics and Space Administration, Redmond, Washington, 1987.

Annexure A: 2-D Validation Summary Report

This section lists the settings used in STAR-CCM+ to simulate a NACA 0018 airfoil on a 2-D mesh. All the settings described in the respective chapter of this dissertation correlates with the settings described in this section.

Summary Report: star_NACA_0018_2D_rev3_0deg

Session Summary

Date 28 Jul 2014 3:02:55 PM
Simulation E:\Akademie\M.Ing Meganies\Maerofoil\VALIDATION\NACA 0018\soos
artikel\star_NACA_0018_2D_rev3_0deg.sim
File size 15 MB
Number of Partitions 1
Number of Restored Partitions 1
Software Summary
Version BuildArch: win64
BuildEnv: intel12.1
ReleaseDate: Thu Jan 31 21:38:48 UTC 2013
ReleaseNumber: 8.02.008

Hardware Summary

Hosts Controller: Armand-PC.nwu.ac.za
Number of Workers: 0

Simulation Properties

1 star_NACA_0018_2D_rev3_0deg			
+-1 Tags			
+-2 3D-CAD Models			
+-3 Filters			
+-4 Parts			
+-1 Background		Contacts	[]
		Descriptions	[Ljava.lang.String;@6aa11982
		Face count	290
		Index	3
		Tags	[]
		Meta Data	{}
+-1 Surfaces			
+-1 Inlet		Tags	[]
		Meta Data	{}
+-2 Outlet		Tags	[]
		Meta Data	{}
+-3 Sides		Tags	[]
		Meta Data	{}
^-4 SP		Tags	[]
		Meta Data	{}
^-2 Curves			
+-1 Edges		Tags	[]
^-2 Intersection		Tags	[]
+-2 Block		Corner 1	[-0.05, -0.05, -5.9396931817445875E-15] m,m,m
		Corner 2	[0.3, 0.05, 0.8999999761581421] m,m,m
		Coordinate System	Laboratory
		Contacts	[]
		Descriptions	[Ljava.lang.String;@64a1d02b
		Face count	12
		Index	2
		Tags	[]
		Meta Data	{}
+-1 Surfaces			
^-1 Block Surface		Tags	[]
		Meta Data	{}
^-2 Curves			
^-1 Block Curve		Tags	[]
+-3 Overset		Contacts	[]
		Descriptions	[Ljava.lang.String;@3b69eaaa
		Face count	1114
		Index	4
		Tags	[]
		Meta Data	{}
+-1 Surfaces			
+-1 Airfoil		Tags	[]
		Meta Data	{}
+-2 Block Surface		Tags	[]
		Meta Data	{}
^-3 SP		Tags	[]
		Meta Data	{}

		^-2 Curves		
		+--1 Block Curve	Tags	[]
		+--2 Edges	Tags	[]
		^-3 Intersection	Tags	[]
		^-4 WindTunnel+NACA0018	Contacts	[]
			Descriptions	[Ljava.lang.String;@26ad5d5f
			Face count	860
			Index	1
			Tags	[]
			Meta Data	{}
		+--1 Surfaces		
		+--1 Airfoil	Tags	[]
			Meta Data	{}
		+--2 Inlet	Tags	[]
			Meta Data	{}
		+--3 Outlet	Tags	[]
			Meta Data	{}
		+--4 Sides	Tags	[]
			Meta Data	{}
		^-5 SP	Tags	[]
			Meta Data	{}
		^-2 Curves		
		^-1 Edges	Tags	[]
		+--5 Operations		
		+--6 Descriptions	Number of children	1
		^-1 Root	Described Parts	[Ljava.lang.String;@419272a9
		+--7 Contacts		
		+--8 Continua	Continua	2
		+--1 Mesh 1	Use Parallel Meshing	false
			Verbose Output	false
			OOO translation	false
			Per-Region Meshing	true
			Interpolation Option	Nearest neighbor
			Interfaces	[Overset Mesh 1]
			Regions	[Overset 2D, Background 2D]
		+--1 Models		
		+--1 Prism Layer Mesher	Stretching Function	Geometric Progression
			Stretching Mode	Wall Thickness
			Gap Fill Percentage	25.0
			Minimum Thickness Percentage	10.0
			Layer Reduction Percentage	50.0
			Boundary March Angle	50.0
			Concave Angle Limit	0.0
			Convex Angle Limit	360.0
			Near Core Layer Aspect Ratio	0.0
			Generate Standard Cells Only	false
			Improve Subsurface Quality	true
		+--2 Surface Remesher	Do curvature refinement	true
			Do proximity refinement	false
			Do compatibility refinement	false
			Retain geometric features	true
			Create aligned meshes	true
			Minimum Face Quality	0.0
			Enable automatic surface repair	false
		^-3 Trimmer	Coordinate System	Laboratory
			Do mesh alignment	false
			Template mesh type	Hexahedra
			Template mesh growth type	Simple
			Run Optimizer	true
		+--2 Reference Values		
		+--1 Base Size	Value	55.0 mm
		+--2 CAD Projection	Project to CAD	true
		+--3 Maximum Cell Size	Size type	Relative to base
		^-1 Relative Size	Percentage of Base	100.0
			Absolute Size	55.0 mm
		+--4 Maximum Core/Prism Transition Ratio	Limit cell size by prism layer thickness	false
		+--5 Number of Prism Layers	Number of Prism Layers	15
		+--6 Prism Layer Thickness	Size type	Absolute
		^-1 Absolute Size	Value	2.5 mm
		+--7 Surface Curvature	Enable curvature deviation distance	false
		^-1 Basic Curvature	# Pts/circle	36.0
		+--8 Surface Growth Rate	Surface Growth Rate	1.3
		+--9 Surface Size	Relative/Absolute	Relative to base
			Size Method	Min and Target
		+--1 Relative Minimum Size	Percentage of Base	5.0
			Absolute Size	2.75 mm
		^-2 Relative Target Size	Percentage of Base	100.0

			Absolute Size	55.0 mm
		+10 Template Growth Rate	Default Growth Rate	Slow
			Boundary Growth Rate	Slow
		^-11 Thickness of Near Wall Prism Layer	Value	0.005 mm
		^-3 Volumetric Controls		
		+1 Volumetric Control 1	Part Group	[]
			Shapes	[Overlap]
		+1 Mesh Conditions		
		+1 Prism Layer Mesher	Customize prism layer mesher	Disabled
		+2 Surface Remesher	Customize surface remesher	Enabled
		^-3 Trimmer	Customize isotropic size	Enabled
			Customize anisotropic size	Disabled
		^-2 Mesh Values		
		^-1 Custom Size	Size type	Relative to base
		^-1 Relative Size	Percentage of Base	10.0
			Absolute Size	5.5 mm
		+2 Volumetric Control 2	Part Group	[]
			Shapes	[Refine1]
		+1 Mesh Conditions		
		+1 Prism Layer Mesher	Customize prism layer mesher	Disabled
		+2 Surface Remesher	Customize surface remesher	Enabled
		^-3 Trimmer	Customize isotropic size	Enabled
			Customize anisotropic size	Disabled
		^-2 Mesh Values		
		^-1 Custom Size	Size type	Relative to base
		^-1 Relative Size	Percentage of Base	20.0
			Absolute Size	11.0 mm
		+3 Volumetric Control 3	Part Group	[]
			Shapes	[Refine2]
		+1 Mesh Conditions		
		+1 Prism Layer Mesher	Customize prism layer mesher	Disabled
		+2 Surface Remesher	Customize surface remesher	Enabled
		^-3 Trimmer	Customize isotropic size	Enabled
			Customize anisotropic size	Disabled
		^-2 Mesh Values		
		^-1 Custom Size	Size type	Relative to base
		^-1 Relative Size	Percentage of Base	30.0
			Absolute Size	16.5 mm
		^-4 Volumetric Control 4	Part Group	[]
			Shapes	[Refine3]
		+1 Mesh Conditions		
		+1 Prism Layer Mesher	Customize prism layer mesher	Disabled
		+2 Surface Remesher	Customize surface remesher	Enabled
		^-3 Trimmer	Customize isotropic size	Enabled
			Customize anisotropic size	Disabled
		^-2 Mesh Values		
		^-1 Custom Size	Size type	Relative to base
		^-1 Relative Size	Percentage of Base	40.0
			Absolute Size	22.0 mm
		^-2 Physics 1 2D	Interfaces	[Overset Mesh 1]
			Regions	[Overset 2D, Background 2D]
		+1 Models		
		+1 All y+ Wall Treatment	Iterative Ustar	false
		+2 Constant Density		
		+3 Gamma ReTheta	Convection	2nd-order
			Sigma_f	1.0
			ca1	2.0
			ca2	0.06
			ce1	1.0
			ce2	50.0
			cTheta_t	0.03
			s1	2.0
			Secondary Gradients	On
			Correlation Method	Suluksna-Juntasaro
			Sigma_ReTheta	2.0
			Intermittency Minimum	1.0E-10
			ReThetaT Minimum	20.0
			Free Stream Edge	UserFreeStreamEdge
		+4 Gas		
		^-1 Air	Database Material	Air (Air) [Standard/Gases]
		^-1 Material Properties		
		+1 Density	Method	Constant
		^-1 Constant	Value	1.225 kg/m^3
		^-2 Dynamic Viscosity	Method	Constant
		^-1 Constant	Value	1.75E-5 Pa-s
		+5 Gradients	Verbose	false
			Gradient Method	Hybrid Gauss-LSQ

			Limiter Method	Venkatakishnan
			Least-Squares Quality Criterion	true
			Cell Skewness Criterion	true
			Chevron-Cell Criterion	true
			Least-Squares Tensor Minimum Eigenvalues Ratio	0.1
			Normalized Flat Cells Curvature Factor	1.0
			Maximum safe (positive) skewness angle (deg.)	75.0
			Minimum unsafe (positive) skewness angle (deg.)	88.0
			Flat Cells Curvature Criterion	true
			+-6 K-Omega Turbulence	
			+-7 Reynolds-Averaged Navier-Stokes	
			+-8 Segregated Flow	
			Convection	2nd-order
			Flow Boundary Diffusion	true
			Minimum Absolute Pressure	1000.0 Pa
			Secondary Gradients	On
			Delta-V Dissipation	Off
			Option	None
			+-9 SST (Menter) K-Omega	
			a1	0.31
			Kappa	0.41
			BetaStar	0.09
			Beta1	0.075
			Sigma_k1	0.85
			Sigma_w1	0.5
			Beta2	0.0828
			Sigma_k2	1.0
			Sigma_w2	0.856
			Convection	2nd-order
			Secondary Gradients	On
			Curvature Correction Option	Off
			Realizability Option	Durbin Scale Limiter
			Compressibility Correction	true
			Low Re Damping Modification	false
			Normal Stress Term	false
			Tke Minimum	1.0E-10
			Sdr Minimum	1.0E-10
			Zeta_Star	1.5
			+-1 Compressibility Parameters	
			+-2 Realizability Coefficient	
			Realizability Coefficient	0.6000000238418579
			+-10 Steady	
			+-11 Turbulent	
			+-12 Two Dimensional	
			+-2 Reference Values	
			+-1 Minimum Allowable Wall Distance	
			Value	1.0E-6 m
			+-2 Reference Pressure	
			Value	101325.0 Pa
			+-3 Initial Conditions	
			+-1 Intermittency	
			Method	Constant
			+-1 Constant	
			Value	1.0
			+-2 Pressure	
			Method	Constant
			+-1 Constant	
			Value	0.0 Pa
			+-3 Turbulence Intensity	
			Method	Constant
			+-1 Constant	
			Value	0.01
			+-4 Turbulence Specification	
			Method	Intensity + Viscosity Ratio
			+-5 Turbulent Velocity Scale	
			Method	Constant
			+-1 Constant	
			Value	1.0 m/s
			+-6 Turbulent Viscosity Ratio	
			Method	Constant
			+-1 Constant	
			Value	10.0
			+-7 Velocity	
			Coordinate System	Laboratory
			Method	Constant
			+-1 Constant	
			Value	[0.0, 0.0, 0.0] m/s
			+-9 Regions	
			Regions	2
			+-1 Background 2D	
			Index	3
			Physics Continuum	Physics 1 2D
			Type	Fluid Region
			Mesh Continuum	Mesh 1
			Parts	[]
			Boundaries	3
			+-1 Inlet	
			Index	18
			Interfaces	
			Type	Velocity Inlet
			Part Surfaces	[]
			+-1 Mesh Conditions	
			+-1 Custom Boundary Growth Rate	
			Custom Boundary Growth Rate	Disabled
			+-2 Custom Surface Curvature	
			Custom curvature	Use Continuum Values
			+-3 Custom Surface Size	
			Custom surface size	Disabled
			+-4 Customize Prism Mesh	
			Customize Prism Mesh	Use Default Values
			+-5 Customize Surface Remeshing	
			Disable surface remeshing	Disabled

				+-2	Physics Conditions		
				+-1	Flow Direction Specification	Method	Components
				+-2	Reference Frame Specification	Option	Lab Frame
				+-3	Turbulence Specification	Method	Intensity + Viscosity Ratio
				^-4	Velocity Specification	Method	Magnitude + Direction
				^-3	Physics Values		
				+-1	Flow Direction	Coordinate System	Laboratory
						Method	Constant
					^-1	Constant	Value
							[1.0, 0.0, 0.0]
				+-2	Turbulence Intensity	Method	Constant
					^-1	Constant	Value
							0.01
				+-3	Turbulent Viscosity Ratio	Method	Constant
					^-1	Constant	Value
							10.0
				^-4	Velocity Magnitude	Method	Constant
					^-1	Constant	Value
							40.0 m/s
				+-2	Outlet	Index	20
						Interfaces	
						Type	Pressure Outlet
						Part Surfaces	[]
				+-1	Mesh Conditions		
				+-1	Custom Boundary Growth Rate	Custom Boundary Growth Rate	Disabled
				+-2	Custom Surface Curvature	Custom curvature	Use Continuum Values
				+-3	Custom Surface Size	Custom surface size	Disabled
				+-4	Customize Prism Mesh	Customize Prism Mesh	Use Default Values
				^-5	Customize Surface Remeshing	Disable surface remeshing	Disabled
				+-2	Physics Conditions		
				+-1	Backflow Direction Specification	Method	Boundary-Normal
				+-2	Pressure Specification	Pressure specification	Environmental
				+-3	Target Mass Flow Option	Target Mass Flow Option	Disabled
				^-4	Turbulence Specification	Method	Intensity + Viscosity Ratio
				^-3	Physics Values		
				+-1	Pressure	Method	Constant
					^-1	Constant	Value
							0.0 Pa
				+-2	Turbulence Intensity	Method	Constant
					^-1	Constant	Value
							0.01
				^-3	Turbulent Viscosity Ratio	Method	Constant
					^-1	Constant	Value
							10.0
				^-3	Sides	Index	19
						Interfaces	
						Type	Wall
						Part Surfaces	[]
				+-1	Mesh Conditions		
				+-1	Custom Boundary Growth Rate	Custom Boundary Growth Rate	Disabled
				+-2	Custom Surface Curvature	Custom curvature	Use Continuum Values
				+-3	Custom Surface Size	Custom surface size	Disabled
				+-4	Customize Prism Mesh	Customize Prism Mesh	Use Default Values
				^-5	Customize Surface Remeshing	Disable surface remeshing	Disabled
				+-2	Physics Conditions		
				+-1	Shear Stress Specification	Method	No-Slip
				+-2	Tangential Velocity Specification	Method	None
						Reference Frame	Relative To Mesh
				^-3	Wall Surface Specification	Method	Smooth
				^-3	Physics Values		
				^-1	Blended Wall Function	Kappa	0.42
						E	9.0
							0
				+-2	Feature Curves	Feature Curves	
				+-3	Mesh Conditions		
				^-1	Customize Prism Mesh	Customize Prism Mesh	Use Default Values
				+-4	Mesh Values		
				^-1	Trimmer Wake Refinement		
				+-5	Physics Conditions		
				+-1	Initial Condition Option	Option	Use Continuum Values
				+-2	Mass Source Option	Mass Source Option	Disabled
				+-3	Momentum Source Option	Momentum Source Option	None
				^-4	Turbulence Source Option	Turbulence Source Option	None
				^-6	Physics Values		
				+-1	Axis	Direction	[0.0, 0.0, 1.0]
						Coordinate System	Laboratory
						Origin	[0.0, 0.0, 0.0] m
				^-2	Motion Specification	Motion	Stationary
						Reference Frame	Lab Reference Frame
				^-2	Overset 2D	Index	2
						Physics Continuum	Physics 1 2D
						Type	Fluid Region
						Mesh Continuum	Mesh 1
						Parts	[]
				+-1	Boundaries	Boundaries	2

		+-1	Airfoil	Index	16
				Interfaces	
				Type	Wall
				Part Surfaces	[]
		+-1	Mesh Conditions		
			+-1 Custom Boundary Growth Rate	Custom Boundary Growth Rate	Disabled
			+-2 Custom Surface Curvature	Custom curvature	Use Continuum Values
			+-3 Custom Surface Size	Custom surface size	Enabled
			+-4 Customize Prism Mesh	Customize Prism Mesh	Use Default Values
			^-5 Customize Surface Remeshing	Disable surface remeshing	Disabled
		+-2	Mesh Values		
			^-1 Surface Size	Relative/Absolute	Relative to base
				Size Method	Min and Target
			+-1 Relative Minimum Size	Percentage of Base	0.1
				Absolute Size	0.05500000000000001 mm
			^-2 Relative Target Size	Percentage of Base	2.0
				Absolute Size	1.1 mm
		+-3	Physics Conditions		
			+-1 Shear Stress Specification	Method	No-Slip
			+-2 Tangential Velocity Specification	Method	None
				Reference Frame	Relative To Mesh
			^-3 Wall Surface Specification	Method	Smooth
		^-4	Physics Values		
			^-1 Blended Wall Function	Kappa	0.42
				E	9.0
		^-2	Block Surface	Index	17
				Interfaces	
				Type	Overset Mesh
				Part Surfaces	[]
		^-1	Mesh Conditions		
			+-1 Custom Boundary Growth Rate	Custom Boundary Growth Rate	Disabled
			+-2 Custom Surface Curvature	Custom curvature	Use Continuum Values
			+-3 Custom Surface Size	Custom surface size	Disabled
			+-4 Customize Prism Mesh	Customize Prism Mesh	Use Default Values
			^-5 Customize Surface Remeshing	Disable surface remeshing	Disabled
		+-2	Feature Curves	Feature Curves	0
		+-3	Mesh Conditions		
			^-1 Customize Prism Mesh	Customize Prism Mesh	Use Default Values
		+-4	Mesh Values		
			^-1 Trimmer Wake Refinement		
		+-5	Physics Conditions		
			+-1 Initial Condition Option	Option	Use Continuum Values
			+-2 Mass Source Option	Mass Source Option	Disabled
			+-3 Momentum Source Option	Momentum Source Option	None
			^-4 Turbulence Source Option	Turbulence Source Option	None
		^-6	Physics Values		
			+-1 Axis	Direction	[0.0, 0.0, 1.0]
				Coordinate System	Laboratory
				Origin	[0.0, 0.0, 0.0] m
			^-2 Motion Specification	Motion	Stationary
				Reference Frame	Lab Reference Frame
+-10			Interfaces	Interfaces	1
				Verbosity	false
		^-1	Overset Mesh 1	Region-0	Overset 2D
				Region-1	Background 2D
				Close Proximity	false
				Verbosity	None
				Type	Overset Mesh Interface
		^-1	Physics Conditions		
			^-1 Interpolation Option	Option	Distance weighted
+-11			Derived Parts	Derived Parts	0
+-12			Solvers		
		+-1	Wall Distance	Verbosity	0
				Parallel memory optimization scaling factor	1.0
				Solver Frozen	false
		+-2	Segregated Flow	Reconstruction Frozen	false
				Temporary Storage Retained	false
				Reconstruction Zeroed	false
				Solver Frozen	false
			+-1 Velocity	Under-Relaxation Factor	0.7
			+-1 Under-Relaxation Factor Ramp	Ramp Method	No Ramp
			^-2 AMG Linear Solver	Verbosity	None
				Epsilon	0.0
				Group Size	2
				Parallel Migration Limit	25
				Enable direct-solver	false

			Maximum direct-solver equations	32
			Convergence Tolerance	0.1
			Cycle Type	Flex Cycle
			Group Size Control	Auto
			Relaxation Scheme	Gauss-Seidel
			Scaling	Disabled
			Subdomain coarsening enabled	true
			Acceleration method	None
			Max Cycles	30
		~-1 Flex Cycle	Sweeps	1
			Restriction Tolerance	0.9
			Prolongation Tolerance	0.5
		~-2 Pressure	Pressure Reference Location	Automatic Selection
			Under-Relaxation Factor	0.3
		+~1 Under-Relaxation Factor Ramp	Ramp Method	No Ramp
		~-2 AMG Linear Solver	Verbosity	None
			Epsilon	0.0
			Group Size	2
			Parallel Migration Limit	25
			Enable direct-solver	false
			Maximum direct-solver equations	32
			Convergence Tolerance	0.1
			Cycle Type	F Cycle
			Group Size Control	Auto
			Relaxation Scheme	Gauss-Seidel
			Scaling	Auto
			Subdomain coarsening enabled	true
			Acceleration method	None
			Max Cycles	30
		~-1 F Cycle	Pre-Sweeps	0
			Max Levels	50
			Post-Sweeps	2
		+~3 K-Omega Turbulence	Under-Relaxation Factor	0.8
			Reconstruction Frozen	false
			Temporary Storage Retained	false
			Reconstruction Zeroed	false
			Solver Frozen	false
		+~1 Under-Relaxation Factor Ramp	Ramp Method	No Ramp
		~-2 AMG Linear Solver	Verbosity	None
			Epsilon	0.0
			Group Size	2
			Parallel Migration Limit	25
			Enable direct-solver	false
			Maximum direct-solver equations	32
			Convergence Tolerance	0.1
			Cycle Type	Flex Cycle
			Group Size Control	Auto
			Relaxation Scheme	Gauss-Seidel
			Scaling	Disabled
			Subdomain coarsening enabled	true
			Acceleration method	None
			Max Cycles	30
		~-1 Flex Cycle	Sweeps	1
			Restriction Tolerance	0.9
			Prolongation Tolerance	0.5
		+~4 K-Omega Turbulent Viscosity	Under-Relaxation Factor	1.0
			Maximum Ratio	100000.0
			Solver Frozen	false
		~-5 GammaReTheta Transition	Freestream Update Policy	Iteration
			Freestream Update Frequency	1
			Under-Relaxation Factor	0.8
			Reconstruction Frozen	false
			Temporary Storage Retained	false
			Reconstruction Zeroed	false
			Solver Frozen	false
		+~1 Under-Relaxation Factor Ramp	Ramp Method	No Ramp
		~-2 AMG Linear Solver	Verbosity	None
			Epsilon	0.0
			Group Size	2
			Parallel Migration Limit	25
			Enable direct-solver	false
			Maximum direct-solver equations	32
			Convergence Tolerance	0.1
			Cycle Type	Flex Cycle
			Group Size Control	Auto
			Relaxation Scheme	Gauss-Seidel
			Scaling	Disabled
			Subdomain coarsening enabled	true

		Acceleration method	None
		Max Cycles	30
	+-1 Flex Cycle	Sweeps	1
		Restriction Tolerance	0.9
		Prolongation Tolerance	0.5
+-13 Stopping Criteria			
	+-1 Maximum Steps	Maximum Steps	2000
		Enabled	true
		Criterion Satisfied	true
		Logical Rule	Or
	+-2 Stop File	Stop Inner Iterations	true
		Path	ABORT
		Enabled	true
		Criterion Satisfied	false
		Logical Rule	Or
+-14 Solution Histories			
+-15 Solution Views			
	+-1 Current Solution	Time Step	0
		Solution Time	0.0
		Iteration	2000
+-16 Reports			
	+-1 Drag Coefficient	Reports	2
		Reference Density	1.225 kg/m^3
		Reference Velocity	40.0 m/s
		Reference Area	0.25 m^2
		Force Option	Pressure + Shear
		Direction	[1.0, 0.0, 0.0]
		Reference Pressure	0.0 Pa
		Number of Bands	0
		Coordinate System	Laboratory
		Representation	Volume Mesh
		Parts	[Overset 2D: Airfoil]
		Smooth Values	false
		Units	
	+-2 Lift Coefficient	Reference Density	1.225 kg/m^3
		Reference Velocity	40.0 m/s
		Reference Area	0.25 m^2
		Force Option	Pressure + Shear
		Direction	[0.0, 1.0, 0.0]
		Reference Pressure	0.0 Pa
		Number of Bands	0
		Coordinate System	Laboratory
		Representation	Volume Mesh
		Parts	[Overset 2D: Airfoil]
		Smooth Values	false
		Units	
+-17 Monitors			
		Monitors	10
		Monitors To Print	[Continuity, X-momentum, Y-momentum, Tke, Sdr, Intermittency, ReTheta_t, Lift Coefficient Monitor, Drag Coefficient Monitor]
		Output Direction	Horizontal
		Heading Print Frequency	10
	+-1 Drag Coefficient Monitor	Report	Drag Coefficient
		Maximum Plot Samples	5000
		Normalization Option	Off
		Trigger	Iteration
		Frequency	1
	+-2 Iteration	Frequency	1
		Normalization Option	Off
		Trigger	Iteration
		Maximum Plot Samples	5000
	+-3 Lift Coefficient Monitor	Report	Lift Coefficient
		Maximum Plot Samples	5000
		Normalization Option	Off
		Trigger	Iteration
		Frequency	1
+-18 Representations			
	+-1 Geometry		
	+-2 Volume Mesh	Interior Faces	173190
		Vertices	87945
		Cells	86458
	+-1 Finite Volume Regions		
	+-1 Background 2D	Interior Faces	133674
		Vertices	67296
		Cells	66632
	+-1 Finite Volume Boundaries		
	+-1 Inlet	Faces	56
	+-2 Outlet	Faces	99

^-3 Sides	Faces	98
^-2 Overset 2D	Interior Faces	39516
	Vertices	20649
	Cells	19826
^-1 Finite Volume Boundaries		
+-1 Airfoil	Faces	695
^-2 Block Surface	Faces	264
^-2 Cell Sets		
+-3 Overset Mesh		
+-4 Initial Surface	Edges	0
	Faces	0
^-1 Regions		
^-5 Remeshed Surface	Edges	0
	Faces	0
^-1 Regions		
+-19 Coordinate Systems		
^-1 Laboratory		
^-1 Local Coordinate Systems		
+-20 Units	Preferred System	Système Internationale
+-21 Tables	Tables	0
+-22 Field Functions		
+-1 UserFreeStreamEdge	Type	Scalar
	Function Name	UserFreeStreamEdge
	Definition	(\$WallDistance<0.0025 && \$\$Centroid[1]<0.15 && \$\$Centroid[1]>0.15) ? 1:0
	Ignore Boundary Values	false
	Assembly code	(if (&& (&& (< \$WallDistance) 0.0025) < \$\$Centroid[1] 0.15)) (> \$\$Centroid[1] 0.15)) 1 0)
	Inverse Distance Weight	false
+-23 Volume Shapes		
+-1 Overlap	Coordinate System	Laboratory
	Corner1	[-0.35, -0.3, -5.9396931817445875E-15] m,m,m
	Corner2	[0.55, 0.3, 0.2] m,m,m
+-2 Refine1	Coordinate System	Laboratory
	Corner1	[0.54, -0.3, -5.9396931817445875E-15] m,m,m
	Corner2	[1.05, 0.3, 0.2] m,m,m
+-3 Refine2	Coordinate System	Laboratory
	Corner1	[1.04, -0.3, -5.9396931817445875E-15] m,m,m
	Corner2	[1.4, 0.3, 0.2] m,m,m
^-4 Refine3	Coordinate System	Laboratory
	Corner1	[1.39, -0.3, -5.9396931817445875E-15] m,m,m
	Corner2	[1.8, 0.3, 0.2] m,m,m
+-24 Data Set Functions	Data Directory	function_data
+-25 User Code		
+-26 Layouts		
^-1 default		
+-27 Data Mappers		
+-28 Motions		
^-1 Stationary		
+-29 Reference Frames		
^-1 Lab Reference Frame		

Solution

Accumulated CPU Time over all processes (s) 1698.8340000000162
Elapsed Time (s) 1698.834000000014
Iterations 2000

Annexure B: 3-D Validation Summary Report

As in the previous section, the settings listed in this summary report correlates with the simulation settings used in STAR-CCM+ to simulate the flow around a 3-D body, the VeloX1 human powered bicycle.

Summary Report: Velox_rev9.0_150

Session Summary

Date 30 Jul 2014 10:13:44 AM
Simulation C:\Users\Armand\Documents\M.Ing Meganies\Meesters\Detail Design\Velox_rev9.0_150.sim
File size 3.9e+02 MB
Number of Partitions 1
Number of Restored Partitions 1

Software Summary

Version BuildArch: win64
BuildEnv: intel12.1
ReleaseDate: Thu Jan 31 21:38:48 UTC 2013
ReleaseNumber: 8.02.008

Hardware Summary

Hosts Controller: Armand-PC.nwu.ac.za
Number of Workers: 0

Simulation Properties

1	Velox_rev9.0_150		
+-1	Tags		
+-2	3D-CAD Models		
+-3	Filters		
+-4	Parts		
	+-1 Windtonnel_rect	Region	Region 1
		Contacts	{}
		Descriptions	[Ljava.lang.String;@3690fae2
		Face count	91252
		Index	1
		Tags	{}
		Meta Data	{}
	+-1 Surfaces		
		+-1 Inlet	Tags {}
			Meta Data {}
			Boundary Region 1: Windtonnel_rect.Inlet
		+-2 Outlet	Tags {}
			Meta Data {}
			Boundary Region 1: Windtonnel_rect.Outlet
		+-3 SP	Tags {}
			Meta Data {}
			Boundary Region 1: Windtonnel_rect.SP
		+-4 VeloX1	Tags {}
			Meta Data {}
			Boundary Region 1: Windtonnel_rect.VeloX1
		+-5 Walls	Tags {}
			Meta Data {}
			Boundary Region 1: Windtonnel_rect.Walls
	+-2 Curves		
		+-1 Edges	Feature Curve Region 1:Default Feature Curve
			Tags {}
+-5	Operations		
+-6	Descriptions	Number of children	1
	+-1 Root	Described Parts	[Ljava.lang.String;@109b77d6
+-7	Contacts		
+-8	Continua	Continua	2
	+-1 Mesh 1	Use Parallel Meshing	false
		Verbose Output	false
		OOB translation	false
		Per-Region Meshing	false
		Interpolation Option	Nearest neighbor
		Interfaces	{}
		Regions	[Region 1]
	+-1 Models		
		+-1 Prism Layer Mesher	Stretching Function Geometric Progression
			Stretching Mode Stretch Factor
			Gap Fill Percentage 25.0
			Minimum Thickness Percentage 10.0
			Layer Reduction Percentage 50.0
			Boundary March Angle 50.0
			Concave Angle Limit 0.0
			Convex Angle Limit 360.0
			Near Core Layer Aspect Ratio 0.0
			Generate Standard Cells Only false
			Improve Subsurface Quality true

			+-2	Surface Remesher	Do curvature refinement	true
					Do proximity refinement	false
					Do compatibility refinement	false
					Retain geometric features	true
					Create aligned meshes	true
					Minimum Face Quality	0.0
					Enable automatic surface repair	false
			^-3	Trimmer	Coordinate System	Laboratory
					Do mesh alignment	false
					Template mesh type	Hexahedra
					Template mesh growth type	Simple
					Run Optimizer	true
			+-2	Reference Values		
			+-1	Base Size	Value	150.0 mm
			+-2	CAD Projection	Project to CAD	true
			+-3	Maximum Cell Size	Size type	Relative to base
			^-1	Relative Size	Percentage of Base	100.0
					Absolute Size	150.0 mm
			+-4	Maximum Core/Prism Transition Ratio	Limit cell size by prism layer thickness	false
			+-5	Number of Prism Layers	Number of Prism Layers	2
			+-6	Prism Layer Stretching	Prism Layer Stretching	1.5
			+-7	Prism Layer Thickness	Size type	Absolute
			^-1	Absolute Size	Value	15.0 mm
			+-8	Surface Curvature	Enable curvature deviation distance	false
			^-1	Basic Curvature	# Pts/circle	36.0
			+-9	Surface Growth Rate	Surface Growth Rate	1.2
			+-10	Surface Size	Size Method	Min and Max
					Relative/Absolute	Relative to base
			+-1	Relative Minimum Size	Percentage of Base	25.0
					Absolute Size	37.5 mm
			^-2	Relative Maximum Size	Percentage of Base	100.0
					Absolute Size	150.0 mm
			^-11	Template Growth Rate	Boundary Growth Rate	Slow
					Default Growth Rate	Slow
			^-3	Volumetric Controls		
			+-1	Volumetric Control 1	Part Group	[]
					Shapes	[Block 1]
			+-1	Mesh Conditions		
			+-1	Prism Layer Mesher	Customize prism layer mesher	Enabled
			+-2	Surface Remesher	Customize surface remesher	Enabled
			^-3	Trimmer	Customize anisotropic size	Disabled
					Customize isotropic size	Disabled
			^-2	Mesh Values		
			+-1	Custom Size	Size type	Relative to base
			^-1	Relative Size	Percentage of Base	10.0
					Absolute Size	15.0 mm
			+-2	Number of Prism Layers	Number of Prism Layers	2
			+-3	Prism Layer Stretching	Prism Layer Stretching	1.5
			^-4	Prism Layer Thickness	Size type	Absolute
			^-1	Absolute Size	Value	20.0 mm
			+-2	Volumetric Control 2	Part Group	[]
					Shapes	[Block 2]
			+-1	Mesh Conditions		
			+-1	Prism Layer Mesher	Customize prism layer mesher	Enabled
			+-2	Surface Remesher	Customize surface remesher	Enabled
			^-3	Trimmer	Customize anisotropic size	Disabled
					Customize isotropic size	Disabled
			^-2	Mesh Values		
			+-1	Custom Size	Size type	Relative to base
			^-1	Relative Size	Percentage of Base	20.0
					Absolute Size	30.0 mm
			+-2	Number of Prism Layers	Number of Prism Layers	2
			+-3	Prism Layer Stretching	Prism Layer Stretching	1.5
			^-4	Prism Layer Thickness	Size type	Absolute
			^-1	Absolute Size	Value	20.0 mm
			+-3	Volumetric Control 3	Part Group	[]
					Shapes	[Block 3]
			+-1	Mesh Conditions		
			+-1	Prism Layer Mesher	Customize prism layer mesher	Enabled
			+-2	Surface Remesher	Customize surface remesher	Enabled
			^-3	Trimmer	Customize anisotropic size	Disabled
					Customize isotropic size	Disabled
			^-2	Mesh Values		
			+-1	Custom Size	Size type	Relative to base
			^-1	Relative Size	Percentage of Base	50.0
					Absolute Size	75.0 mm
			+-2	Number of Prism Layers	Number of Prism Layers	2

			+3	Prism Layer Stretching	Prism Layer Stretching	1.5
			^-4	Prism Layer Thickness	Size type	Absolute
				^-1 Absolute Size	Value	20.0 mm
			+4	Volumetric Control 4	Part Group	[]
					Shapes	[Block 4]
			+1	Mesh Conditions		
				+1 Prism Layer Mesher	Customize prism layer mesher	Enabled
				+2 Surface Remesher	Customize surface remesher	Enabled
				^-3 Trimmer	Customize anisotropic size	Disabled
					Customize isotropic size	Disabled
			^-2	Mesh Values		
			+1	Custom Size	Size type	Relative to base
				^-1 Relative Size	Percentage of Base	75.0
					Absolute Size	112.5 mm
			+2	Number of Prism Layers	Number of Prism Layers	2
			+3	Prism Layer Stretching	Prism Layer Stretching	1.5
			^-4	Prism Layer Thickness	Size type	Absolute
				^-1 Absolute Size	Value	20.0 mm
			^-5	Volumetric Control 5	Part Group	[]
					Shapes	[Block 5]
			+1	Mesh Conditions		
				+1 Prism Layer Mesher	Customize prism layer mesher	Enabled
				+2 Surface Remesher	Customize surface remesher	Enabled
				^-3 Trimmer	Customize anisotropic size	Disabled
					Customize isotropic size	Disabled
			^-2	Mesh Values		
			+1	Custom Size	Size type	Relative to base
				^-1 Relative Size	Percentage of Base	5.0
					Absolute Size	7.5 mm
			+2	Number of Prism Layers	Number of Prism Layers	2
			+3	Prism Layer Stretching	Prism Layer Stretching	1.5
			^-4	Prism Layer Thickness	Size type	Absolute
				^-1 Absolute Size	Value	20.0 mm
			^-2	Physics 1	Interfaces	[]
					Regions	[Region 1]
			+1	Models		
			+1	All y+ Wall Treatment	Iterative Ustar	false
			+2	Constant Density		
			+3	Gamma ReTheta	Free Stream Edge	FreeStreamEdge
					Convection	2nd-order
					Sigma_f	1.0
					ca1	2.0
					ca2	0.06
					ce1	1.0
					ce2	50.0
					cTheta_t	0.03
					s1	2.0
					Secondary Gradients	On
					Correlation Method	Suluksna-Juntasaro
					Sigma_ReTheta	2.0
					Intermittency Minimum	1.0E-10
					ReThetaT Minimum	20.0
			+4	Gas		
			^-1	Air	Database Material	Air (Air) [Standard/Gases]
			^-1	Material Properties		
			+1	Density	Method	Constant
				^-1 Constant	Value	1.225 kg/m^3
			^-2	Dynamic Viscosity	Method	Constant
				^-1 Constant	Value	1.85508E-5 Pa-s
			+5	Gradients	Verbose	false
					Flat Cells Curvature Criterion	true
					Gradient Method	Hybrid Gauss-LSQ
					Limiter Method	Venkatakrisnan
					Least-Squares Quality Criterion	true
					Cell Skewness Criterion	true
					Chevron-Cell Criterion	true
					Least-Squares Tensor Minimum	0.1
					Eigenvalues Ratio	
					Normalized Flat Cells Curvature Factor	1.0
					Maximum safe (positive) skewness	75.0
					angle (deg.)	
					Minimum unsafe (positive) skewness	88.0
					angle (deg.)	
			+6	Gravity		
			+7	K-Omega Turbulence		
			+8	Reynolds-Averaged Navier-Stokes		
			+9	Segregated Flow	Convection	2nd-order

		Minimum Absolute Pressure	1000.0 Pa
		Secondary Gradients	On
		Delta-V Dissipation	Off
		Flow Boundary Diffusion	true
	+-10	SST (Menter) K-Omega	a1
			Kappa
			BetaStar
			Beta1
			Sigma_k1
			Sigma_w1
			Beta2
			Sigma_k2
			Sigma_w2
			Convection
			Secondary Gradients
			Curvature Correction Option
			Realizability Option
			Compressibility Correction
			Low Re Damping Modification
			Normal Stress Term
			Tke Minimum
			Sdr Minimum
			Zeta_Star
	+-1	Compressibility Parameters	
		-2 Realizability Coefficient	Realizability Coefficient
	+-11	Steady	
	+-12	Three Dimensional	
	-13	Turbulent	
	+2	Reference Values	
		+1 Gravity	Value
			[0.0, -9.81, 0.0] m/s^2
		+2 Reference Altitude	Value
			[0.0, 0.0, 0.0] m
		+3 Minimum Allowable Wall Distance	Value
			1.0E-6 m
		-4 Reference Pressure	Value
			101325.0 Pa
	-3	Initial Conditions	
		+1 Intermittency	Method
			Constant
		-1 Constant	Value
			1.0
		+2 Pressure	Method
			Constant
		-1 Constant	Value
			0.0 Pa
		+3 Turbulence Intensity	Method
			Constant
		-1 Constant	Value
			0.01
		+4 Turbulence Specification	Method
			Intensity + Viscosity Ratio
		+5 Turbulent Velocity Scale	Method
			Constant
		-1 Constant	Value
			1.0 m/s
		+6 Turbulent Viscosity Ratio	Method
			Constant
		-1 Constant	Value
			10.0
		-7 Velocity	Coordinate System
			Laboratory
			Method
			Constant
		-1 Constant	Value
			[0.0, 0.0, 0.0] m/s
	+9	Regions	Regions
			1
		-1 Region 1	Index
			0
			Type
			Fluid Region
			Mesh Continuum
			Mesh 1
			Parts
			[Windtonnel_rect]
			Physics Continuum
			Physics 1
		+1 Boundaries	Boundaries
			5
		+1 Windtonnel_rect.Inlet	Index
			2
			Type
			Velocity Inlet
			Part Surfaces
			[Windtonnel_rect.Inlet]
			Interfaces
		+1 Mesh Conditions	
		+1 Custom Boundary Growth Rate	Custom Boundary Growth Rate
			Disabled
		+2 Custom Surface Curvature	Custom curvature
			Use Continuum Values
		+3 Custom Surface Size	Custom surface size
			Disabled
		+4 Customize Prism Mesh	Customize Prism Mesh
			Use Default Values
		-5 Customize Surface Remeshing	Disable surface remeshing
			Disabled
		+2 Physics Conditions	
		+1 Flow Direction Specification	Method
			Components
		+2 Reference Frame Specification	Option
			Lab Frame
		+3 Turbulence Specification	Method
			Intensity + Viscosity Ratio
		-4 Velocity Specification	Method
			Magnitude + Direction
		-3 Physics Values	
		+1 Flow Direction	Coordinate System
			Laboratory
			Method
			Constant
		-1 Constant	Value
			[1.0, 0.0, 0.0]
		+2 Turbulence Intensity	Method
			Constant
		-1 Constant	Value
			0.01
		+3 Turbulent Viscosity Ratio	Method
			Constant
		-1 Constant	Value
			10.0

			^-4 Velocity Magnitude	Method	Constant	
			^-1 Constant	Value	28.0 m/s	
		+2	Windtonnel_rect.Outlet	Index	3	
				Type	Pressure Outlet	
				Part Surfaces	[Windtonnel_rect.Outlet]	
				Interfaces		
			+1 Mesh Conditions			
			+1 Custom Boundary Growth Rate	Custom Boundary Growth Rate	Disabled	
			+2 Custom Surface Curvature	Custom curvature	Use Continuum Values	
			+3 Custom Surface Size	Custom surface size	Disabled	
			+4 Customize Prism Mesh	Customize Prism Mesh	Use Default Values	
			^-5 Customize Surface Remeshing	Disable surface remeshing	Disabled	
		+2	Physics Conditions			
			+1 Backflow Direction Specification	Method	Boundary-Normal	
			+2 Pressure Specification	Pressure specification	Environmental	
			+3 Target Mass Flow Option	Target Mass Flow Option	Disabled	
			^-4 Turbulence Specification	Method	Intensity + Viscosity Ratio	
			^-3 Physics Values			
			+1 Pressure	Method	Constant	
				^-1 Constant	Value	0.0 Pa
			+2 Turbulence Intensity	Method	Constant	
				^-1 Constant	Value	0.01
			^-3 Turbulent Viscosity Ratio	Method	Constant	
				^-1 Constant	Value	10.0
		+3	Windtonnel_rect.SP	Index	5	
				Type	Symmetry Plane	
				Part Surfaces	[Windtonnel_rect.SP]	
				Interfaces		
			^-1 Mesh Conditions			
			+1 Custom Boundary Growth Rate	Custom Boundary Growth Rate	Disabled	
			+2 Custom Surface Curvature	Custom curvature	Use Continuum Values	
			+3 Custom Surface Size	Custom surface size	Disabled	
			+4 Customize Prism Mesh	Customize Prism Mesh	Use Default Values	
			^-5 Customize Surface Remeshing	Disable surface remeshing	Disabled	
		+4	Windtonnel_rect.VeloX1	Index	1	
				Type	Wall	
				Part Surfaces	[Windtonnel_rect.VeloX1]	
				Interfaces		
			+1 Mesh Conditions			
			+1 Custom Boundary Growth Rate	Custom Boundary Growth Rate	Disabled	
			+2 Custom Surface Curvature	Custom curvature	Use Continuum Values	
			+3 Custom Surface Size	Custom surface size	Enabled	
			+4 Customize Prism Mesh	Customize Prism Mesh	Specify Custom Values	
			^-5 Customize Surface Remeshing	Disable surface remeshing	Disabled	
		+2	Mesh Values			
			+1 Number of Prism Layers	Number of Prism Layers	15	
			+2 Prism Layer Stretching	Prism Layer Stretching	1.5	
			+3 Prism Layer Thickness	Size type	Absolute	
				^-1 Absolute Size	Value	10.0 mm
			^-4 Surface Size	Size Method	Min and Max	
				Relative/Absolute	Relative to base	
			+1 Relative Minimum Size	Percentage of Base	0.5	
				Absolute Size	0.75 mm	
			^-2 Relative Maximum Size	Percentage of Base	15.0	
				Absolute Size	22.5 mm	
		+3	Physics Conditions			
			+1 Shear Stress Specification	Method	No-Slip	
			+2 Tangential Velocity Specification	Method	None	
				Reference Frame	Relative To Mesh	
			^-3 Wall Surface Specification	Method	Smooth	
			^-4 Physics Values			
			^-1 Blended Wall Function	Kappa	0.42	
				E	9.0	
			^-5 Windtonnel_rect.Walls	Index	4	
				Type	Wall	
				Part Surfaces	[Windtonnel_rect.Walls]	
				Interfaces		
			+1 Mesh Conditions			
			+1 Custom Boundary Growth Rate	Custom Boundary Growth Rate	Disabled	
			+2 Custom Surface Curvature	Custom curvature	Use Continuum Values	
			+3 Custom Surface Size	Custom surface size	Disabled	
			+4 Customize Prism Mesh	Customize Prism Mesh	Specify Custom Values	
			^-5 Customize Surface Remeshing	Disable surface remeshing	Disabled	
		+2	Mesh Values			
			+1 Number of Prism Layers	Number of Prism Layers	2	
			+2 Prism Layer Stretching	Prism Layer Stretching	1.5	
			^-3 Prism Layer Thickness	Size type	Absolute	

			^-1 Absolute Size	Value	17.0 mm
			+3 Physics Conditions		
			+1 Shear Stress Specification	Method	No-Slip
			+2 Tangential Velocity Specification	Method	None
				Reference Frame	Relative To Mesh
			^-3 Wall Surface Specification	Method	Smooth
			^-4 Physics Values		
			^-1 Blended Wall Function	Kappa	0.42
				E	9.0
			+2 Feature Curves	Feature Curves	1
			^-1 Default Feature Curve	Part Curves	[Windtonnel_rect.Edges]
			^-1 Mesh Conditions		
			^-1 Custom Surface Size	Custom surface size	Disabled
			+3 Mesh Conditions		
			^-1 Customize Prism Mesh	Customize Prism Mesh	Use Default Values
			+4 Mesh Values		
			^-1 Trimmer Wake Refinement		
			+5 Physics Conditions		
			+1 Initial Condition Option	Option	Use Continuum Values
			+2 Mass Source Option	Mass Source Option	Disabled
			+3 Momentum Source Option	Momentum Source Option	None
			^-4 Turbulence Source Option	Turbulence Source Option	None
			^-6 Physics Values		
			+1 Axis	Direction	[0.0, 0.0, 1.0]
				Coordinate System	Laboratory
				Origin	[0.0, 0.0, 0.0] m
			^-2 Motion Specification	Reference Frame	Lab Reference Frame
				Motion	Stationary
			+10 Derived Parts	Derived Parts	0
			+11 Solvers		
			+1 Wall Distance	Verbosity	0
				Parallel memory optimization scaling factor	1.0
				Solver Frozen	false
			+2 Segregated Flow	Reconstruction Zeroed	false
				Reconstruction Frozen	false
				Temporary Storage Retained	false
				Solver Frozen	false
			+1 Velocity	Under-Relaxation Factor	0.7
			+1 Under-Relaxation Factor Ramp	Ramp Method	No Ramp
			^-2 AMG Linear Solver	Verbosity	None
				Max Cycles	30
				Epsilon	0.0
				Group Size	4
				Parallel Migration Limit	25
				Enable direct-solver	false
				Maximum direct-solver equations	32
				Convergence Tolerance	0.1
				Cycle Type	Flex Cycle
				Group Size Control	Auto
				Relaxation Scheme	Gauss-Seidel
				Scaling	Disabled
				Subdomain coarsening enabled	true
				Acceleration method	None
			^-1 Flex Cycle	Sweeps	1
				Restriction Tolerance	0.9
				Prolongation Tolerance	0.5
			^-2 Pressure	Under-Relaxation Factor	0.3
				Pressure Reference Location	Automatic Selection
			+1 Under-Relaxation Factor Ramp	Ramp Method	No Ramp
			^-2 AMG Linear Solver	Verbosity	None
				Max Cycles	30
				Epsilon	0.0
				Group Size	4
				Parallel Migration Limit	25
				Enable direct-solver	false
				Maximum direct-solver equations	32
				Convergence Tolerance	0.1
				Cycle Type	V Cycle
				Group Size Control	Auto
				Relaxation Scheme	Gauss-Seidel
				Scaling	Auto
				Subdomain coarsening enabled	true
				Acceleration method	Conjugate Gradient
			^-1 V Cycle	Pre-Sweeps	1
				Max Levels	50
				Post-Sweeps	1

+-3	K-Omega Turbulence	Under-Relaxation Factor	0.8
		Reconstruction Zeroed	false
		Reconstruction Frozen	false
		Temporary Storage Retained	false
		Solver Frozen	false
	+-1 Under-Relaxation Factor Ramp	Ramp Method	No Ramp
	^-2 AMG Linear Solver	Verbosity	None
		Max Cycles	30
		Epsilon	0.0
		Group Size	4
		Parallel Migration Limit	25
		Enable direct-solver	false
		Maximum direct-solver equations	32
		Convergence Tolerance	0.1
		Cycle Type	Flex Cycle
		Group Size Control	Auto
		Relaxation Scheme	Gauss-Seidel
		Scaling	Disabled
		Subdomain coarsening enabled	true
		Acceleration method	None
	^-1 Flex Cycle	Sweeps	1
		Restriction Tolerance	0.9
		Prolongation Tolerance	0.5
	+-4 K-Omega Turbulent Viscosity	Maximum Ratio	100000.0
		Under-Relaxation Factor	1.0
		Solver Frozen	false
	^-5 GammaReTheta Transition	Freestream Update Frequency	1
		Freestream Update Policy	Iteration
		Under-Relaxation Factor	0.8
		Reconstruction Zeroed	false
		Reconstruction Frozen	false
		Temporary Storage Retained	false
		Solver Frozen	false
	+-1 Under-Relaxation Factor Ramp	Ramp Method	No Ramp
	^-2 AMG Linear Solver	Verbosity	None
		Max Cycles	30
		Epsilon	0.0
		Group Size	4
		Parallel Migration Limit	25
		Enable direct-solver	false
		Maximum direct-solver equations	32
		Convergence Tolerance	0.1
		Cycle Type	Flex Cycle
		Group Size Control	Auto
		Relaxation Scheme	Gauss-Seidel
		Scaling	Disabled
		Subdomain coarsening enabled	true
		Acceleration method	None
	^-1 Flex Cycle	Sweeps	1
		Restriction Tolerance	0.9
		Prolongation Tolerance	0.5
+-12	Stopping Criteria		
	+-1 Maximum Steps	Maximum Steps	2000
		Criterion Satisfied	true
		Logical Rule	Or
		Enabled	true
	^-2 Stop File	Stop Inner Iterations	true
		Path	ABORT
		Criterion Satisfied	false
		Logical Rule	Or
		Enabled	true
+-13	Solution Histories		
+-14	Solution Views		
	^-1 Current Solution	Time Step	0
		Solution Time	0.0
		Iteration	2000
+-15	Reports	Reports	1
	^-1 Force 1	Direction	[1.0, 0.0, 0.0]
		Reference Pressure	0.0 Pa
		Number of Bands	0
		Force Option	Pressure + Shear
		Coordinate System	Laboratory
		Representation	Geometry
		Parts	[Region 1: Windtonnel_rect.VeloX1]
		Smooth Values	false
		Units	N

+-16 Monitors		Monitors To Print	[Continuity, X-momentum, Y-momentum, Z-momentum, Tke, Sdr, ReTheta_t, Intermittency, Force 1 Monitor]
		Output Direction	Horizontal
		Heading Print Frequency	10
		Monitors	10
	+-1 Force 1 Monitor	Report	Force 1
		Trigger	Iteration
		Normalization Option	Off
		Maximum Plot Samples	5000
		Frequency	1
	+-2 Iteration	Trigger	Iteration
		Frequency	1
		Normalization Option	Off
		Maximum Plot Samples	5000
+-17 Representations			
	+-1 Geometry		
	+-2 Initial Surface	Faces	91252
		Edges	861
	^-1 Regions		
	^-1 Region 1	Faces	91252
		Edges	861
	+-1 Boundaries		
		+-1 Windtonnel_rect.Inlet	Faces 2
		+-2 Windtonnel_rect.Outlet	Faces 2
		+-3 Windtonnel_rect.SP	Faces 734
		+-4 Windtonnel_rect.VeloX1	Faces 90391
		^-5 Windtonnel_rect.Walls	Faces 123
	^-2 Feature Curves		
	^-1 Default Feature Curve	Edges	861
	+-3 Remeshed Surface	Faces	261106
		Edges	3105
	^-1 Regions		
	^-1 Region 1	Faces	261106
		Edges	3105
	+-1 Boundaries		
		+-1 Windtonnel_rect.Inlet	Faces 726
		+-2 Windtonnel_rect.Outlet	Faces 718
		+-3 Windtonnel_rect.SP	Faces 69297
		+-4 Windtonnel_rect.VeloX1	Faces 99972
		^-5 Windtonnel_rect.Walls	Faces 90393
	^-2 Feature Curves		
	^-1 Default Feature Curve	Edges	3105
	^-4 Volume Mesh	Cells	1874908
		Interior Faces	5552599
		Vertices	1969310
	+-1 Finite Volume Regions		
	^-1 Region 1	Cells	1874908
		Interior Faces	5552599
		Vertices	1969310
	^-1 Finite Volume Boundaries		
		+-1 Windtonnel_rect.Inlet	Faces 615
		+-2 Windtonnel_rect.Outlet	Faces 618
		+-3 Windtonnel_rect.SP	Faces 57925
		+-4 Windtonnel_rect.VeloX1	Faces 66113
		^-5 Windtonnel_rect.Walls	Faces 43674
	^-2 Cell Sets		
+-18 Coordinate Systems			
	^-1 Laboratory		
	^-1 Local Coordinate Systems		
+-19 Units		Preferred System	Système Internationale
+-20 Tables		Tables	0
+-21 Field Functions			
	+-1 FreeStreamEdge	Type	Scalar
		Definition	(\$WallDistance>0.01) ? 1:0
		Ignore Boundary Values	false
		Function Name	FreeStreamEdge
		Assembly code	(if (> \$WallDistance) 0.01) 1 0)
		Inverse Distance Weight	false
	+-2 UserFreeStreamEdge	Type	Scalar
		Definition	(\$WallDistance>0.08) ? 1:0
		Ignore Boundary Values	false
		Function Name	UserFieldFunction_1
		Assembly code	(if (> \$WallDistance) 0.08) 1 0)
		Inverse Distance Weight	false
+-22 Volume Shapes			

+-1	Block 1	Corner1	[-0.2, -0.15, -1.1192331612619455E-6] m,m,m
		Corner2	[3.0, 1.0, 0.75] m,m,m
		Coordinate System	Laboratory
+-2	Block 2	Corner1	[-0.4, -0.15, -1.1192331612619455E-6] m,m,m
		Corner2	[7.0, 1.2, 0.75] m,m,m
		Coordinate System	Laboratory
+-3	Block 3	Corner1	[-0.6, -0.15, -1.1192331612619455E-6] m,m,m
		Corner2	[11.0, 1.4, 0.75] m,m,m
		Coordinate System	Laboratory
+-4	Block 4	Corner1	[-0.8, -0.15, -1.1192331612619455E-6] m,m,m
		Corner2	[16.0, 1.6, 0.75] m,m,m
		Coordinate System	Laboratory
+-5	Block 5	Corner1	[-0.029999999329447746, -0.15000000596046448, -5.59618911211146E-7] m,m,m
		Corner2	[2.75, 0.019999999552965164, 0.25] m,m,m
		Coordinate System	Laboratory
+-23	Data Set Functions	Data Directory	function_data
+-24	User Code		
+-25	Layouts		
^-1	default		
+-26	Data Mappers		
+-27	Motions		
^-1	Stationary		
+-28	Reference Frames		
^-1	Lab Reference Frame		

Solution

Accumulated CPU Time over all processes (s) 61656.233999999764

Elapsed Time (s) 61656.24899999971

Iterations 2000

Annexure C: Detail Design Analysis Summary Report

The following summary report lists the STAR-CCM+ settings used during the 3-D evaluation of the solar powered vehicle in question. It is based on the settings of the previous section.

Summary Report: Model 1

Session Summary

Date 07 Sep 2015 7:14:32 PM
Simulation C:\Users\Armand\Documents\M.Ing Meganies\Meesters\Detail Design
REMEDY\1\Model 1.sim
File size 2.7e+03 MB
Number of Partitions 1
Number of Restored Partitions 32

Software Summary

Version BuildArch: win64
BuildEnv: intel12.1
ReleaseDate: Thu Feb 12 02:40:23 UTC 2015
ReleaseNumber: 10.02.010

Hardware Summary

Hosts Controller: Armand-PC.fritz.box
Number of Workers: 0

Simulation Properties

1 Model 1			
+-1 Tags			
+-2 Contacts			
+-3 3D-CAD Models			
+-4 Parts			
	+-1 Rev1	Metadata	{}
		Index	1
		Descriptions	Root
		Face Count	265634
+-1 Surfaces			
	+-1 Floor	Metadata	{}
	+-2 Inlet	Metadata	{}
	+-3 Outlet	Metadata	{}
	+-4 rev1	Metadata	{}
	+-5 Walls	Metadata	{}
+-2 Curves			
+-5 Operations			
+-6 Descriptions			
	+-1 Root	Number of Children	2
		Described Parts	Rev1
	+-2 Latest	Described Parts	Rev1
		Faces	265634
		Vertices	132813
+-7 Continua			
	+-1 Mesh 1	Continua	2
		Regions	[Region 2]
		Interfaces	[]
		OOB Translation	false
		Verbose Output	false
		Per-Region Meshing	false
		Enable Parallel Meshing	false
		Interpolation Option	Nearest neighbor
+-1 Models			
	+-1 Prism Layer Mesher	Stretching Function	Geometric Progression
		Stretching Mode	Stretch Factor
		Gap Fill Percentage	25.0
		Minimum Thickness Percentage	10.0
		Layer Reduction Percentage	50.0
		Boundary March Angle	50.0

				Concave Angle Limit	0.0
				Convex Angle Limit	360.0
				Near Core Layer Aspect Ratio	0.0
				Generate Standard Cells Only	false
				Improve Subsurface Quality	true
			+-2 Surface Remesher	Do Curvature Refinement	true
				Do Proximity Refinement	false
				Do Compatibility Refinement	false
				Retain Geometric Features	false
				Create Aligned Meshes	false
				Minimum Face Quality	0.0
				Enable Automatic Surface Repair	false
			^-3 Trimmer	Coordinate System	Laboratory
				Do Mesh Alignment	false
				Template Mesh Type	Hexahedra
				Template Mesh Growth Type	Simple
				Run Optimizer	true
			+-2 Reference Values		
			+-1 Base Size	Value	250.0 mm
			+-2 CAD Projection	Project to CAD	false
			+-3 Maximum Cell Size	Size Type	Relative to base
			^-1 Relative Size	Percentage of Base	100.0
				Absolute Size	250.0 mm
			+-4 Maximum Core/Prism	Limit Cell Size by Prism Layer	false
Transition Ratio				Thickness	
			+-5 Number of Prism	Number of Prism Layers	2
Layers					
			+-6 Prism Layer Stretching	Prism Layer Stretching	1.5
			+-7 Prism Layer Thickness	Size Type	Absolute
			^-1 Absolute Size	Value	15.0 mm
			+-8 Surface Curvature	Enable Curvature Deviation Distance	false
			^-1 Basic Curvature	# Pts/circle	36.0
			+-9 Surface Growth Rate	Surface Growth Rate	1.2
			+-10 Surface Size	Relative/Absolute	Relative to base
				Size Method	Min and Max
			+-1 Relative Minimum	Percentage of Base	25.0
Size					
				Absolute Size	62.5 mm
			^-2 Relative Maximum	Percentage of Base	100.0
Size					
				Absolute Size	250.0 mm
			^-11 Template Growth	Default Growth Rate	Slow
Rate					
				Boundary Growth Rate	Slow
			^-3 Volumetric Controls		
			+-1 Volumetric Control 1		
			+-1 Mesh Conditions		
			+-1 Prism Layer	Customize Prism Layer Mesher	Enabled
Mesher					
			+-2 Surface	Customize Surface Remesher	Enabled
Remesher					
			^-3 Trimmer	Customize Isotropic Size	Disabled
				Customize Anisotropic Size	Disabled
			^-2 Mesh Values		
			+-1 Custom Size	Size Type	Relative to base
			^-1 Relative Size	Percentage of Base	10.0
				Absolute Size	25.0 mm
			+-2 Number of Prism	Number of Prism Layers	2
Layers					
			+-3 Prism Layer	Prism Layer Stretching	1.5
Stretching					

^-4 Prism Layer	Size Type	Absolute
Thickness ^-1 Absolute Size	Value	20.0 mm
+-2 Volumetric Control 2		
+-1 Mesh Conditions		
+-1 Prism Layer	Customize Prism Layer Mesher	Disabled
Mesher +-2 Surface	Customize Surface Remesher	Enabled
Remesher ^-3 Trimmer	Customize Isotropic Size	Disabled
	Customize Anisotropic Size	Disabled
^-2 Mesh Values		
^-1 Custom Size	Size Type	Relative to base
^-1 Relative Size	Percentage of Base	20.0
	Absolute Size	50.0 mm
+-3 Volumetric Control 3		
+-1 Mesh Conditions		
+-1 Prism Layer	Customize Prism Layer Mesher	Disabled
Mesher +-2 Surface	Customize Surface Remesher	Enabled
Remesher ^-3 Trimmer	Customize Isotropic Size	Disabled
	Customize Anisotropic Size	Disabled
^-2 Mesh Values		
^-1 Custom Size	Size Type	Relative to base
^-1 Relative Size	Percentage of Base	30.0
	Absolute Size	75.0 mm
+-4 Volumetric Control 4		
+-1 Mesh Conditions		
+-1 Prism Layer	Customize Prism Layer Mesher	Disabled
Mesher +-2 Surface	Customize Surface Remesher	Enabled
Remesher ^-3 Trimmer	Customize Isotropic Size	Disabled
	Customize Anisotropic Size	Disabled
^-2 Mesh Values		
^-1 Custom Size	Size Type	Relative to base
^-1 Relative Size	Percentage of Base	40.0
	Absolute Size	100.0 mm
^-5 Volumetric Control 5		
+-1 Mesh Conditions		
+-1 Prism Layer	Customize Prism Layer Mesher	Disabled
Mesher +-2 Surface	Customize Surface Remesher	Enabled
Remesher ^-3 Trimmer	Customize Isotropic Size	Disabled
	Customize Anisotropic Size	Disabled
^-2 Mesh Values		
^-1 Custom Size	Size Type	Relative to base
^-1 Relative Size	Percentage of Base	1.0
	Absolute Size	2.5 mm
^-2 Physics 1	Regions	[Region 2]
	Interfaces	[]
+-1 Models		
+-1 All y+ Wall Treatment	Iterative Ustar	false
+-2 Cell Quality		
Remediation +-3 Constant Density		
+-4 Gamma ReTheta	Free Stream Edge	FreeStreamEdge
	Correlation Method	Suluksna-Juntasaro
	Sigma_f	1.0
	Sigma_ReTheta	2.0

			ca1	2.0
			ca2	0.06
			ce1	1.0
			ce2	50.0
			cTheta_t	0.03
			s1	2.0
			Intermittency Minimum	1.0E-10
			ReThetaT Minimum	20.0
			Secondary Gradients	On
			Convection	2nd-order
		+-5 Gas		
		`-1 Air	Database Material	Air (Air) [Standard/Gases]
		`-1 Material		
Properties				
		+-1 Density	Method	Constant
		`-1 Constant	Value	1.225 kg/m^3
		`-2 Dynamic	Method	Constant
Viscosity				
		`-1 Constant	Value	1.85508E-5 Pa-s
		+-6 Gradients	Gradient Method	Hybrid Gauss-LSQ
			Limiter Method	Venkatakrishnan
			Custom Accuracy Level Selector	2.0
			Verbose	false
			Least-Squares Quality Criterion	true
			Flat Cells Curvature Criterion	true
			Cell Skewness Criterion	true
			Chevron-Cell Criterion	true
			Least-Squares Tensor Minimum	0.1
			Eigenvalues Ratio	
			Normalized Flat Cells Curvature Factor	1.0
			Maximum Safe (Positive) Skewness	75.0
			Angle (deg)	
			Minimum Unsafe (Positive) Skewness	88.0
			Angle (deg)	
			Use TVB Gradient Limiting	false
			Acceptable Field Variation (Factor)	0.05
		+-7 Gravity		
		+-8 K-Omega Turbulence		
		+-9 Reynolds-Averaged		
Navier-Stokes				
		+-10 Segregated Flow	Minimum Absolute Pressure	1000.0 Pa
			Flow Boundary Diffusion	true
			Unsteady Flux Dissipation Corrections	false
			Limit Acoustic-CFL Option	Per-Model
			Secondary Gradients	On
			Convection	2nd-order
			Delta-V Dissipation	Off
		+-11 SST (Menter)	Curvature Correction Option	Off
K-Omega				
			Realizability Option	Durbin Scale Limiter
			Compressibility Correction	true
			Low Re Damping Modification	false
			Convection	2nd-order
			Normal Stress Term	false
			Tke Minimum	1.0E-10
			Sdr Minimum	1.0E-10
			Secondary Gradients	On
			Kappa	0.41
			BetaStar	0.09
			Beta1	0.075
			Sigma_k1	0.85
			Sigma_w1	0.5

				Beta2	0.0828
				Sigma_k2	1.0
				Sigma_w2	0.856
				a1	0.31
			+--1 Compressibility	Zeta_Star	1.5
Parameters					
			`-2 Realizability	Realizability Coefficient	0.6000000238418579
Coefficient					
			+--12 Steady		
			+--13 Three Dimensional		
			`-14 Turbulent		
			+--2 Reference Values		
			+--1 Gravity	Value	[0.0, -9.81, 0.0] m/s^2
			+--2 Reference Altitude	Value	[0.0, 0.0, 0.0] m
			+--3 Minimum Allowable	Value	1.0E-6 m
Wall Distance					
			`-4 Reference Pressure	Value	101325.0 Pa
			`-3 Initial Conditions		
			+--1 Intermittency	Method	Constant
			`-1 Constant	Value	1.0
			+--2 Pressure	Method	Constant
			`-1 Constant	Value	0.0 Pa
			+--3 Turbulence Intensity	Method	Constant
			`-1 Constant	Value	0.01
			+--4 Turbulence	Method	Intensity + Viscosity Ratio
Specification					
			+--5 Turbulent Velocity	Method	Constant
Scale					
			`-1 Constant	Value	1.0 m/s
			+--6 Turbulent Viscosity	Method	Constant
Ratio					
			`-1 Constant	Value	10.0
			`-7 Velocity	Method	Constant
				Coordinate System	Laboratory
			`-1 Constant	Value	[28.0, 0.0, 0.0] m/s
			+--8 Regions	Regions	1
			`-1 Region 2	Index	1
				Mesh Continuum	Mesh 1
				Physics Continuum	Physics 1
				Type	Fluid Region
				Allow Per-Part Values	false
			+--1 Boundaries	Boundaries	8
			+--1 Floor	Index	30
				Interfaces	
				Type	Wall
				Allow Per-Surface Values	false
			+--1 Mesh Conditions		
			+--1 Custom	Custom Boundary Growth Rate	Disabled
Boundary Growth Rate					
			+--2 Custom Surface	Custom Curvature	Use Continuum Values
Curvature					
			+--3 Custom Surface	Custom Surface Size	Disabled
Size					
			+--4 Customize Prism	Customize Prism Mesh	Use Default Values
Mesh					
			`-5 Customize	Disable Surface Remeshing	Disabled
Surface Remeshing					
			+--2 Physics Conditions		
			+--1 Reference Frame	Option	Region Reference Frame
Specification					
			+--2 Shear Stress	Method	No-Slip
Specification					

+-3 Tangential	Method	Vector
Velocity Specification		
^-4 Wall Surface	Method	Smooth
Specification		
^-3 Physics Values		
+-1 Blended Wall	E	9.0
Function		
	Kappa	0.42
^-2 Relative Velocity	Method	Constant
	Coordinate System	Laboratory
^-1 Constant	Value	[28.0, 0.0, 0.0] m/s
+-2 Inlet	Index	33
	Interfaces	
	Type	Velocity Inlet
	Allow Per-Surface Values	false
+-1 Mesh Conditions		
+-1 Custom	Custom Boundary Growth Rate	Disabled
Boundary Growth Rate		
+-2 Custom Surface	Custom Curvature	Use Continuum Values
Curvature		
+-3 Custom Surface	Custom Surface Size	Disabled
Size		
+-4 Customize Prism	Customize Prism Mesh	Use Default Values
Mesh		
^-5 Customize	Disable Surface Remeshing	Disabled
Surface Remeshing		
+-2 Physics Conditions		
+-1 Flow Direction	Method	Components
Specification		
+-2 Reference Frame	Option	Lab Frame
Specification		
+-3 Turbulence	Method	Intensity + Viscosity Ratio
Specification		
^-4 Velocity	Method	Magnitude + Direction
Specification		
^-3 Physics Values		
+-1 Flow Direction	Method	Constant
	Coordinate System	Laboratory
^-1 Constant	Value	[1.0, 0.0, 0.0]
+-2 Turbulence	Method	Constant
Intensity		
^-1 Constant	Value	0.01
+-3 Turbulent	Method	Constant
Viscosity Ratio		
^-1 Constant	Value	10.0
^-4 Velocity	Method	Constant
Magnitude		
^-1 Constant	Value	28.0 m/s
+-3 Model1 Body	Index	42
	Interfaces	
	Type	Wall
	Allow Per-Surface Values	false
+-1 Mesh Conditions		
+-1 Custom	Custom Boundary Growth Rate	Disabled
Boundary Growth Rate		
+-2 Custom Surface	Custom Curvature	Use Continuum Values
Curvature		
+-3 Custom Surface	Custom Surface Size	Enabled
Size		
+-4 Customize Prism	Customize Prism Mesh	Specify Custom Values
Mesh		
^-5 Customize	Disable Surface Remeshing	Disabled
Surface Remeshing		

				+-2	Mesh Values		
				+-1	Number of Prism	Number of Prism Layers	15
Layers							
				+-2	Prism Layer	Prism Layer Stretching	1.5
Stretching							
				+-3	Prism Layer	Size Type	Absolute
Thickness							
				`-1	Absolute Size	Value	10.0 mm
				`-4	Surface Size	Relative/Absolute	Relative to base
						Size Method	Min and Max
				+-1	Relative	Percentage of Base	1.0
Minimum Size							
						Absolute Size	2.5 mm
				`-2	Relative	Percentage of Base	10.0
Maximum Size							
						Absolute Size	25.0 mm
				+-3	Physics Conditions		
				+-1	Reference Frame	Option	Region Reference Frame
Specification							
				+-2	Shear Stress	Method	No-Slip
Specification							
				+-3	Tangential	Method	Fixed
Velocity Specification							
				`-4	Wall Surface	Method	Smooth
Specification							
				`-4	Physics Values		
				`-1	Blended Wall	E	9.0
Function							
						Kappa	0.42
		+-4			Model1 Canopy	Index	41
						Interfaces	
						Type	Wall
						Allow Per-Surface Values	false
				+-1	Mesh Conditions		
				+-1	Custom	Custom Boundary Growth Rate	Disabled
Boundary Growth Rate							
				+-2	Custom Surface	Custom Curvature	Use Continuum Values
Curvature							
				+-3	Custom Surface	Custom Surface Size	Enabled
Size							
				+-4	Customize Prism	Customize Prism Mesh	Specify Custom Values
Mesh							
				`-5	Customize	Disable Surface Remeshing	Disabled
Surface Remeshing							
				+-2	Mesh Values		
				+-1	Number of Prism	Number of Prism Layers	15
Layers							
				+-2	Prism Layer	Prism Layer Stretching	1.5
Stretching							
				+-3	Prism Layer	Size Type	Absolute
Thickness							
				`-1	Absolute Size	Value	10.0 mm
				`-4	Surface Size	Relative/Absolute	Relative to base
						Size Method	Min and Max
				+-1	Relative	Percentage of Base	0.1
Minimum Size							
						Absolute Size	0.25 mm
				`-2	Relative	Percentage of Base	1.0
Maximum Size							
						Absolute Size	2.5 mm
				+-3	Physics Conditions		
				+-1	Reference Frame	Option	Region Reference Frame
Specification							

+-2 Shear Stress	Method	No-Slip
Specification		
+-3 Tangential	Method	Fixed
Velocity Specification		
^-4 Wall Surface	Method	Smooth
Specification		
^-4 Physics Values		
^-1 Blended Wall	E	9.0
Function		
+-5 Model1 Leading	Kappa	0.42
Edge	Index	35
	Interfaces	
	Type	Wall
	Allow Per-Surface Values	false
+-1 Mesh Conditions		
+-1 Custom	Custom Boundary Growth Rate	Disabled
Boundary Growth Rate		
+-2 Custom Surface	Custom Curvature	Use Continuum Values
Curvature		
+-3 Custom Surface	Custom Surface Size	Enabled
Size		
+-4 Customize Prism	Customize Prism Mesh	Specify Custom Values
Mesh		
^-5 Customize	Disable Surface Remeshing	Disabled
Surface Remeshing		
+-2 Mesh Values		
+-1 Number of Prism	Number of Prism Layers	15
Layers		
+-2 Prism Layer	Prism Layer Stretching	1.5
Stretching		
+-3 Prism Layer	Size Type	Absolute
Thickness		
^-1 Absolute Size	Value	10.0 mm
^-4 Surface Size	Relative/Absolute	Relative to base
	Size Method	Min and Max
+-1 Relative	Percentage of Base	0.1
Minimum Size		
	Absolute Size	0.25 mm
^-2 Relative	Percentage of Base	1.0
Maximum Size		
	Absolute Size	2.5 mm
+-3 Physics Conditions		
+-1 Reference Frame	Option	Region Reference Frame
Specification		
+-2 Shear Stress	Method	No-Slip
Specification		
+-3 Tangential	Method	Fixed
Velocity Specification		
^-4 Wall Surface	Method	Smooth
Specification		
^-4 Physics Values		
^-1 Blended Wall	E	9.0
Function		
+-6 Model1 Trailing Edge	Kappa	0.42
	Index	36
	Interfaces	
	Type	Wall
	Allow Per-Surface Values	false
+-1 Mesh Conditions		
+-1 Custom	Custom Boundary Growth Rate	Disabled
Boundary Growth Rate		

+-2 Custom Surface Curvature	Custom Curvature	Use Continuum Values
+-3 Custom Surface Size	Custom Surface Size	Enabled
+-4 Customize Prism Mesh	Customize Prism Mesh	Specify Custom Values
^-5 Customize Surface Remeshing	Disable Surface Remeshing	Disabled
+-2 Mesh Values		
+-1 Number of Prism Layers	Number of Prism Layers	15
+-2 Prism Layer Stretching	Prism Layer Stretching	1.5
+-3 Prism Layer Thickness	Size Type	Absolute
^-1 Absolute Size	Value	10.0 mm
^-4 Surface Size	Relative/Absolute	Relative to base
	Size Method	Min and Max
+-1 Relative Minimum Size	Percentage of Base	0.1
	Absolute Size	0.25 mm
^-2 Relative Maximum Size	Percentage of Base	1.0
	Absolute Size	2.5 mm
+-3 Physics Conditions		
+-1 Reference Frame Specification	Option	Region Reference Frame
+-2 Shear Stress Specification	Method	No-Slip
+-3 Tangential Velocity Specification	Method	Fixed
^-4 Wall Surface Specification	Method	Smooth
^-4 Physics Values		
^-1 Blended Wall Function	E	9.0
	Kappa	0.42
+-7 Outlet	Index	32
	Interfaces	
	Type	Pressure Outlet
	Allow Per-Surface Values	false
+-1 Mesh Conditions		
+-1 Custom Boundary Growth Rate	Custom Boundary Growth Rate	Disabled
+-2 Custom Surface Curvature	Custom Curvature	Use Continuum Values
+-3 Custom Surface Size	Custom Surface Size	Disabled
+-4 Customize Prism Mesh	Customize Prism Mesh	Use Default Values
^-5 Customize Surface Remeshing	Disable Surface Remeshing	Disabled
+-2 Physics Conditions		
+-1 Backflow Direction Specification	Method	Boundary-Normal
+-2 Pressure Jump Option	Option	None
+-3 Pressure Specification	Pressure Specification	Environmental
+-4 Reference Frame Specification	Option	Lab Frame
+-5 Target Mass Flow Option	Target Mass Flow Option	Disabled

^-6 Turbulence	Method	Intensity + Viscosity Ratio
Specification		
^-3 Physics Values		
+-1 Pressure	Method	Constant
^-1 Constant	Value	0.0 Pa
+-2 Turbulence	Method	Constant
Intensity		
^-1 Constant	Value	0.01
^-3 Turbulent	Method	Constant
Viscosity Ratio		
^-1 Constant	Value	10.0
^-8 Walls	Index	31
	Interfaces	
	Type	Symmetry Plane
	Allow Per-Surface Values	false
^-1 Mesh Conditions		
+-1 Custom	Custom Boundary Growth Rate	Disabled
Boundary Growth Rate		
+-2 Custom Surface	Custom Curvature	Use Continuum Values
Curvature		
+-3 Custom Surface	Custom Surface Size	Disabled
Size		
+-4 Customize Prism	Customize Prism Mesh	Use Default Values
Mesh		
^-5 Customize	Disable Surface Remeshing	Disabled
Surface Remeshing		
+-2 Feature Curves	Feature Curves	2
+-1 Feature Curve 1		
^-1 Mesh Conditions		
^-1 Custom Surface	Custom Surface Size	Disabled
Size		
^-2 Feature Curve 2		
^-1 Mesh Conditions		
^-1 Custom Surface	Custom Surface Size	Disabled
Size		
+-3 Mesh Conditions		
^-1 Customize Prism	Customize Prism Mesh	Use Default Values
Mesh		
+-4 Mesh Values		
^-1 Trimmer Wake		
Refinement		
+-5 Physics Conditions		
+-1 Initial Condition Option	Option	Use Continuum Values
+-2 Mass Source Option	Mass Source Option	Disabled
+-3 Momentum Source	Momentum Source Option	None
Option		
^-4 Turbulence Source	Turbulence Source Option	None
Option		
^-6 Physics Values		
+-1 Axis	Coordinate System	Laboratory
	Origin	[0.0, 0.0, 0.0] m
	Direction	[0.0, 0.0, 1.0]
^-2 Motion Specification	Motion	Stationary
	Reference Frame	Lab Reference Frame
+ -9 Injectors	Tree Threshold	10
+ -10 Derived Parts	Derived Parts	2
+-1 plane section	Origin	[7.000000000000001, 3.105005340576173, 0.2045] m,m,m
	Coordinate System	Laboratory
	Normal	[0.0, 0.0, 1.0] m,m,m
	Section Mode	Single Section
	Displayed Index	-1

line	^-1 Single section		Offset	0.0 m
	^-2 separation/attachment		Line Quality	90.0
			Line Strength	0.0
			Minimum Line Length	0.0 m
			Display Attachment Lines	true
			Display Separation Lines	true
^-1 Scalar Physical Quantity			Value	0.0 m
+-11 Solvers				
+-1 Partitioning			Solver Frozen	false
			Partitioning Method	Per-Region
+-2 Wall Distance			Solver Frozen	false
			Verbosity	0
			Minimum Tree Size Threshold	500000
+-3 Segregated Flow			Freeze Flow	false
			Reconstruction Frozen	false
			Reconstruction Zeroed	false
			Temporary Storage Retained	false
			Continuity Initialization	false
+-1 Velocity			Under-Relaxation Factor	0.7
			Dynamic Local Under-Relaxation	false
+-1 Under-Relaxation			Ramp Method	No Ramp
Factor Ramp				
^-2 AMG Linear Solver			Max Cycles	30
			Verbosity	None
			Enable Direct Solver	false
			Maximum Direct Solver Equations	32
			Convergence Tolerance	0.1
			Epsilon	0.0
			Cycle Type	Flex Cycle
			Group Size Control	Auto
			Group Size	4
			Relaxation Scheme	Gauss-Seidel
			Acceleration Method	None
			Scaling	Disabled
^-1 Flex Cycle			Restriction Tolerance	0.9
			Prolongation Tolerance	0.5
			Sweeps	1
^-2 Pressure			Under-Relaxation Factor	0.3
			Pressure Reference Location	Automatic Selection
+-1 Under-Relaxation			Ramp Method	No Ramp
Factor Ramp				
^-2 AMG Linear Solver			Max Cycles	30
			Verbosity	None
			Enable Direct Solver	false
			Maximum Direct Solver Equations	32
			Convergence Tolerance	0.1
			Epsilon	0.0
			Cycle Type	V Cycle
			Group Size Control	Auto
			Group Size	4
			Relaxation Scheme	Gauss-Seidel
			Acceleration Method	Conjugate Gradient
			Scaling	Auto
^-1 V Cycle			Pre-Sweeps	1
			Post-Sweeps	1
			Max Levels	50
+-4 K-Omega Turbulence			Solver Frozen	false
			Reconstruction Frozen	false
			Reconstruction Zeroed	false
			Temporary Storage Retained	false
			Under-Relaxation Factor	0.8

		+--1 Under-Relaxation Factor	Ramp Method	No Ramp
Ramp				
		`-2 AMG Linear Solver	Max Cycles	30
			Verbosity	None
			Enable Direct Solver	false
			Maximum Direct Solver Equations	32
			Convergence Tolerance	0.1
			Epsilon	0.0
			Cycle Type	Flex Cycle
			Group Size Control	Auto
			Group Size	4
			Relaxation Scheme	Gauss-Seidel
			Acceleration Method	None
			Scaling	Disabled
		`-1 Flex Cycle	Restriction Tolerance	0.9
			Prolongation Tolerance	0.5
			Sweeps	1
		+--5 K-Omega Turbulent	Solver Frozen	false
Viscosity				
			Under-Relaxation Factor	1.0
			Maximum Ratio	100000.0
		`-6 GammaReTheta Transition	Solver Frozen	false
			Reconstruction Frozen	false
			Reconstruction Zeroed	false
			Temporary Storage Retained	false
			Under-Relaxation Factor	0.8
			Freestream Update Frequency	1
			Freestream Update Policy	Iteration
		+--1 Under-Relaxation Factor	Ramp Method	No Ramp
Ramp				
		`-2 AMG Linear Solver	Max Cycles	30
			Verbosity	None
			Enable Direct Solver	false
			Maximum Direct Solver Equations	32
			Convergence Tolerance	0.1
			Epsilon	0.0
			Cycle Type	Flex Cycle
			Group Size Control	Auto
			Group Size	4
			Relaxation Scheme	Gauss-Seidel
			Acceleration Method	None
			Scaling	Disabled
		`-1 Flex Cycle	Restriction Tolerance	0.9
			Prolongation Tolerance	0.5
			Sweeps	1
		+--12 Stopping Criteria		
		+--1 Maximum Steps	Enabled	true
			Maximum Steps	3000
			Logical Rule	Or
			Criterion Satisfied	true
		`-2 Stop File	Enabled	true
			Stop Inner Iterations	true
			Path	ABORT
			Logical Rule	Or
			Criterion Satisfied	false
		+--13 Solution Histories		
		+--14 Solution Views		
		`-1 Current Solution	Iteration	3000
			Time Step	0
			Solution Time	0.0
		+--15 Reports	Reports	2
		+--1 Drag Force	Units	N

		Coordinate System	Laboratory
		Direction	[1.0, 0.0, 0.0]
		Force Option	Pressure + Shear
		Reference Pressure	0.0 Pa
		Number of Bands	0
		Representation	Initial Surface
		Smooth Values	false
		Units	m^2
		View Up	[0.0, 1.0, 0.0] m,m,m
		Normal	[1.0, 0.0, 0.0] m,m,m
		Coordinate System	Laboratory
		Representation	Initial Surface
		Monitors	11
		Monitors To Print	[Continuity, X-momentum, Y-momentum, Z-momentum, Tke, Sdr, Intermittency, ReTheta_t, Drag Force Monitor]
		Output Direction	Horizontal
		Heading Print Frequency	10
		Enabled	true
		Trigger	Iteration
		Normalization Option	Off
		Maximum Plot Samples	5000
		Iteration Frequency	1
		Start Iteration	0
		Enable Stop	false
		Stop Iteration	0
		Maximum Plot Samples	5000
		Maximum Plot Samples	5000
		Representations	
		Geometry	
		Latest	
		Initial Surface	Faces 265634
			Edges 1517
		Regions	
		Region 2	Faces 265634
			Edges 1517
		Boundaries	
		Floor	Faces 1111
		Inlet	Faces 44
		Model1 Body	Faces 250626
		Model1 Canopy	Faces 981
		Model1 Leading	Faces 11415
		Model1 Trailing	Faces 1165
		Outlet	Faces 44
		Walls	Faces 248
		Feature Curves	
		Feature Curve 1	Edges 528
		Feature Curve 2	Edges 989
		Remeshed Surface	Faces 1129884
			Edges 6633
		Regions	
		Region 2	Faces 1129884
			Edges 6633
		Boundaries	
		Floor	Faces 205451
		Inlet	Faces 6318
		Model1 Body	Faces 622456
		Model1 Canopy	Faces 11933
		Model1 Leading	Faces 184484

			+-6 Model1 Trailing	Faces	37813
Edge			+-7 Outlet	Faces	6291
			`-8 Walls	Faces	55138
			`-2 Feature Curves		
			+-1 Feature Curve 1	Edges	5379
			`-2 Feature Curve 2	Edges	1254
			`-4 Volume Mesh	Cells	13845993
				Interior Faces	41339807
				Vertices	14325205
			+-1 Finite Volume Regions		
			`-1 Region 2	Cells	13845993
				Interior Faces	41339807
				Vertices	14325205
			`-1 Finite Volume		
Boundaries					
			+-1 Floor	Faces	122476
			+-2 Inlet	Faces	1920
			+-3 Model1 Body	Faces	447082
			+-4 Model1 Canopy	Faces	7167
			+-5 Model1 Leading	Faces	81013
Edge					
			+-6 Model1 Trailing	Faces	21428
Edge					
			+-7 Outlet	Faces	1920
			`-8 Walls	Faces	17081
			`-2 Cell Sets		
+-18 Coordinate Systems					
			`-1 Laboratory		
			`-1 Local Coordinate		
Systems					
+-19 Units			Preferred System	Systeme Internationale	
+-20 Tables			Tables	0	
+-21 Field Functions					
			+-1 FreeStreamEdge	Function Name	FreeStreamEdge
				Inverse Distance Weight	false
				Type	Scalar
				Assembly Code	(if (> \${WallDistance} 0.01) 1 0)
				Definition	(\${WallDistance}>0.01) ? 1:0
				Ignore Boundary Values	false
			+-2 Absolute Pressure	Function Name	AbsolutePressure
				Inverse Distance Weight	false
				Type	Scalar
			+-3 Absolute Total Pressure	Function Name	AbsoluteTotalPressure
				Inverse Distance Weight	false
				Type	Scalar
			+-4 Area	Function Name	Area
				Inverse Distance Weight	false
				Type	Vector
			+-5 Axial Velocity	Function Name	AxialVelocity
				Inverse Distance Weight	false
				Type	Scalar
			+-6 Bad Cell Indicator	Function Name	BadCellFlag
				Inverse Distance Weight	false
				Type	Scalar
			+-7 Boundary Circumferential	Function Name	BCB_BinIndex
Bin Index					
				Inverse Distance Weight	false
				Type	Scalar
			+-8 Boundary Circumferential	Function Name	BCB_BinPitch
Bin Pitch					
				Inverse Distance Weight	false

		Type	Scalar
+-9	Boundary Circumferential	Function Name	BCB_Theta
Bin Theta		Inverse Distance Weight	false
		Type	Scalar
+-10	Boundary Index	Function Name	BoundaryIndex
		Inverse Distance Weight	false
		Type	Scalar
+-11	Boundary Sliver Cell	Function Name	BoundarySliverCellIndicator
Indicator		Inverse Distance Weight	false
		Type	Scalar
		Cell Quality Tolerance	1.0E-6
+-12	Cell Aspect Ratio	Function Name	CellAspectRatio
		Inverse Distance Weight	false
		Type	Scalar
+-13	Cell Index	Function Name	LocalCellIndex
		Inverse Distance Weight	false
		Type	Scalar
+-14	Cell Part Index	Function Name	CellGeometryPartIndex
		Inverse Distance Weight	false
		Type	Scalar
+-15	Cell Quality	Function Name	CellQuality
		Inverse Distance Weight	false
		Type	Scalar
+-16	Cell Relative Velocity	Function Name	CellRelativeVelocity
		Inverse Distance Weight	false
		Type	Vector
+-17	Centroid	Function Name	Centroid
		Inverse Distance Weight	false
		Type	Position
+-18	Chevron Quality	Function Name	ChevronQuality
Indicator		Inverse Distance Weight	false
		Type	Scalar
		Chevron Threshold Angle	1.396263 radian
		Cell Aspect Ratio Tolerance	0.1
+-19	ChimeraGridTestData	Function Name	ChimeraGridTestData
		Inverse Distance Weight	false
		Type	Scalar
+-20	Connected Cells	Function Name	ConnectedCells
		Inverse Distance Weight	false
		Type	Scalar
+-21	Density	Function Name	Density
		Inverse Distance Weight	false
		Type	Scalar
+-22	DeviationDistance.Root	Function Name	DeviationDistance.Root
		Inverse Distance Weight	false
		Type	Scalar
+-23	Dynamic Viscosity	Function Name	DynamicViscosity
		Inverse Distance Weight	false
		Type	Scalar
+-24	EdgeLength 3	Function Name	EdgeLength
		Inverse Distance Weight	false
		Type	Scalar
		Edge Length Option	Average
+-25	Effective Intermittency	Function Name	EffectiveIntermittency
		Inverse Distance Weight	false
		Type	Scalar
+-26	Effective Viscosity	Function Name	EffectiveViscosity
		Inverse Distance Weight	false
		Type	Scalar

+-27	Element Type	Function Name	ElemType
		Inverse Distance Weight	false
		Type	Scalar
+-28	Face Index	Function Name	LocalFaceIndex
		Inverse Distance Weight	false
		Type	Scalar
+-29	Face Part Surface Index	Function Name	FacePartSurfaceIndex
		Inverse Distance Weight	false
		Type	Scalar
+-30	Face Type	Function Name	FaceType
		Inverse Distance Weight	false
		Type	Scalar
+-31	Face Validity	Function Name	FaceValidity
		Inverse Distance Weight	false
		Type	Scalar
+-32	FaceProximity	Function Name	FaceProximity
		Inverse Distance Weight	false
		Type	Scalar
+-33	FaceQuality 3	Function Name	FaceQuality
		Inverse Distance Weight	false
		Type	Scalar
+-34	Flength Correlation	Function Name	FlengthCorrelation
		Inverse Distance Weight	false
		Type	Scalar
+-35	Flow Direction	Function Name	FlowAngle
		Inverse Distance Weight	false
		Type	Vector
+-36	FreeEdges 3	Function Name	FreeEdges
		Inverse Distance Weight	false
		Type	Scalar
+-37	Gamma Re_theta	Function Name	FreestreamCriticalTransitionRe
	Freestream Critical TransitionRe		
		Inverse Distance Weight	false
		Type	Scalar
+-38	Gauss/LSQ gradient	Function Name	GaussLSQ::Beta
	blending factor		
		Inverse Distance Weight	false
		Type	Scalar
+-39	Helicity	Function Name	Helicity
		Inverse Distance Weight	false
		Type	Scalar
+-40	Interface Parent	Function Name	InterfaceParentBoundaryIndex
	Boundary Index		
		Inverse Distance Weight	false
		Type	Scalar
+-41	Interface Parent Face	Function Name	InterfaceParent
	Index		
		Inverse Distance Weight	false
		Type	Scalar
+-42	Intermittency	Function Name	Intermittency
		Inverse Distance Weight	false
		Type	Scalar
+-43	Iteration	Function Name	Iteration
		Inverse Distance Weight	false
		Type	Scalar
+-44	Lamb Vector	Function Name	LambVector
		Inverse Distance Weight	false
		Type	Vector
+-45	Lambda 2 criterion	Function Name	Lambda2
		Inverse Distance Weight	false
		Type	Scalar
+-46	Least Squares Quality	Function Name	LeastSquaresQuality

		Inverse Distance Weight	false
		Type	Scalar
		Evaluation Method	Approximate
		Include Boundary Faces	false
	+-47	Mass Flow Rate	Function Name
		Inverse Distance Weight	FaceFlux
		Type	false
		Type	Scalar
	+-48	Mass Flux	Function Name
		Inverse Distance Weight	MassFlux
		Type	false
		Type	Scalar
	+-49	Mass Imbalance	Function Name
		Inverse Distance Weight	MassImbalance
		Type	false
		Type	Scalar
	+-50	Non-Conformal Interface	Function Name
			NonConformalInterfaceFace
	Faces		
		Inverse Distance Weight	false
		Type	Scalar
	+-51	NonmanifoldEdges 3	Function Name
		Inverse Distance Weight	NonmanifoldEdges
		Type	false
		Type	Scalar
	+-52	Overset Cell Status	Function Name
		Inverse Distance Weight	OversetCellStatus
		Type	false
		Type	Scalar
	+-53	Overset Cell Type	Function Name
		Inverse Distance Weight	OversetCellType
		Type	false
		Type	Scalar
	+-54	Overset Error Status	Function Name
		Inverse Distance Weight	OversetErrorStatus
		Type	false
		Type	Scalar
	+-55	Overset Grid Volume	Function Name
		Inverse Distance Weight	Overset Grid Volume
		Type	false
		Type	Scalar
	+-56	Partition	Function Name
		Inverse Distance Weight	PartitionId
		Type	false
		Type	Scalar
	+-57	Periodic Index	Function Name
		Inverse Distance Weight	PeriodicIndex
		Type	false
		Type	Scalar
	+-58	PiercedFaces	Function Name
		Inverse Distance Weight	PiercedFaces
		Type	false
		Type	Scalar
	+-59	Position	Function Name
		Inverse Distance Weight	Position
		Type	false
		Type	Position
	+-60	Pressure	Function Name
		Inverse Distance Weight	Pressure
		Type	false
		Type	Scalar
	+-61	Pressure Coefficient	Function Name
		Inverse Distance Weight	PressureCoefficient
		Type	false
		Type	Scalar
		Reference Density	1.225 kg/m^3
		Reference Pressure	0.0 Pa
		Reference Velocity	28.0 m/s
	+-62	Prism Layer Cells	Function Name
		Inverse Distance Weight	PrismLayerCells
		Type	false
		Type	Scalar
	+-63	Q-Criterion	Function Name
		Inverse Distance Weight	Qcriterion
		Type	false
		Type	Scalar
	+-64	Radial Coordinate	Function Name
		Inverse Distance Weight	RadialCoordinate
		Type	false
		Type	Scalar

+-65	Radial Velocity	Function Name	RadialVelocity
		Inverse Distance Weight	false
		Type	Scalar
+-66	Region Index	Function Name	RegionIndex
		Inverse Distance Weight	false
		Type	Scalar
+-67	Relative Tangential Velocity	Function Name	RelativeTangentialVelocity
		Inverse Distance Weight	false
		Type	Scalar
+-68	Relative Total Pressure	Function Name	RelativeTotalPressure
		Inverse Distance Weight	false
		Type	Scalar
+-69	Relative Velocity	Function Name	RelativeVelocity
		Inverse Distance Weight	false
		Type	Vector
+-70	Report: Drag Force	Function Name	DragForceReport
		Inverse Distance Weight	false
		Type	Scalar
+-71	Report: Frontal Area	Function Name	FrontalAreaReport
		Inverse Distance Weight	false
		Type	Scalar
+-72	ReThetaCrit Correlation	Function Name	ReThetaCritCorrelation
		Inverse Distance Weight	false
		Type	Scalar
+-73	ReThetaT Correlation	Function Name	ReThetaTCorrelation
		Inverse Distance Weight	false
		Type	Scalar
+-74	Skewness Angle	Function Name	SkewnessAngle
		Inverse Distance Weight	false
		Type	Scalar
+-75	Skin Friction Coefficient	Function Name	SkinFrictionCoefficient
		Inverse Distance Weight	false
		Type	Scalar
		Reference Density	1.0 kg/m^3
		Reference Velocity	1.0 m/s
+-76	Specific Dissipation Rate	Function Name	SpecificDissipationRate
		Inverse Distance Weight	false
		Type	Scalar
+-77	Specific Dissipation Rate Reynolds Number	Function Name	SpecificDissipationRateRe
		Inverse Distance Weight	false
		Type	Scalar
+-78	Static Pressure	Function Name	StaticPressure
		Inverse Distance Weight	false
		Type	Scalar
+-79	SurfacePatch 3	Function Name	SurfacePatch
		Inverse Distance Weight	false
		Type	Scalar
+-80	Tangential Velocity	Function Name	TangentialVelocity
		Inverse Distance Weight	false
		Type	Scalar
+-81	Total Pressure	Function Name	TotalPressure
		Inverse Distance Weight	false
		Type	Scalar
+-82	Total Pressure Coefficient	Function Name	TotalPressureCoefficient
		Inverse Distance Weight	false
		Type	Scalar
		Reference Density	1.0 kg/m^3
		Reference Pressure	0.0 Pa
		Reference Velocity	1.0 m/s

+-83 Transition Momentum Thickness Reynolds Number	Function Name	TransitionReTheta
	Inverse Distance Weight	false
	Type	Scalar
+-84 Turbulence Intensity	Function Name	TurbulenceIntensity
	Inverse Distance Weight	false
	Type	Scalar
+-85 Turbulent Charge	Function Name	TurbulentCharge
	Inverse Distance Weight	false
	Type	Scalar
+-86 Turbulent Kinetic Energy	Function Name	TurbulentKineticEnergy
	Inverse Distance Weight	false
	Type	Scalar
+-87 Turbulent Viscosity	Function Name	TurbulentViscosity
	Inverse Distance Weight	false
	Type	Scalar
+-88 Turbulent Viscosity Ratio	Function Name	TurbulentViscosityRatio
	Inverse Distance Weight	false
	Type	Scalar
+-89 Unmatched Edge Count	Function Name	UnmatchedEdgeCount
	Inverse Distance Weight	false
	Type	Scalar
+-90 Ustar	Function Name	Ustar
	Inverse Distance Weight	false
	Type	Scalar
+-91 Velocity	Function Name	Velocity
	Inverse Distance Weight	false
	Type	Vector
+-92 Virtual Local Heat Transfer Coefficient	Function Name	VirtualLocalHeatTransferCoefficient
	Inverse Distance Weight	false
	Type	Scalar
	Reference Specific Heat	1000.0 J/kg-K
	Reference Prandtl Number	0.7
	Reference Turbulent Prandtl Number	0.9
+-93 Visualization Cell Status	Function Name	VisCellStatus
	Inverse Distance Weight	false
	Type	Scalar
+-94 Volume	Function Name	Volume
	Inverse Distance Weight	false
	Type	Scalar
+-95 Volume Change	Function Name	VolumeChange
	Inverse Distance Weight	false
	Type	Scalar
+-96 Vorticity	Function Name	VorticityVector
	Inverse Distance Weight	false
	Type	Vector
+-97 Wall Distance	Function Name	WallDistance
	Inverse Distance Weight	false
	Type	Scalar
+-98 Wall Shear Stress	Function Name	WallShearStress
	Inverse Distance Weight	false
	Type	Vector
+-99 Wall Y+	Function Name	WallYplus
	Inverse Distance Weight	false
	Type	Scalar
+-100 Zero Face Area Indicator	Function Name	ZeroFaceAreaIndicator
	Inverse Distance Weight	false
	Type	Scalar
	Zero Face Area Tolerance	1.0842021724855044E-16
+-22 Volume Shapes		

+-1	Block 1	Corner 1	[-0.6, -0.8, -0.3] m,m,m
		Corner 2	[5.0, 0.8, 2.0] m,m,m
		Coordinate System	Laboratory
+-2	Block 2	Corner 1	[-1.0, -0.8, -0.5] m,m,m
		Corner 2	[10.0, 1.2, 2.2] m,m,m
		Coordinate System	Laboratory
+-3	Block 3	Corner 1	[-1.4, -0.8, -0.7] m,m,m
		Corner 2	[15.0, 1.6, 2.4] m,m,m
		Coordinate System	Laboratory
+-4	Block 4	Corner 1	[-1.8, -0.8, -0.9] m,m,m
		Corner 2	[20.0, 2.0, 2.6] m,m,m
		Coordinate System	Laboratory
+-5	Block 5	Corner 1	[0.95, -0.65, 1.45] m,m,m
		Corner 2	[1.4, -0.55, 1.56] m,m,m
		Coordinate System	Laboratory
+-6	Block 6	Corner 1	[2.95, -0.65, 1.45] m,m,m
		Corner 2	[3.4, -0.55, 1.56] m,m,m
		Coordinate System	Laboratory
+-7	Block 7	Corner 1	[2.95, -0.65, 0.15] m,m,m
		Corner 2	[3.4, -0.55, 0.26] m,m,m
		Coordinate System	Laboratory
+-8	Block 8	Corner 1	[0.95, -0.65, 0.15] m,m,m
		Corner 2	[1.4, -0.55, 0.26] m,m,m
		Coordinate System	Laboratory
+-23	Data Set Functions	Data Directory	function_data
+-24	User Code		
+-25	Layouts		
+-1	default		
+-26	Data Mappers		
+-27	Motions		
+-1	Stationary		
+-28	Reference Frames		
+-1	Lab Reference Frame		
+-29	Filters		
+-30	Update Events	Event Count	0
		Event Names	

Solution

Accumulated CPU Time over all processes (s) 869664.0753800018
Elapsed Time (s) 28339.38052725792
Iterations 3000

Annexure D: Final Remarks

Due to time constraints, the vehicle built to compete in the Sasol Solar Challenge, was the baseline model as described in Section 7.2.1. The moulds for the vehicle were constructed exactly as defined by the CAD part of the model. These moulds were split so that the different body panels would fit exactly onto each other. During the manufacturing process of the vehicle, several implications became apparent. These included that the solar panels would not fit onto the top of the car, so the split line of the top body panel had to be adjusted. The bypass diodes were installed onto the wrong solar panels, so the bubble canopy of the vehicle was modified to be smaller. This resulted in a poorer canopy design, possibly resulting in higher drag, but the frontal area was decreased. This effect was simulated and the poorer canopy design had a 0.6% in drag force. This is negligible so the effects of the poorer aerodynamic shape and the reduction in the frontal area eliminated one another almost completely.

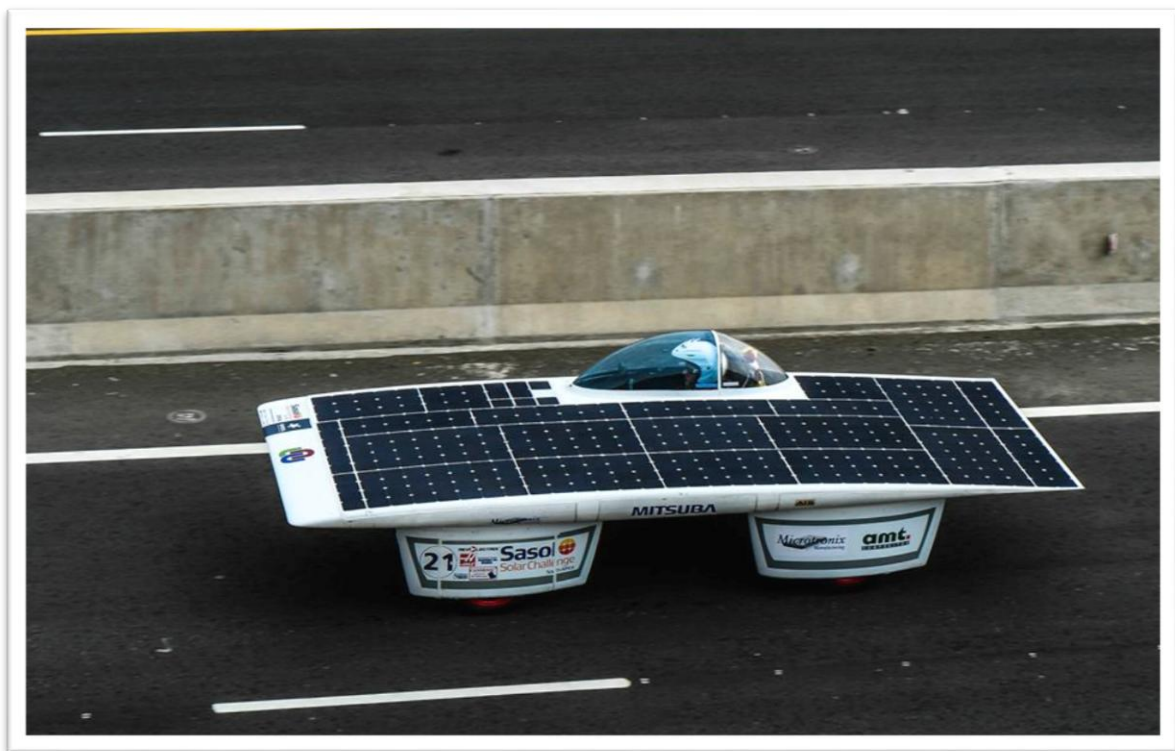


Figure 97 - NWU Solar Car competing in the Sasol Solar Challenge 2014

In Figure 97 the modified bubble canopy design can be observed. This is one of the main components that requires a redesign and the design should not only be modified. Further implication with the rear suspension

meant that the vehicle ride height was approximately 50 mm higher than initially designed for. This would also increase the drag force of the vehicle.

The centre of gravity was theoretically determined to be 617 mm behind the centre of the front wheels and 864 mm left of the left-hand wheel track. It is also 250 mm above ground level. This puts the static margin at almost 10% in front of the centre of the vehicle. This ensured that the vehicle was extremely stable on the road. When the vehicle was driven around the Zwartkops Raceway in Centurion, Pretoria, the top speed achieved was 130 km/h. This could be achieved without the vehicle experiencing any traction loss on the wheels and is testament to the stability of the aerodynamic design.

The results of the vehicle during the race is summarised as follows:

- Final over-all distance covered by the team in the vehicle was 2360 km. This puts the team in an over-all 4th place and second of the South African teams that participated.
- Record for the furthest distance travelled by a South African team in a single day was 416 km.

Finally, it should be noted that the vehicle would have performed better if more time was available to test the vehicle. If faults with the electrical system, such as the faulty battery pack, had been discovered earlier, it could have been corrected in time for the start of the race.

Below follows more photographs of the vehicle during the Sasol Solar Challenge 2014.



Figure 98 - Sirius X25 headed to Kroonstad on Day 1 of the Sasol Solar Challenge 2014

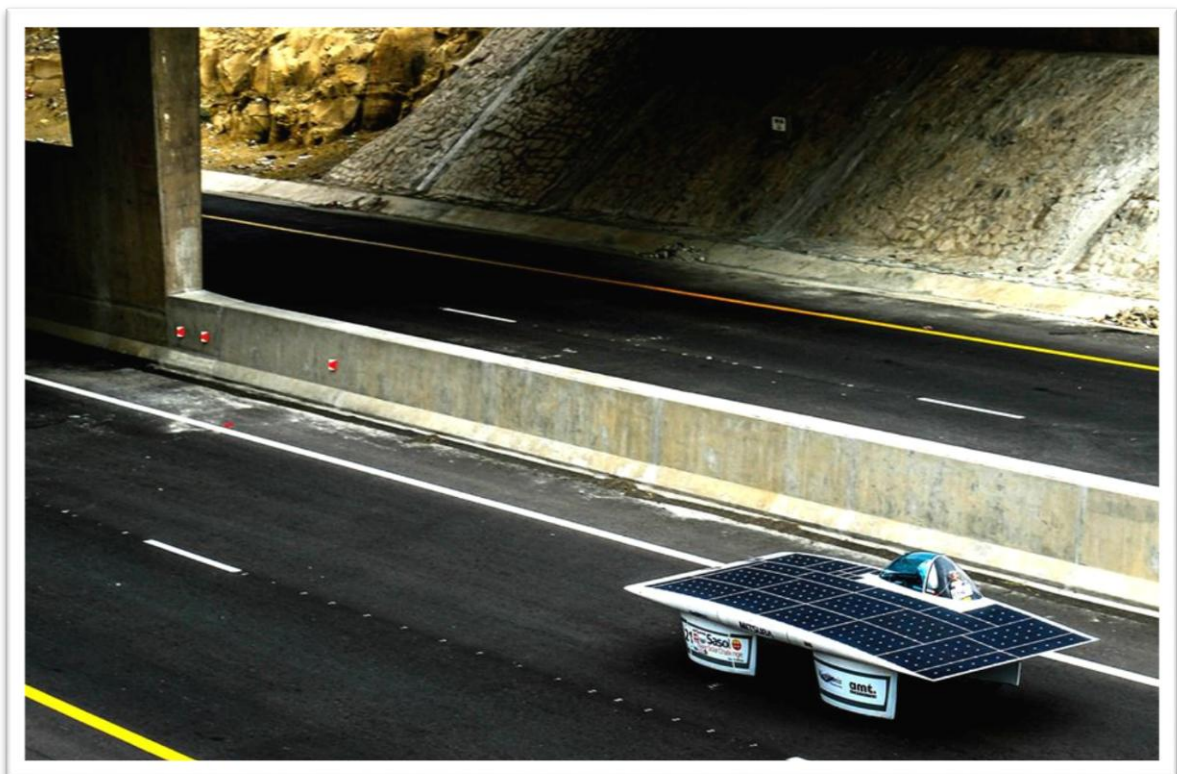


Figure 99 – Sirius X25 travelling on the N1 highway between Kroonstad and Bloemfontein, Free State.



Figure 100 – On route to Hanover in the Karoo



Figure 101 - Sirius X25 completing a loop in the Karoo



Figure 102 - The vehicle on its way to Port Elizabeth, Eastern Cape

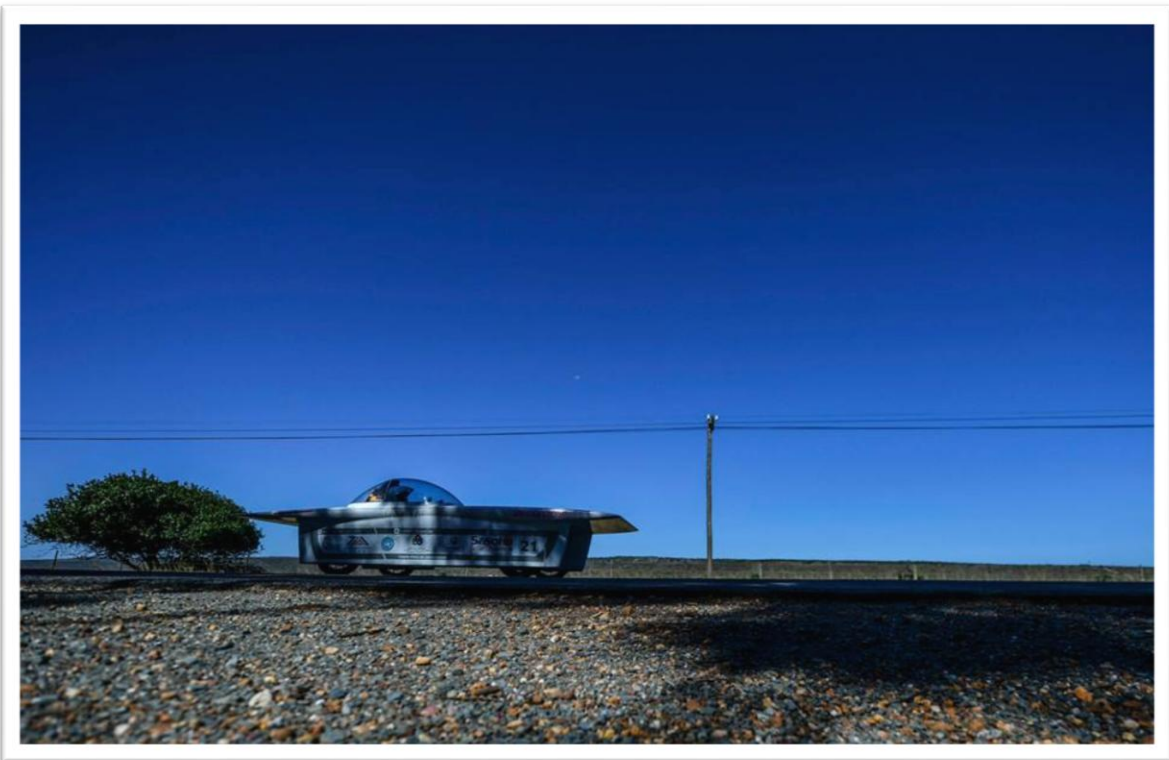


Figure 103 - Doing a loop between Heidelberg and Witsand in the Western Cape



Figure 104 - Control stop in Heidelberg, Western Cape



Figure 105 - Sirius X25 and team at the finish in Cape Town