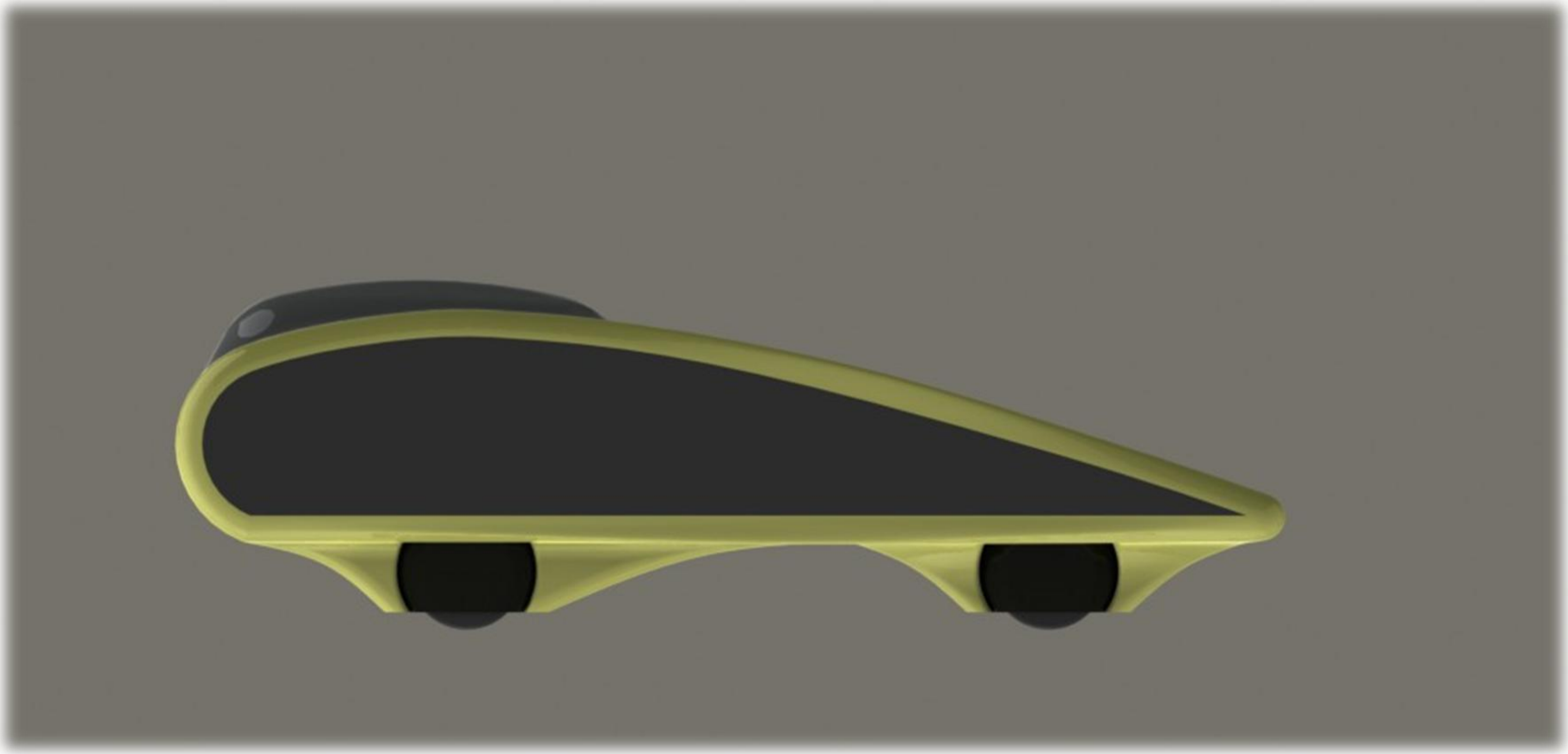
Design of the Solar Car



Author: Mohammad Foroughi

Table of Contents

Introduction

The purpose of the project was to explore the possibilities of CFD analysis, extend the knowledge about solving engineering problem and put into practice the theoretical skills gained previously.

The aim was to design a competitive vehicle bodyshell which complied with the requirements of WSC challenge class vehicle. The vehicle was intended to be accommodation the most aerodynamic solution to perform competitively. The final design was then tested with CosmosFloWorks where the suitable boundary condition and surface goals been set to simulate the real environment. The results from these tests were critically analyzed and explanation given.

The report consists of 4 parts, in the first part the description of the design is given with explanation why the particular solutions were chosen. Second part includes the results of the CFD tests together with analysis. The hand calculation for finding the maximum speed of vehicle are shown and explained in the third part. The fourth part consists of final design.

Part 1:

Design of the Solar Car

Methodology

In order to achieve the goals set at the beginning of the assignment - learning the practical skills of CFD using CosmosFloWorks, the project was divided into several separate tasks.

- General study about aerodynamics of solar vehicle and background of CFD.
- Choosing the realistic design for the solar vehicle including aerofoil section.
- Modelling the vehicle with SolidWorks 2008 - in order to prevent errors later when running CFD (CosmosFloWorks), the bodyshell was made from the beginning as one solid part and was an attempt not to have any assemblies, except from wheels, driver and battery which were given from beginning of the assignment.
- Calculating driver position, battery and the number of solar panels which will be fit to upper surface of bodyshell. The dimensions of each panel are 0.5 meters by 0.25 meters and each may be assumed to develop an effective propulsive power of 35watts after all losses have been taken into account.
- Selecting (SELIG 3021) aerofoil section and modifying it for bodyshell vehicle.

First step of design was to select suitable aerofoil section for the bodyshell vehicle to reduce the drag force and improve the aerodynamics of the vehicle. Important aspect of aerodynamic analysis is determining the drag and lift forces. Drag force is the force which applies in wind flow direction and the lift force which applies normal to the drag force.

As figure 1 shows, the aerofoil section was imported into SolidWorks 2008 as points by calculating upper and lower surface using Excel, then all the points were joined together in order to extrude the sketch. From there the aerofoil section was modified to make enough room for the driver position and to get smooth flat surface on the upper surface for fitting the panels in one straight line.

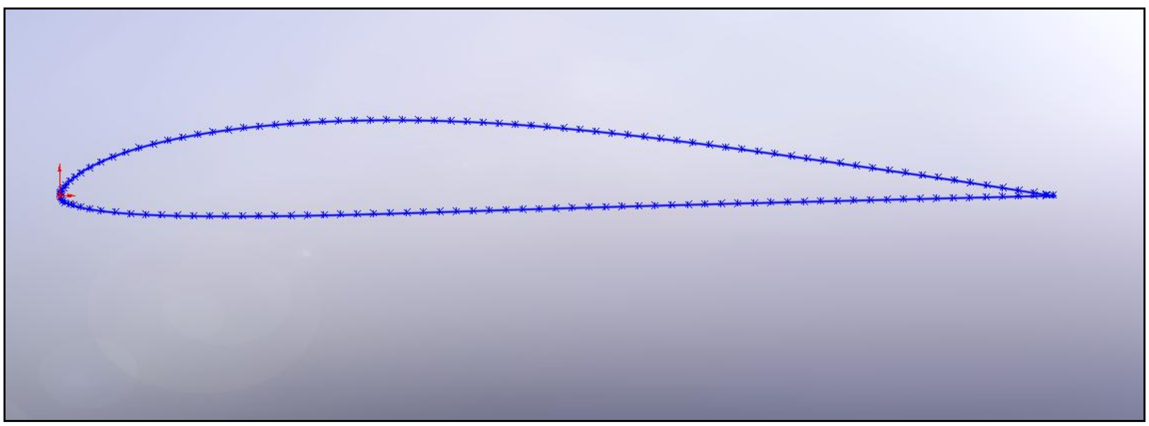


Figure 1.Sketch of Selig 3021

After extruding the sketch the driver cabin was created on front of the bodyshell with seat angle of 115.08., also small space to fit the battery which was mounted to the bodyshell vehicle. After that the positions of the wheels were created for the vehicle.

The next step was to smooth the sharp edges by using the fillet and chamfer tool to improve the aerodynamics of the vehicle.

Once the design was finished, the next step was to assemble the driver, battery and wheels.

Number of the wheels designed for this particular bodyshell was three - to make the vehicle lighter. These were assembled to the vehicle as shown in Figure 2.

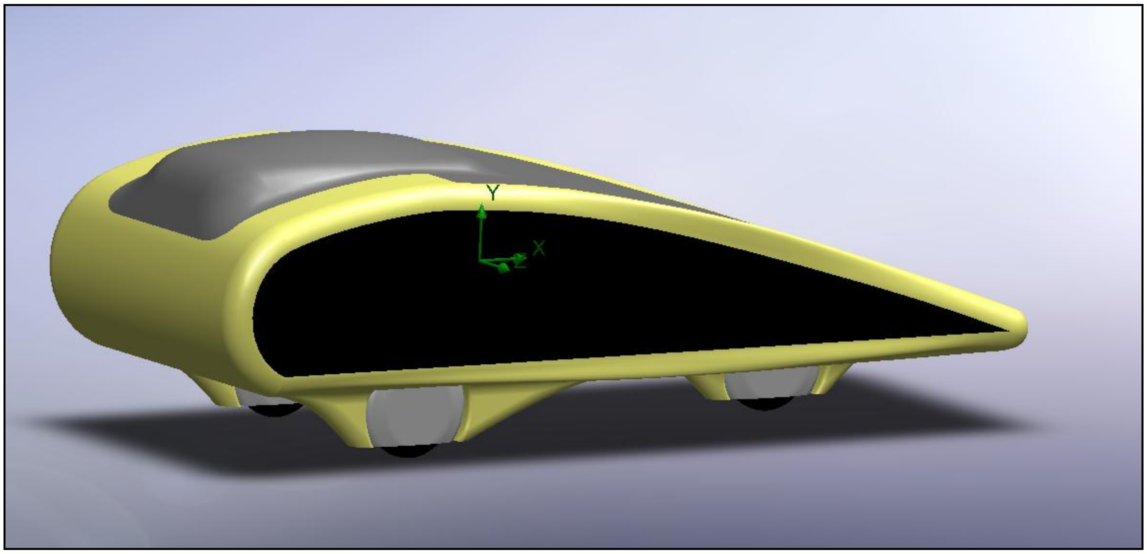


Figure 2, Wheels assembled to the vehicle

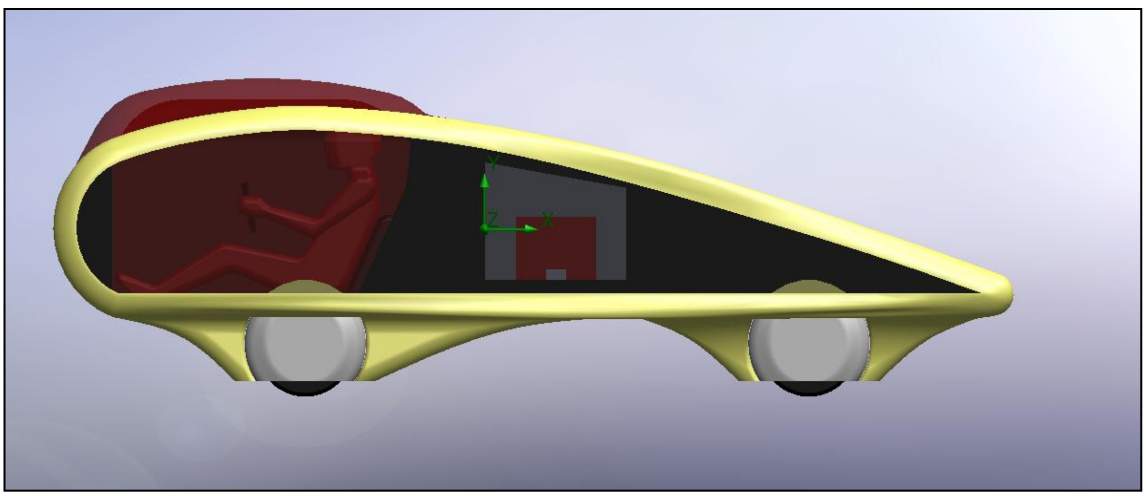


Figure 3, Vehicle with wheels, battery pack and driver

As shown in Figure 3, the vehicle was designed around the driver, battery and wheels which had been provided. According to the figures above, none of the provided parts such as driver and wheels have been redesigned and the aerodynamics of the wheels was kept as original.

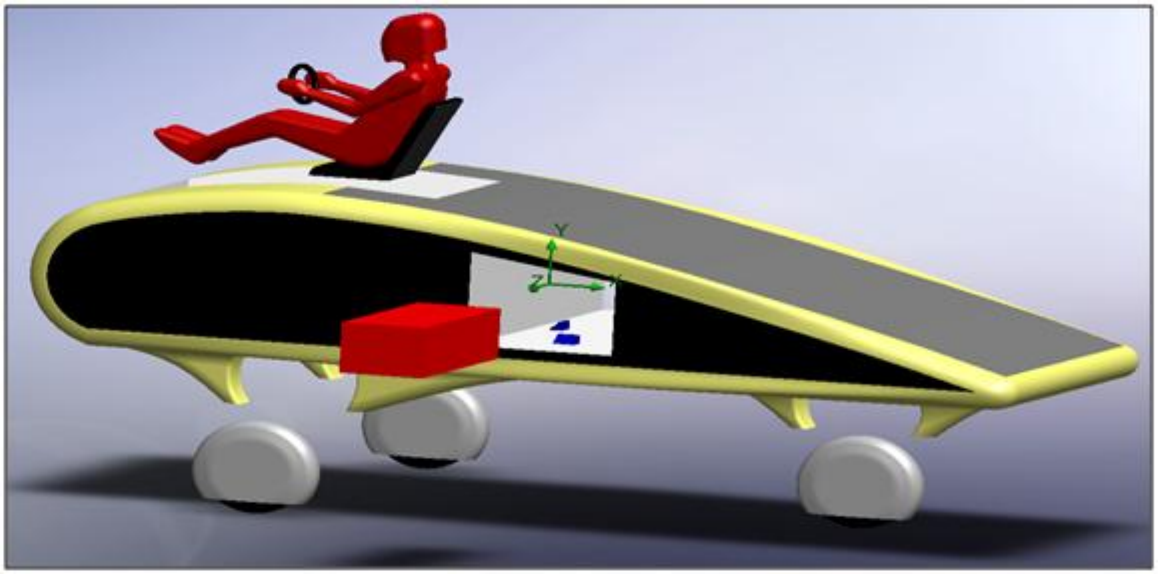


Figure 4, Exploded view of the solar vehicle

The Figure 4 shows the exploded view of the driver, battery and wheels which were assembled to the solar vehicle.

Overall dimensions of the vehicle:

4765mm long by 1800mm wide by 1505mm in height.
The driver is totally enclosed within the bodyshell and the driver's head has been protected by a canopy with a minimum all-round clearance of 100mm and ground clearance of the vehicle is 375mm.

Note:
The drawing of the solar vehicle is available at the end of the report which indicates the dimensions of the car, driver, battery, wheels and the ground clearance.

Part 2:

CFD Analysis of the Solar Car

After finishing the design of the solar vehicle the next step was to run the CFD analysis by opening the model in CosmosFlowWorks.

- Unit system: SI (m-kg-s) was selected for the unit system.
- Analysis type: external flow and also, for ignoring the driver and battery within the bodyshell the close cavities needed to be considered there for the exclude cavities without the flow condition and exclude internal space been selected for the model.
- Fluid: from fluid wizard option gas/air was selected and type of the flow was set to turbulent only.
- Wall condition: adiabatic wall with roughness of 0 micrometer.
- Initial and ambit condition: temperature has been set to 20C or 293.15 Kelvin, pressure to 101325Pa and the velocity in X direction to 15m/s.
- Mesh size: mesh sizes of 2, 3 and 4 were selected for the analysis (Figures 4, 5 and 6).

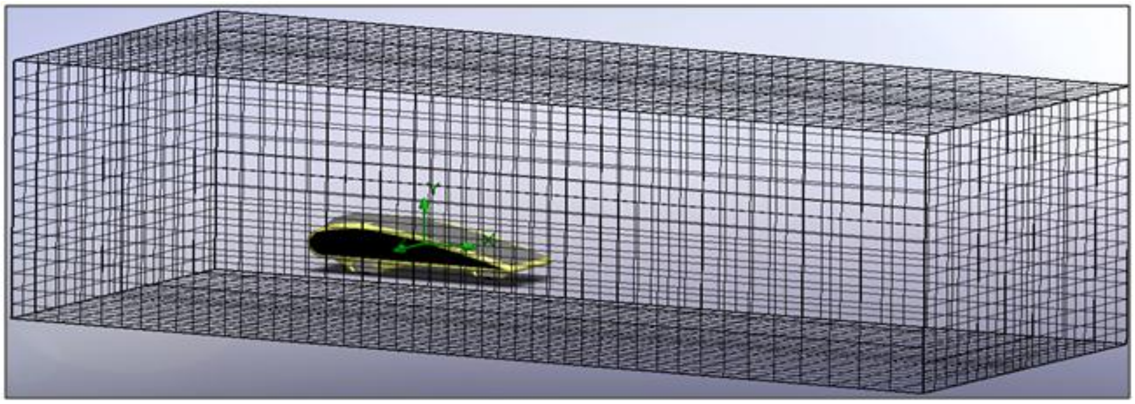


Figure 4, Mesh size 2

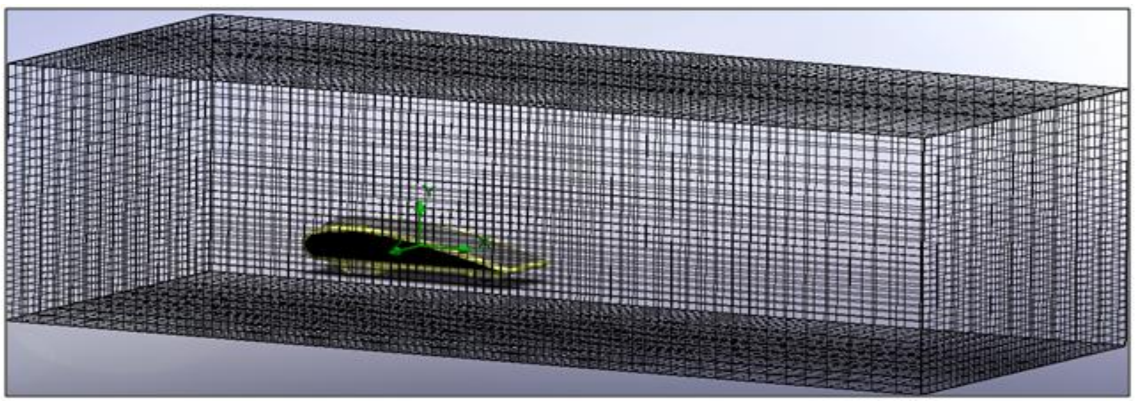


Figure 5, Mesh size 3

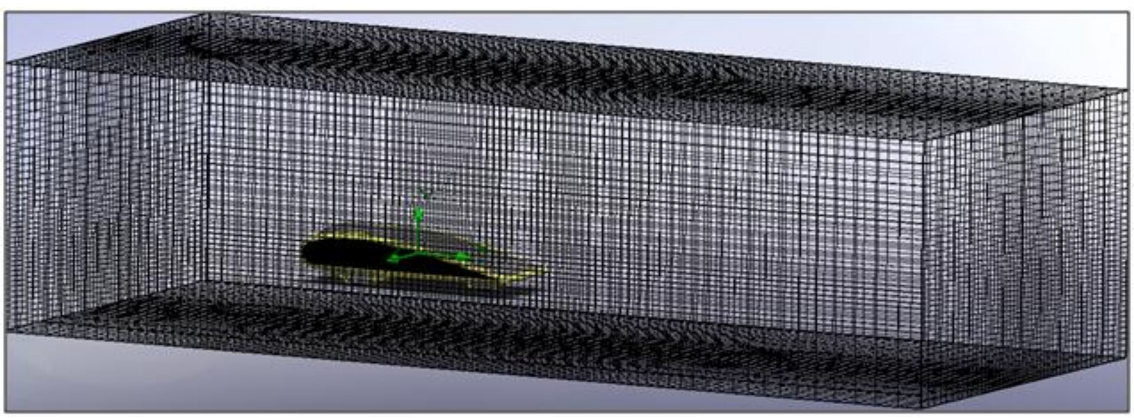


Figure 6, Mesh size 4

- Computational Domain: adjustment for computational domain should be done by considering the vehicle size and requirement (Figure 7). Table 8 shows the dimensions of the computational domain which were set for the solar vehicle.

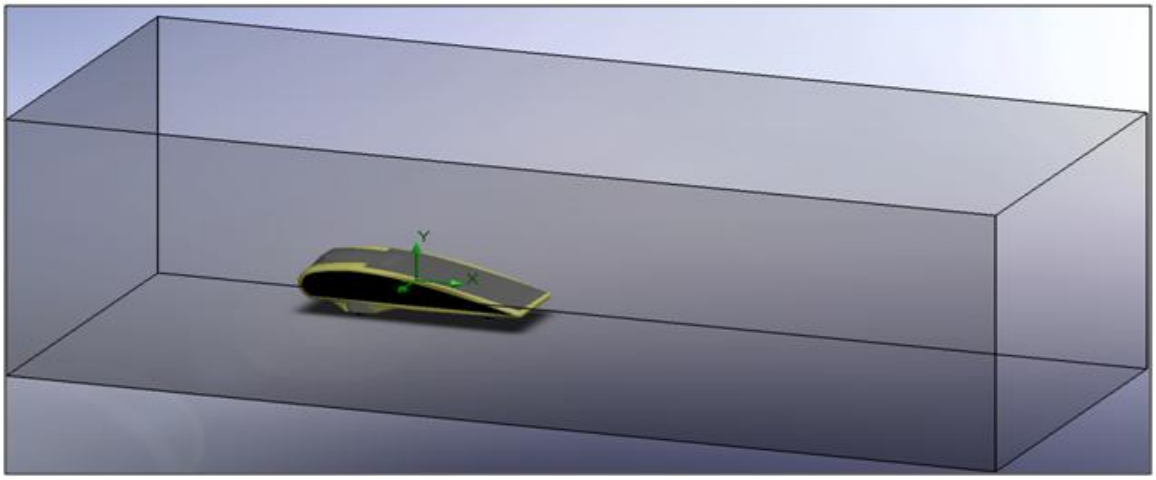


Figure 7, Computational Domain

Domain Size	
X min	-7m
X max	14m
Y min	-2m
Y max	5m
Z min	-4m
Z max	4.5m

Table 1, Computational domain dimensions

The volume of the domain can be calculated by using the height, length and width values.

Flow domain volume = (7+14) X (2+5) X (4+4.5) = 1249.5 units^3

- Calculation control option: in this section the analysis interval (travel) was changed to manual with value of 1.5 (as it was required).

- Boundary conditions: as this is an external flow analysis no boundary conditions are required.

- Goals: total force both in X and Y direction has been set in order to find the results of the lift and drag forces respectively, which applied on vehicle in required condition.

For doing the CFD analysis a wind tunnel should be assumed around the vehicle body, the important things in aerodynamic analysis are how to determine the drag and lift forces. Drag force is a force which applies in the wind flow direction and the lift force is perpendicular to the drag force. The solution which FloWorks uses to approach the problem is finite element analysis. In finite element analysis solid body will be splinted into cells. The number of these can be different, depending on circumstances such as accuracy, shape of the body and etc. In this project it depends on the size of mesh which could be 2,3,4,5. By changing the size of mesh the number of cells would change as well.

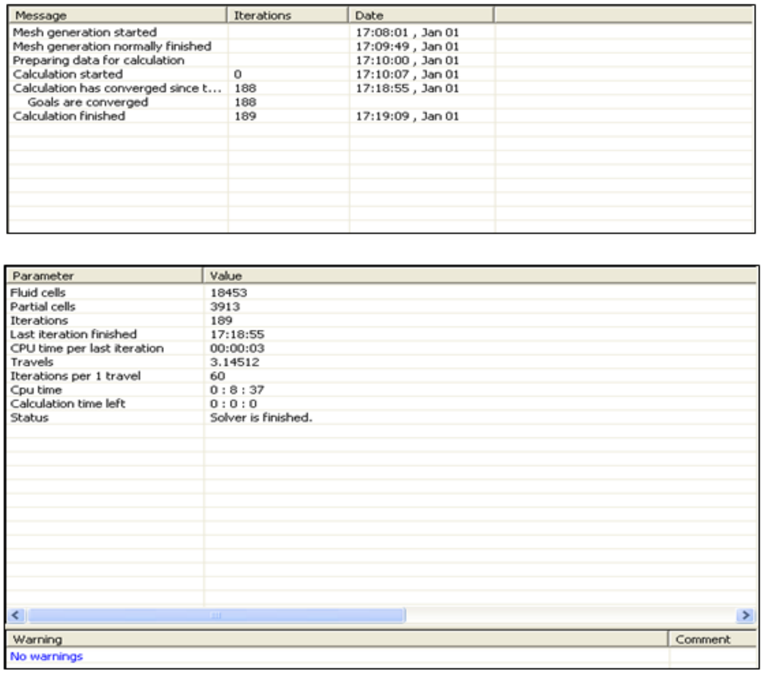
Before doing the CFD analysis some circumstances should be assumed such as adiabatic walls which avoid heat energy to go out of the system. The flow velocity is 15m/s and the temperature is 20 C or 293.15 Kelvin.

The CFD analysis of the vehicle body has been done with three different mesh sizes (2, 3 and 4).

The dimensions of the wind tunnel (Computational Domain) are directly affecting the number of cells and the amount of calculations. So in this analysis, as the forces in Z direction were not highly important, the height of the tunnel (Computational domain) was assumed as small as possible, to avoid the large amount of calculations.

The type of flow was assumed as turbulent air. After calculations finished, the surface parameter, surface plot and cut plot were derived to support the results.

Mesh size 2:



Message	Iterations	Date
Mesh generation started		17:08:01 , Jan 01
Mesh generation normally finished		17:09:49 , Jan 01
Preparing data for calculation		17:10:00 , Jan 01
Calculation started	0	17:10:07 , Jan 01
Calculation has converged since t...	188	17:18:55 , Jan 01
Goals are converged	188	
Calculation finished	189	17:19:09 , Jan 01

Parameter	Value
Fluid cells	18453
Partial cells	3913
Iterations	189
Last iteration finished	17:18:55
CPU time per last iteration	00:00:03
Travels	3.14512
Iterations per 1 travel	60
Cpu time	0 : 8 : 37
Calculation time left	0 : 0 : 0
Status	Solver is finished.

Warning	Comment
No warnings	

Figure 9

Figure 9 shows the number of Fluid cells, iterations, travels, and iterations per 1 travel.

Figure 10, X and Y goal plot graph

The graph above (Figure 10) shows the goal plot for mesh 2.

Mesh size	Fluid cell	Iterations per travel	Travels	Iterations	CPU Time	Drag	Lift
2	18453	60	3.145	189	08:37	68.8153	111.644

Surface parameters for mesh size 2:

Local parameters					
Parameter	**Minimum**	**Maximum**	**Average**	**Bulk Average**	**Surface area [m^2]**
Pressure [Pa]	101156	101407	101309		26.3318
Total Pressure [Pa]	101156	101407	101309		26.3318
Temperature [K]	293.206	295.466	293.25		26.3318
Density [kg/m^3]	1.19377	1.20473	1.20356		26.3318
Velocity [m/s]	0	0	0		26.3318
X-component of Velocity [m/s]	0	0	0		26.3318
Y-component of Velocity [m/s]	0	0	0		26.3318
Z-component of Velocity [m/s]	0	0	0		26.3318
Mach Number [ ]	0	0	0		26.3318
Heat Transfer Coefficient [W/m^2/K]	0	0	0		26.3536
Shear Stress [Pa]	2.06528E-09	6.21433	0.364095		26.3536
Fluid Temperature [K]	293.206	295.466	293.25		26.3318
Total Temperature [K]	293.206	295.466	293.25		26.3318
Heat Flux [W/m^2]	0	0	0		26.3536
X-component of Heat Flux [W/m^2]	0	0	0		26.3536
Y-component of Heat Flux [W/m^2]	0	0	0		26.3536
Z-component of Heat Flux [W/m^2]	0	0	0		26.3536

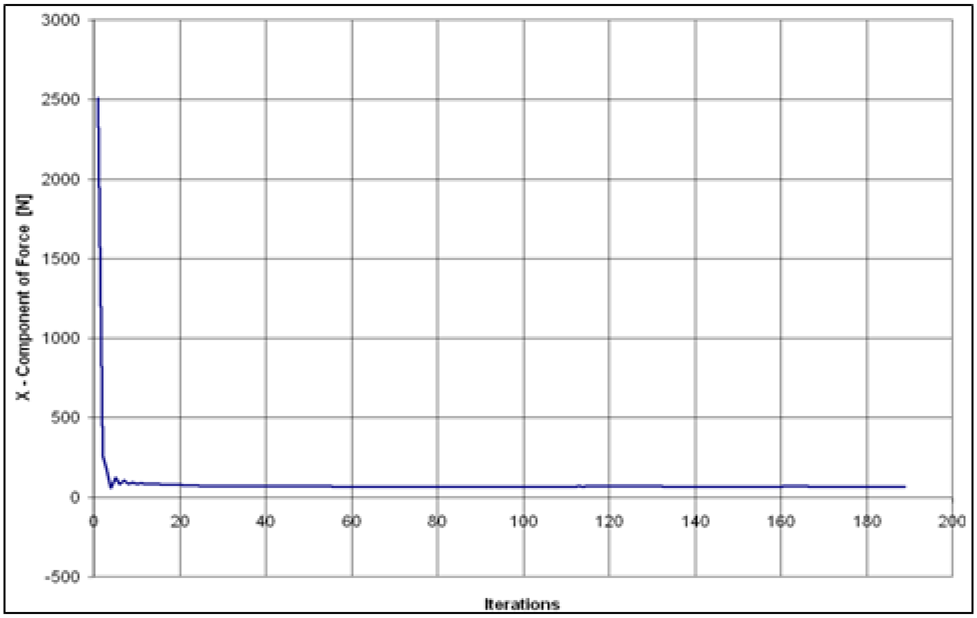
Integral parameters					
Parameter	Value	X-component	Y-component	Z-component	Surface area [m^2]
Heat Transfer Rate [W]	0	0	0	0	26.3536
Normal Force [N]	128.18	59.6257	111.974	18.3499	26.3536
Shear Force [N]	9.19594	9.18956	-0.329928	-0.0919276	26.3536
Force [N]	132.413	68.8153	111.644	18.2579	26.3536
Torque [N*m]	35.4457	-17.976	19.3191	23.665	26.3536
Surface Area [m^2]	26.3536	-0.00150007	0.0454803	-0.000526401	26.3536
Torque of Normal Force [N*m]	34.805	-17.9611	17.8945	23.8449	26.3536
Torque of Shear Force [N*m]	1.43603	-0.0148756	1.42465	-0.179807	26.3536
Uniformity Index [ ]	1				26.3318
CAD Fluid Area [m^2]	28.1851				28.1851

X and Y component of force for mesh size 2:

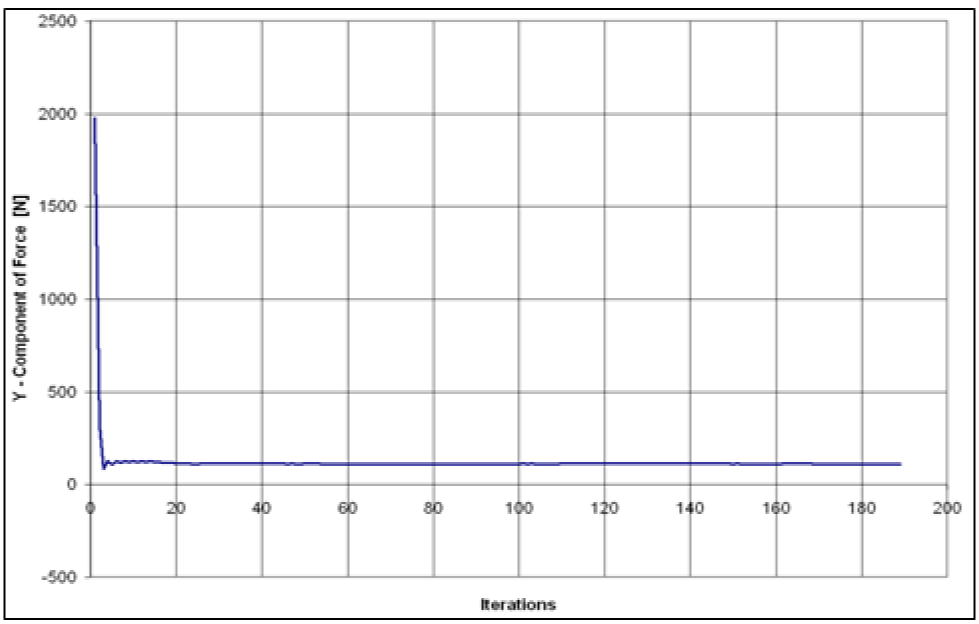
Iterations: 189

Analysis interval: 91

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]	Use In Convergence	Delta	Criteria
SG X - Component of Force	[N]	68.81527435	68.81484042	68.24104457	69.38390679	100	Yes	1.14286222	3.285184432
SG Y - Component of Force	[N]	111.6441114	112.2063353	111.3796627	113.1576511	100	Yes	1.301428016	1.386636983

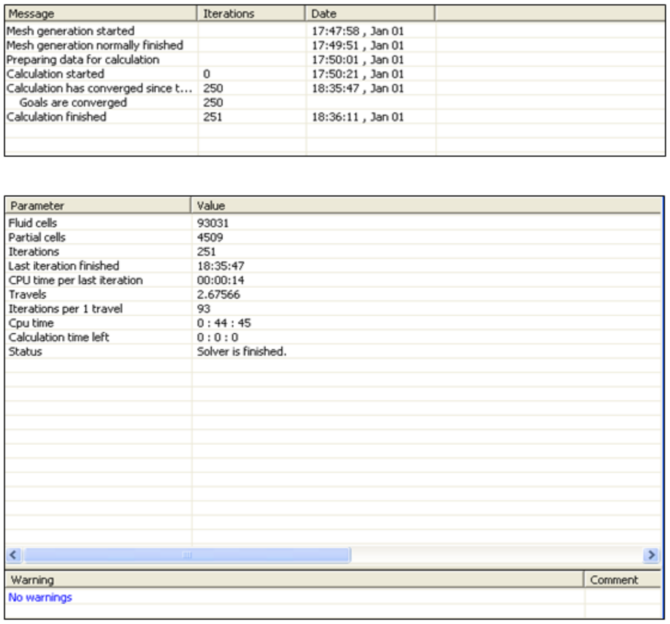


SG X - Component of Force 1



SG Y - Component of Force 1

Mesh size 3:



Message	Iterations	Date
Mesh generation started		17:47:58 , Jan 01
Mesh generation normally finished		17:49:51 , Jan 01
Preparing data for calculation		17:50:01 , Jan 01
Calculation started	0	17:50:21 , Jan 01
Calculation has converged since t...	250	18:35:47 , Jan 01
Goals are converged	250	
Calculation finished	251	18:36:11 , Jan 01

Parameter	Value
Fluid cells	93031
Partial cells	4509
Iterations	251
Last iteration finished	18:35:47
CPU time per last iteration	00:00:14
Travels	2.67566
Iterations per 1 travel	93
Cpu time	0 : 44 : 45
Calculation time left	0 : 0 : 0
Status	Solver is finished.

Warning	Comment
No warnings	

Figure 11

Figure 11 shows the number of Fluid cells, iterations, travels, and iterations per 1 travel for mesh size 3.

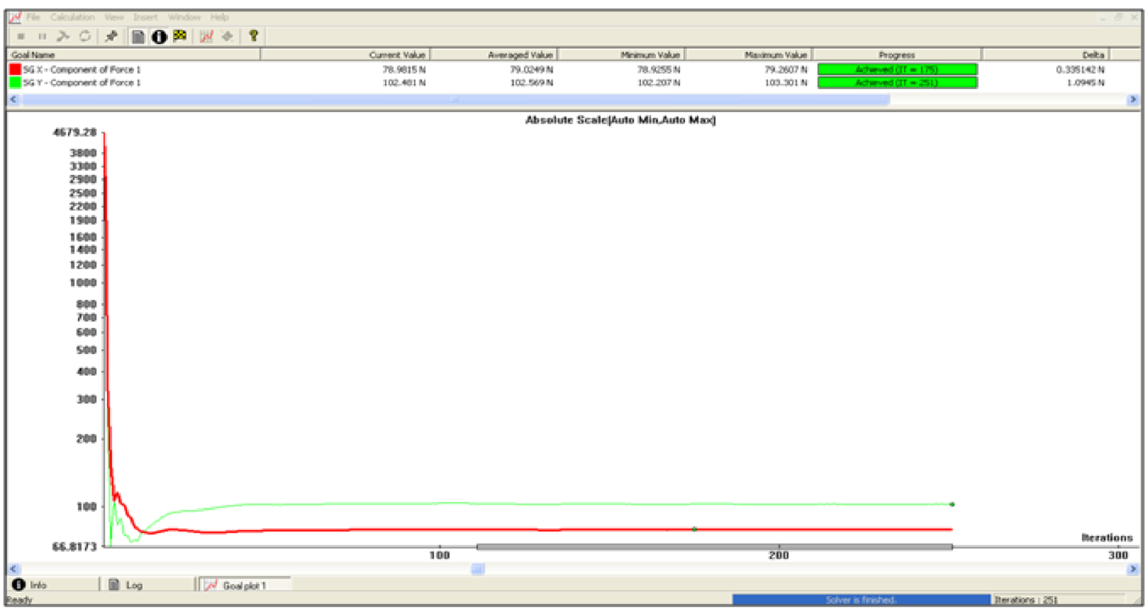


Figure 12, Goal plot for X and Y components.

Figure 12 shows the goal plot for X and Y components in mesh size 3

Mesh size	Fluid cell	Iterations per travel	Travels	Iterations	CPU Time	Drag	Lift
3	93031	93	2.675	251	44:45	78.9815	102.481

Surface parameter for mesh size 3:

Local parameters					
Parameter	Minimum	Maximum	Average	Bulk Average	Surface area [m^2]
Pressure [Pa]	101166	101465	101308		27.1167
Total Pressure [Pa]	101166	101465	101308		27.1167
Temperature [K]	293.185	295.67	293.25		27.1167
Density [kg/m^3]	1.19302	1.20531	1.20356		27.1167
Velocity [m/s]	0	0	0		27.1167
X-component of Velocity [m/s]	0	0	0		27.1167
Y-component of Velocity [m/s]	0	0	0		27.1167
Z-component of Velocity [m/s]	0	0	0		27.1167
Mach Number [ ]	0	0	0		27.1167
Heat Transfer Coefficient [W/m^2/K]	0	0	0		27.3243
Shear Stress [Pa]	2.64583E-09	5.02475	0.395348		27.3243
Fluid Temperature [K]	293.185	295.67	293.25		27.1167
Total Temperature [K]	293.185	295.67	293.25		27.1167
Heat Flux [W/m^2]	0	0	0		27.3243
X-component of Heat Flux [W/m^2]	0	0	0		27.3243
Y-component of Heat Flux [W/m^2]	0	0	0		27.3243
Z-component of Heat Flux [W/m^2]	0	0	0		27.3243

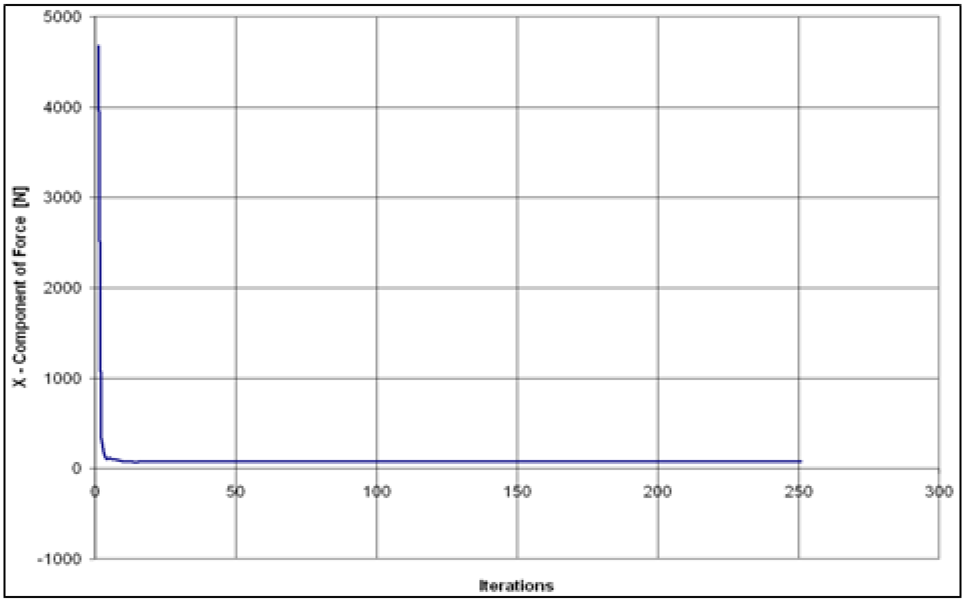
Integral parameters					
Parameter	Value	X-component	Y-component	Z-component	Surface area [m^2]
Heat Transfer Rate [W]	0	0	0	0	27.3243
Normal Force [N]	124.486	68.6964	102.674	15.3507	27.3243
Shear Force [N]	10.2869	10.2851	-0.193694	-0.0307257	27.3243
Force [N]	130.288	78.9815	102.481	15.3199	27.3243
Torque [N*m]	26.6792	-15.3365	20.9615	-6.09837	27.3243
Surface Area [m^2]	27.3243	1.48336E-06	0.052116	0.00130847	27.3243
Torque of Normal Force [N*m]	25.1576	-15.3217	19.1329	-5.6639	27.3243
Torque of Shear Force [N*m]	1.87951	-0.0147913	1.82855	-0.434471	27.3243
Uniformity Index [ ]	1				27.1167
CAD Fluid Area [m^2]	28.1851				28.1851

X and Y component of force for mesh size 3:

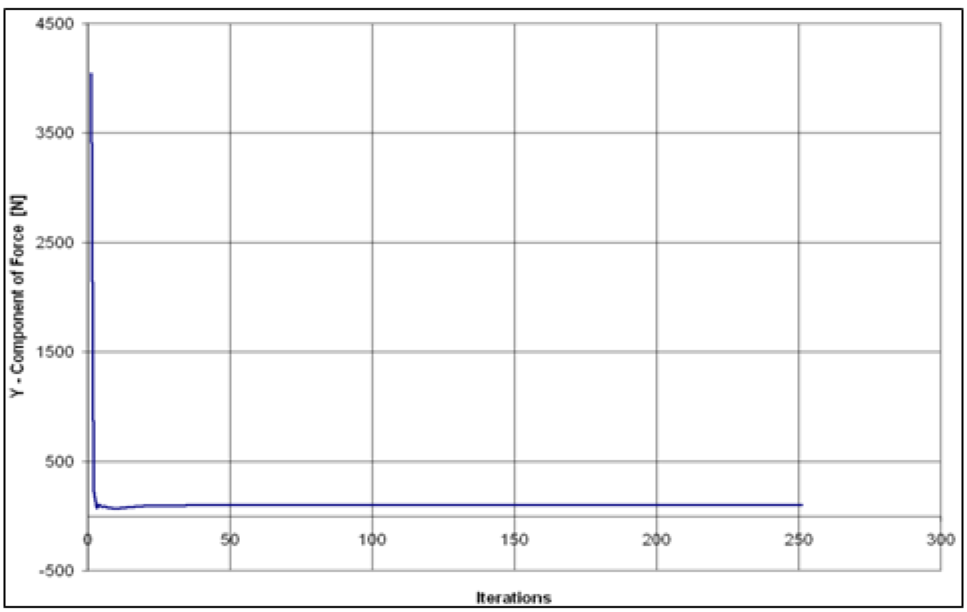
Iterations: 251

Analysis interval: 141

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]	Use In Convergence	Delta	Criteria
SG X - Component of Force	[N]	78.98146299	79.0249172	78.9255133	79.26065496	100	Yes	0.335141656	2.379981082
SG Y - Component of Force	[N]	102.4807156	102.569004	102.2066334	103.3011327	100	Yes	1.094499237	1.178225298

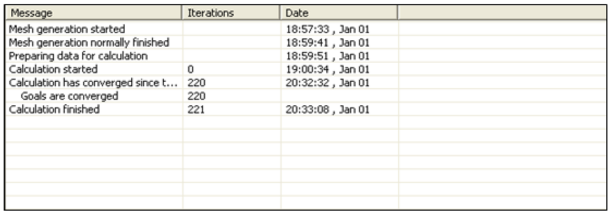


SG X - Component of Force 1

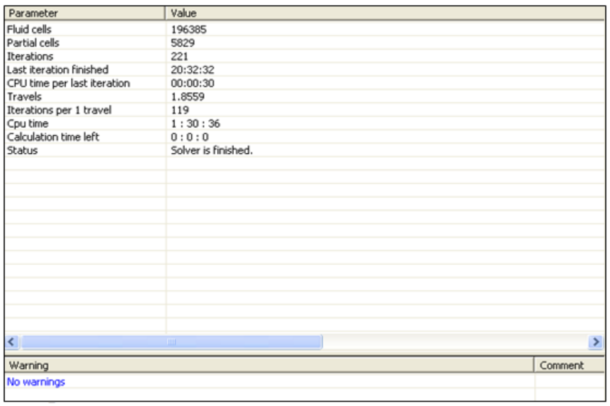


SG Y - Component of Force 1

Mesh size 4:



Message	Iterations	Date
Mesh generation started		18:57:33 , Jan 01
Mesh generation normally finished		18:59:41 , Jan 01
Preparing data for calculation		18:59:51 , Jan 01
Calculation started	0	19:00:34 , Jan 01
Calculation has converged since t...	220	20:32:32 , Jan 01
Goals are converged	220	
Calculation finished	221	20:33:08 , Jan 01



Parameter	Value
Fluid cells	196385
Partial cells	5829
Iterations	221
Last iteration finished	20:32:32
CPU time per last iteration	00:00:30
Travels	1.8559
Iterations per 1 travel	119
Cpu time	1 : 30 : 36
Calculation time left	0 : 0 : 0
Status	Solver is finished.

Warning	Comment
No warnings	

Figure 13

Figure 13 shows the number of Fluid cells, iterations, travels, and iterations per 1 travel for mesh size 4.

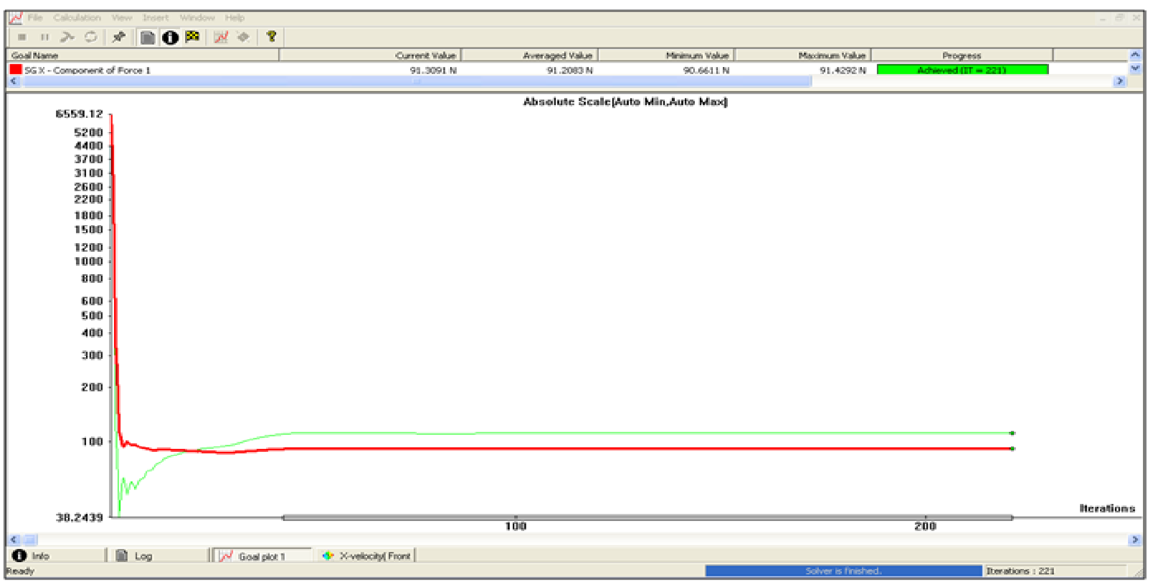


Figure 14, Goal plot for X and Y components

Figure 14 shows the goal plot for X and Y components in mesh size 4.

Mesh size	Fluid cell	Iterations per travel	Travels	Iterations	CPU Time	Drag	Lift
4	196385	119	1.855	221	1:30:36	91.3091	111.4

Surface parameter for mesh size 4:

Local parameters					
Parameter	Minimum	Maximum	Average	Bulk Average	Surface area [m^2]
Pressure [Pa]	100914	102047	101307		27.3797
Total Pressure [Pa]	100914	102047	101307		27.3797
Temperature [K]	62.8039	8925.95	293.337		27.3797
Density [kg/m^3]	0.0593121	2.05873	1.20346		27.3797
Velocity [m/s]	0	0	0		27.3797
X-component of Velocity [m/s]	0	0	0		27.3797
Y-component of Velocity [m/s]	0	0	0		27.3797
Z-component of Velocity [m/s]	0	0	0		27.3797
Mach Number [ ]	0	0	0		27.3797
Heat Transfer Coefficient [W/m^2/K]	0	0	0		27.5307
Shear Stress [Pa]	0	133.185	0.414347		27.5307
Fluid Temperature [K]	62.8039	8925.95	293.337		27.3797
Total Temperature [K]	62.8039	8925.95	293.337		27.3797
Heat Flux [W/m^2]	0	0	0		27.5307
X-component of Heat Flux [W/m^2]	0	0	0		27.5307
Y-component of Heat Flux [W/m^2]	0	0	0		27.5307
Z-component of Heat Flux [W/m^2]	0	0	0		27.5307

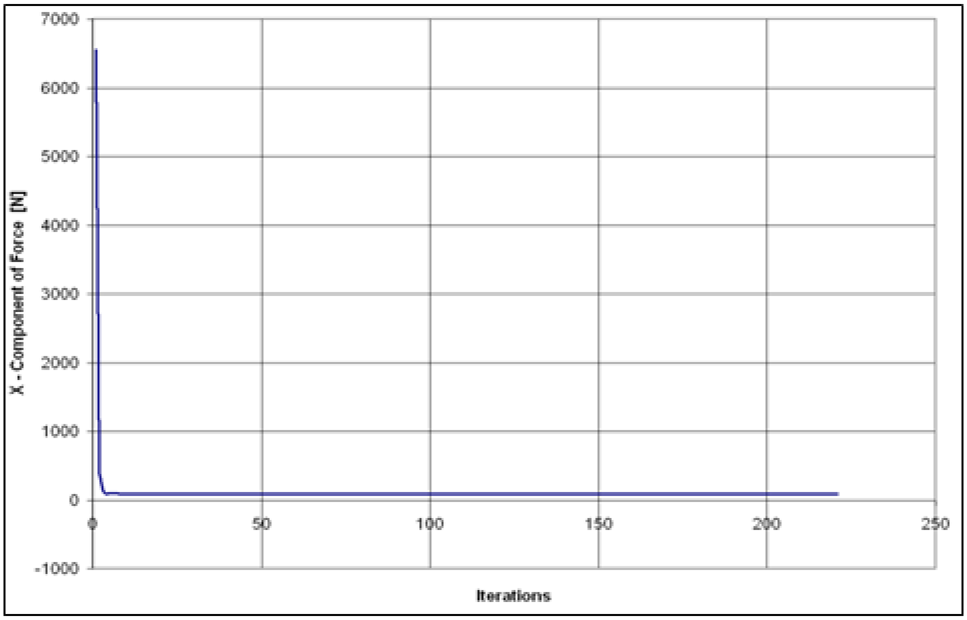
Integral parameters					
Parameter	Value	X-component	Y-component	Z-component	Surface area [m^2]
Heat Transfer Rate [W]	0	0	0	0	27.5307
Normal Force [N]	138.311	80.6182	111.58	13.443	27.5307
Shear Force [N]	10.6924	10.6909	-0.179927	-0.00427737	27.5307
Force [N]	144.665	91.3091	111.4	13.4387	27.5307
Torque [N*m]	27.0708	-17.1616	19.7899	6.8317	27.5307
Surface Area [m^2]	27.5307	0.00100963	0.0429975	-0.00107396	27.5307
Torque of Normal Force [N*m]	25.8756	-17.125	17.9727	7.298	27.5307
Torque of Shear Force [N*m]	1.87638	-0.0365285	1.81715	-0.466301	27.5307
Uniformity Index [ ]	1				27.3797
CAD Fluid Area [m^2]	28.1851				28.1851

X and Y component of force for mesh size 4:

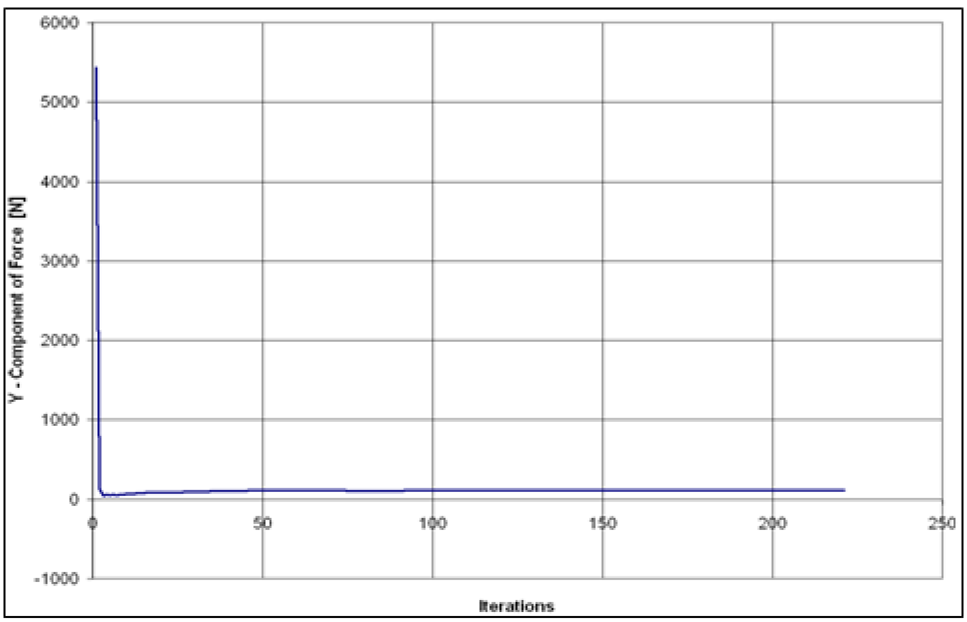
Iterations: 221

Analysis interval: 179

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]	Use In Convergence	Delta	Criteria
SG X - Component of Force	[N]	91.30910316	91.20829954	90.66108872	91.42918189	100	Yes	0.768093173	0.824552416
SG Y - Component of Force	[N]	111.3997808	111.1989847	109.7852931	111.6032635	100	Yes	1.817970389	2.199961303



SG X - Component of Force 1



SG Y - Component of Force 1

X-velocity on the cut plot (Mesh 4)

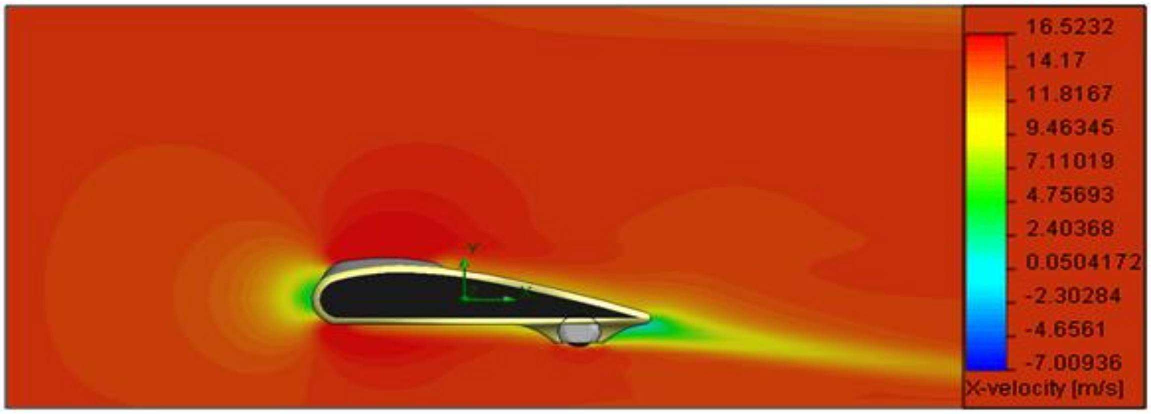


Figure 15, Velocity in X direction

Figure 15 shows the velocity in the X direction on the cut plot over the solar vehicle, the area in red colour has the highest velocity and the lowest velocity is shown in blue colour. As shown on the figure above and the graph on the right-hand side, the X-velocity value is between 9.46m/s and 11.81m/s just before hitting the front of the vehicle, and decreases up to 4.75m/s. As the flow travels over the driver's head (upper surface) x-velocity reaches to the highest speed which is around the 16.52m/s and reduces as its goes further down the vehicle.

The lowest value of the x-velocity can be seen in the light blue colour where the flow passes the back end of the vehicle or the gap between the front and rear wheels with the vehicle body (Appendix 1, figure 3). The value of the x-velocity in blue area can reach up to -7.009m/s, this area has a vacuum and drop in pressure as well.

Figures 1, 2, 3 and 4 in appendix 1 show the different view of the X-velocity traveling over the vehicle body on the cut plot.

Total pressure on the cut plot (Mesh 4)

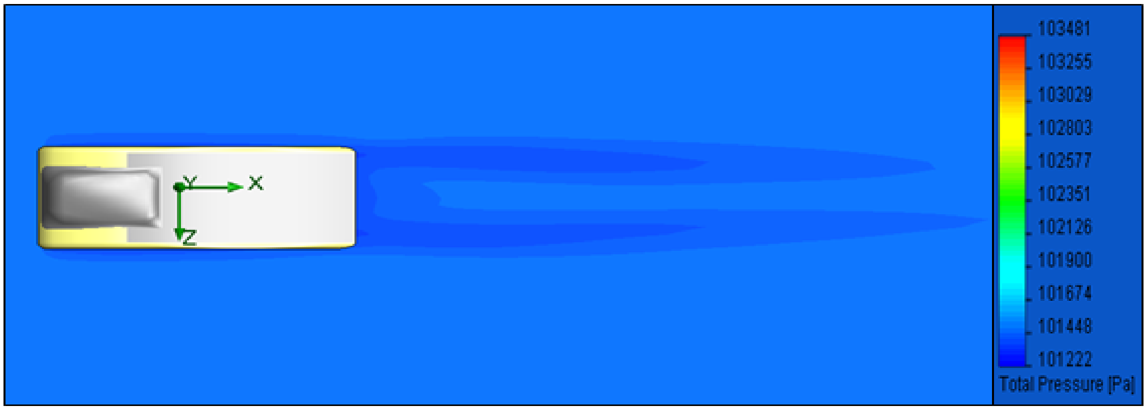


Figure 16, Total pressure on the cut plot

Figure 16 shows the total pressure on the cut plot over the solar vehicle. Total flow pressure on the both side of vehicle (left and right) reach between 101448 Pa to 101222 Pa and remains same as it travels further back of the vehicle.

Also figure 8 (Appendix 1) shows the low value of the total flow pressure on the upper and lower surface of the vehicle drop to 101448 Pa and even goes to 101222 Pa after flow passed the rear wheel of the vehicle.

Pressure on the cut plot (Mesh 4)

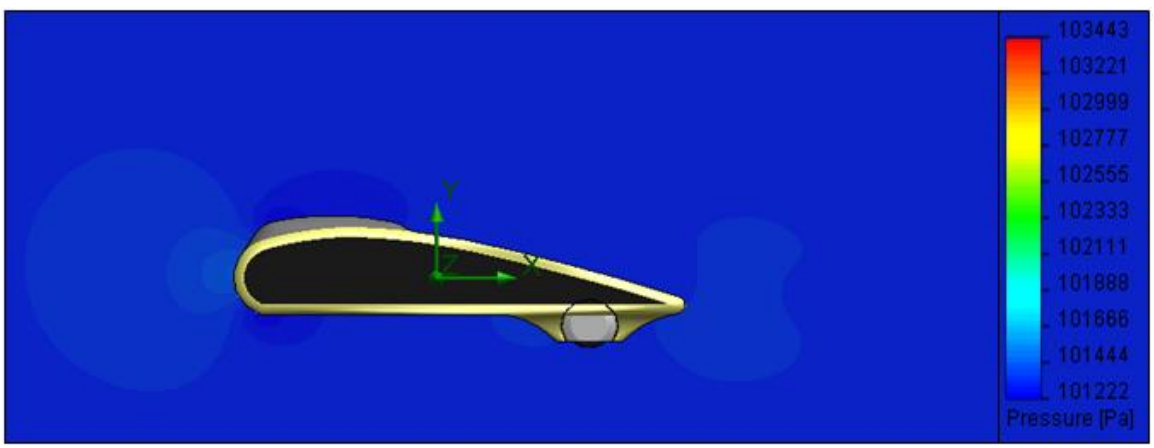


Figure 17, Pressure on the cut plot

Figure 17 above shows the flow pressure over the vehicle on the cut plot, the above right hand side graph indicate the value of flow pressure.

The area in light blue has highest pressure, as it can be seen from Figure 17 the highest flow pressure is in front of the vehicle as it hits the body and decreases to 101222 Pa when traveling over the driver's head (upper surface).

Velocity on the Vector Plot (Mesh 4)

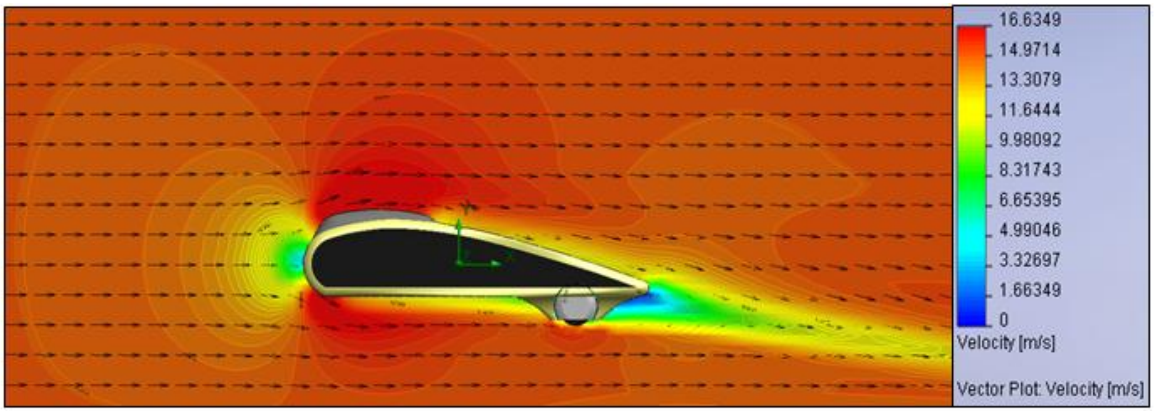


Figure 18, Velocity on the Vector Plot

Figure 18 shows the velocity on the vector plot with Isolines. From the figure above and the right-hand side table it can be seen that the area in red colour has the highest flow velocity with value of 16.63m/s and the area in blue colour has the lowest flow velocity with the value of 0m/s. Black arrows shows the direction of the flow traveling over the vehicle body.

As it is shown in the figure the flow velocity is between 13.30ms/ to 4.99m/s on front of the vehicle and as it travel over the driver cabin roof (upper surface) it reaches to the highest speed of 16.63m/s and rapidly decreases as it goes to the back of the vehicle and increases when passes the body.

The lowest flow velocity is around the back of the vehicle which is 0 and shows in blue colour, this blue area has vacuum and drop in pressure as well.

Figures 9, 10, 11, 12 from (appendix 1) show the different view of Velocity, Total pressure and Pressure on the Vector Plot.

Surface plot (Mesh 4)

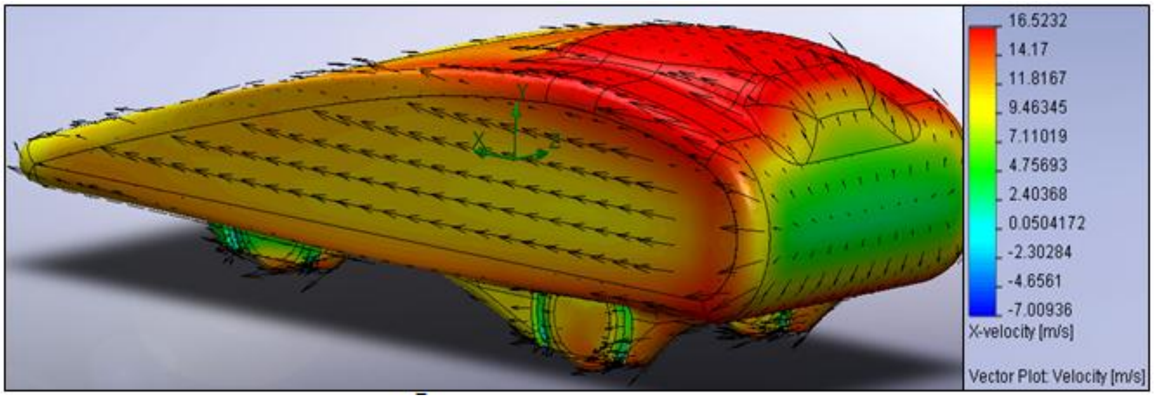


Figure 19 X-velocity on the Vector Plot

Surface plot display the parameter distribution on the solar vehicle body or surface. Figure 19 shows the flow velocity in X direction over the vehicle surface, also the black arrows indicate the direction of flow which hits the front of the vehicle and travels over the surface or body of the vehicle.

Value of the velocity is between 4.75m/s to 7.11m/s when flow hit the front of the vehicle and increases as it goes further over the upper surface, the highest velocity can seen over the driver head which has the value of 16.52m/s.

Flow trajectory (Mesh 4)

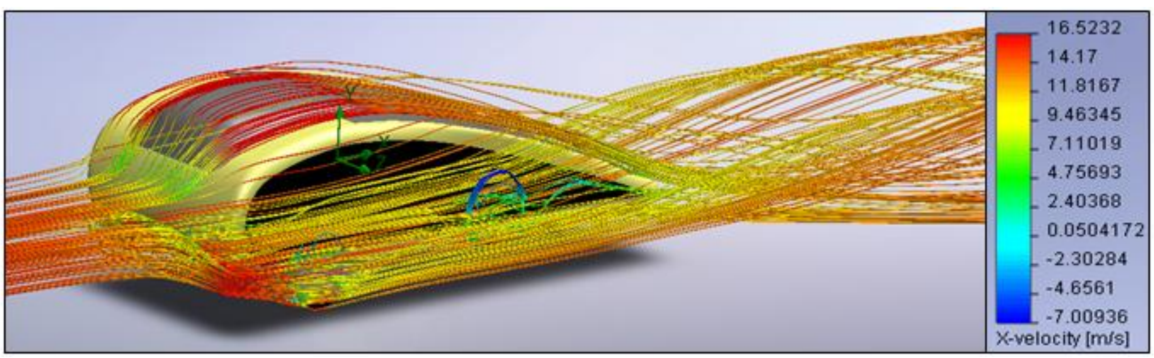


Figure 2,. Flow trajectory

Figure 20 show that how the air separates after hitting the body of vehicle. It shows that how an aerodynamic shape can change the direction of the fluid which is very useful in redesigning process. Above figure shows the flow direction in X-Velocity and separation of flow after hitting front of the vehicle, also it shows how the air travels over the vehicle surface.

Appendix 3 (figure 2 and 3) shows the total pressure, total temperature for the flow trajectories in different views.

Comparison of Drag, Lift, Fluid Cell and V/F ratio of mesh 2, 3 and 4

	Drag	**Lift**	**Fluid Cell**	**Volume**	**V/F**
Mesh 2	68.8153	111.644	18453	1249.5	0.0677
Mesh 3	78.9815	102.481	93031	1249.5	0.0134
Mesh 4	91.3091	111.4	196385	1249.5	0.0063

Where

(From the tables)

Drag = X Component

Lift = Y Component

V/F = Volume of Computational Domain divided by number of fluid cell

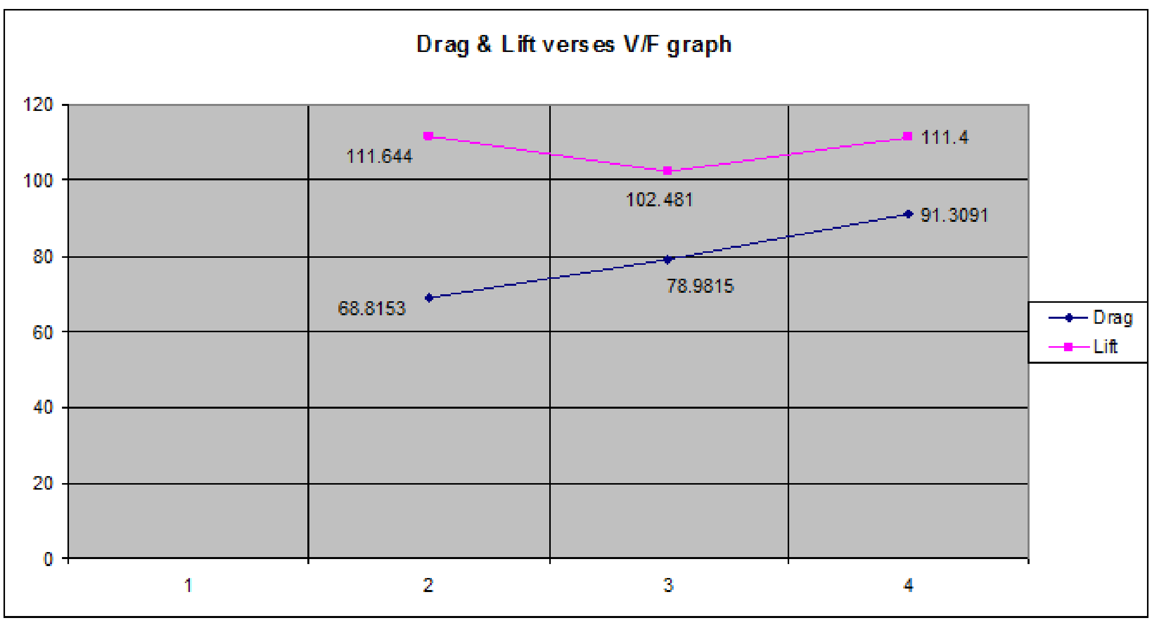


Figure 2,. Drag and Lift vs. V/F

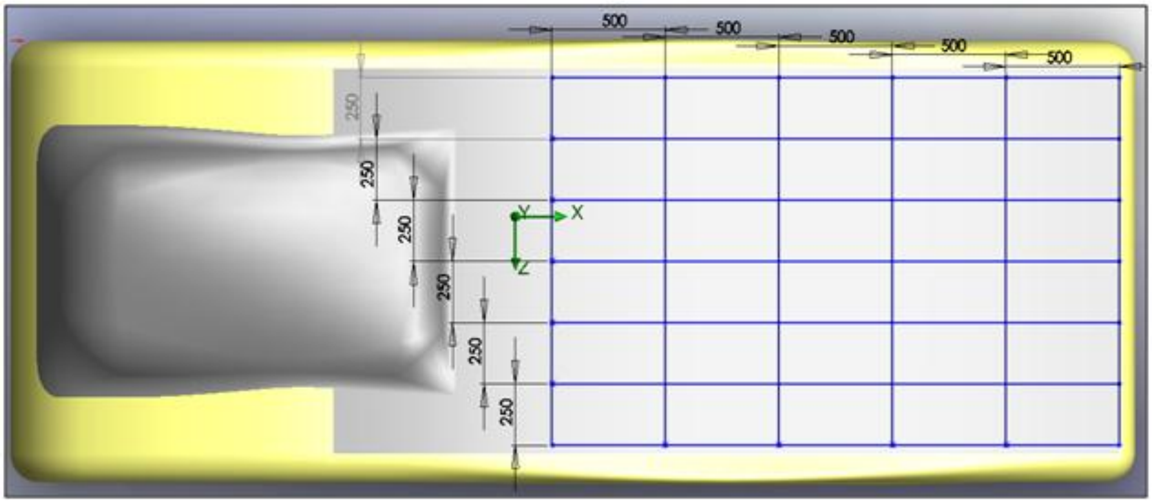


Figure 22, Upper surface of solar car

Figure 22 show the sketch of the solar panel on the upper surface, 30 panels been fitted on the upper surface and all of them follow the straight line (there is no angle and all the panels been fitted on the flat surface of the vehicle).

Each individual panel has dimensions of 0.5metre by 0.25metre and each assumed to develop an effective propulsive power of 35watts after all losses have been taken into account.

Part 3:

Hand Calculations

Mesh size 2

a =1 b = $4n*10^{-6}$

n = number of wheels → n = 3 → b = $12*10^{-6}$

Equations

$F = a+bV^4+cV^2$

$P = F.V$ → $\frac{P}{V} = a+bV^4 + cV^2$

$F = F_{Rolling} + F_{Drag}$

Ignoring $F_{rolling}$:

$F = cV^2$ $P = FV$

$\frac{1050}{V} = cV^2$ $\frac{P}{V} = F$ $\frac{1050}{V} = F$

$1050 = cV^3$

From the result $F_{drag} = 68.81$ N

$68.84=cV^2$

$\frac{68.81}{15^2} = c$ → $c = 0.3058 \approx 0.306$

$F = a+bV^4+cV^2$

$\frac{1050}{V} = 1+12*10^{-6}V^4 + 0.306V^2$

$1050 = V + 12*10^{-6}V^5 + 0.306V^3$

$F(x) = 12*10^{-6}V^5 + 0.306V^3 + V -1050$

$F'(x) = 60*10^{-6}V^4 + 0.918V^2 + 1$

Using Newton-Raphson method:

$$x_{n+1} = x_n - \frac{F(x)}{F'(x)}$$

When $x = 14.9$ $f(x) = -14.05 < 0$

$x = 15.0$ $f(x) = 6.8625 > 0$

Solution is between 14.9 and 15

$x_0 = 14.9$

	F(x)	F'(x)
$x_0 = 14.9$	-14.05483291	207.7624864
$x_1 = 14.9676$	0.05433403	209.6700086
$x_2 = 14.96735$	$-8.56571*10^{-3}$	209.6615231
$x_3 = 14.96734$	$-1.7923*10^{-4}$	209.6626545
$x_4 =$ 14.9734		

The calculation converges to 14.973 ms^{-1}. The max speed with 30 panels with power production of 1050W is 14.973 ms^{-1}.

14.973 x 2.237 = 33.494 → **V = 33.494 mph**

Mesh size 3

Drag = 78.9815

$F = cV^2$ Ignoring rolling resistance

$78.9815 = c*15^2 \rightarrow c = 0.351028$

$P = FV$

$$\frac{1050}{V} = F$$

$$F = a + bV^4 + cV^2$$

Substitute $\frac{1050}{V} = F$ to $F = a + bV^4 + cV^2$

$$\frac{1050}{V} = a + bV^4 + cV^2$$

$$\frac{1050}{V} = 1 + 12*10^{-6}*V^4 + 0.351V^2$$

$$1050 = V + 12*10^{-6}V^5 + 0.351V^3$$

$$F(x) = 12*10^{-6}V^5 + 0.351V^3 + V - 1050$$

$$F'(x) = 60*10^{-6}V^4 + 1.053V^2 + 1$$

Using Newton-Raphson method:

$$x_{n+1} = x_n - \frac{F(x)}{F'(x)}$$

When $x = 14.3$	$F(x) = -2.12 < 0$
$x = 14.4$	$F(x) = 19.91 > 0$

Solution lies between 14.3 and 14.4

$x_0 = 14.3$

	F(x)	F'(x)
$x_0 = 14.3$	19.91046771	218.8396396
$x_1 = 14.209$	-21.91453405	216.0418641
$x_2 = 14.310$	0.06222072	219.1452284
$x_3 = 14.3097$	$-3.52145*10^{-3}$	219.1359764
$x_4 = 14.3097$		

Answer converges 14.3097

Max velocity is $\underline{14.3097\ ms^{-1}}$

14.3097 x 2.237 = 32.01 → **V = 32.01 mph**

Mesh size 4

Drag is 91.3091

$F = cV^2$ when ignoring rolling resistance.

$91.3091 = c*15^2$

c = 0.4058

$F(x) = 12*10^{-6}V^5 + 0.406V^3 + V - 1050$

$F'(x) = 60*10^{-6}V^4 + 1.218V^2 + 1$

Using Newton-Raphson method:

$$x_{n+1} = x_n - \frac{F(x)}{F'(x)}$$

When x = 13.6 F(x) = -9.54 < 0

x = 13.7 F(x) = 13.4 > 0

Solution lies between 13.6 and 13.7

$x_0 = 13.6$

	F(x)	F'(x)
$x_0 = 13.6$	-9.5417591	228.3338921
$x_1 = 13.6417$	$9.12451*10^{-3}$	229.7428051
$x_2 = 13.6416$	-0.0138496	229.7394211
$x_3 = 13.64166$	$-6.518*10^{-5}$	229.7414515
$x_4 = 13.6416$		

Answer is $\underline{13.6416\ ms^{-1}}$.

13.6416 x 2.237 = 30.516 $\rightarrow$ **V = 30.516 mph**

Part 4:

Discussion

From the CFD results and hand calculation can see that the bodyshell vehicle has average Max Velocity of 14.30m/s. also from the lift to drag graph (figure 21) can see that the drag increases rapidly when the mesh setting increase.

The aerofoil section (SELIG 3021) been selected for this particular design to improve the aerodynamic of the vehicle by creating less sharp edges (avoid sharp edges) and reduce the drag force to achieve the high performance.

By looking at the CFD result (drag forces) the value of the Max Velocity from three different mesh setting can tell that the vehicle has low speed and high drag.

There are some area can be highlighting it for the week performance of the vehicle:

1.) Number of the solar panel: in order to produce more power the number of the panels on the upper surface needs to be increase up to 48, to do that the vehicle need to be redesign in order to fit 48 panels on the upper surface.

2.) Driver position: driver seat position need to be change to the higher angle of seat which keep the driver in the lower position, by putting the driver in the lower position the cabin height will be reduce also the maximum height of the vehicle would be decrease, by doing that the flow mass fraction (fraction between the air and the vehicle surface) would be reduce as it goes over the driver cabin and the upper surface of the vehicle, also temperature will decrease too.

3.) Position of the driver cabin : driver cabin need to move from front of the vehicle to back of the vehicle, by designing the vehicle in that way the angle of attack will be change which would affect the lift and drag forces (lest drag), by decreasing the drag force vehicle will achieve the better performance with higher speed.

Appendix 1

Cut Plot, Vector Plot and Isolines on the Cut Plot for the following parameters

1. ***X-Velocity***
2. ***Velocity***
3. ***Total Pressure***
4. ***Pressure***
5. ***Temperature***

<u>X-Velocity on the Cut Plot</u>

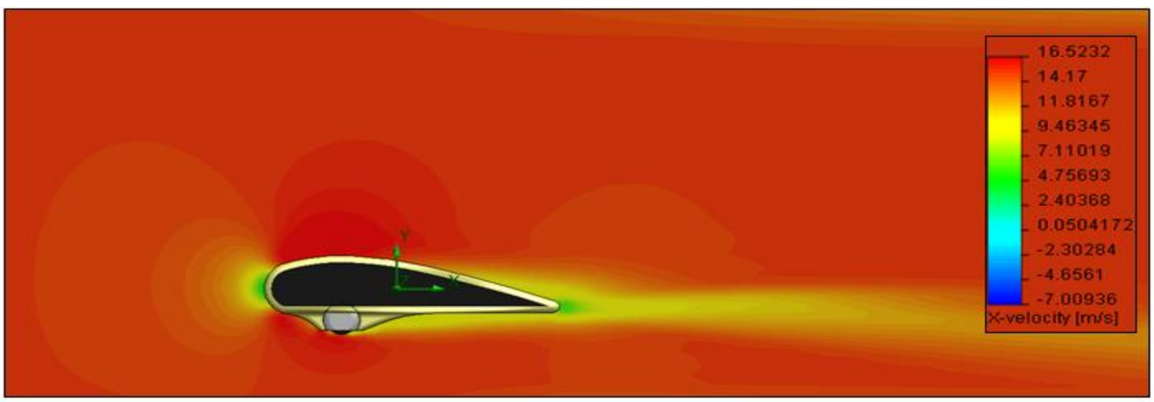
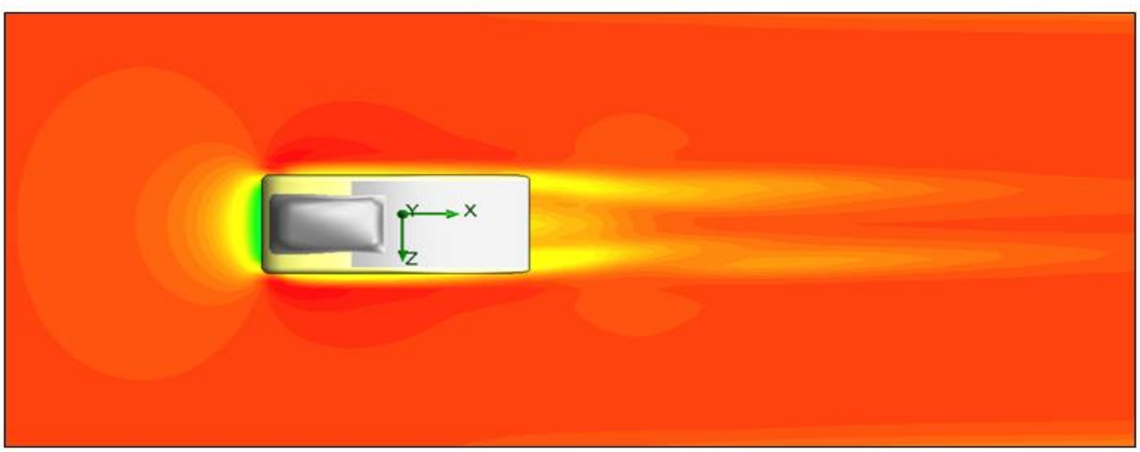
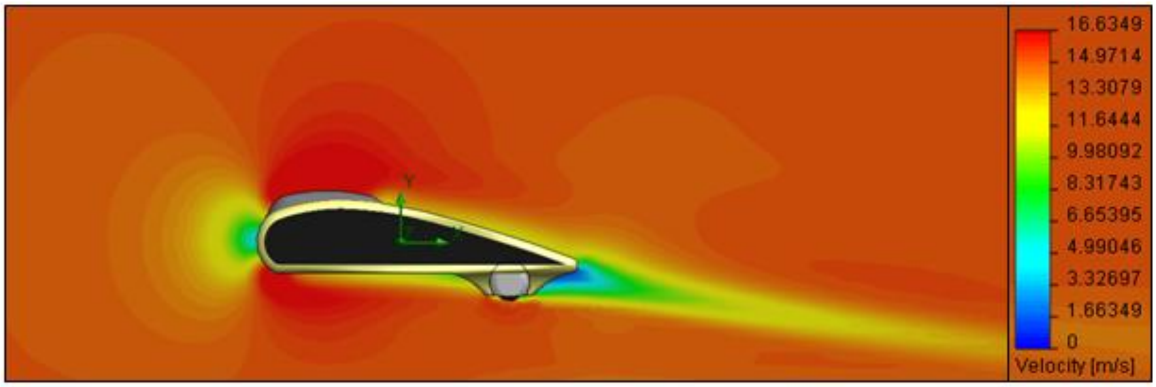
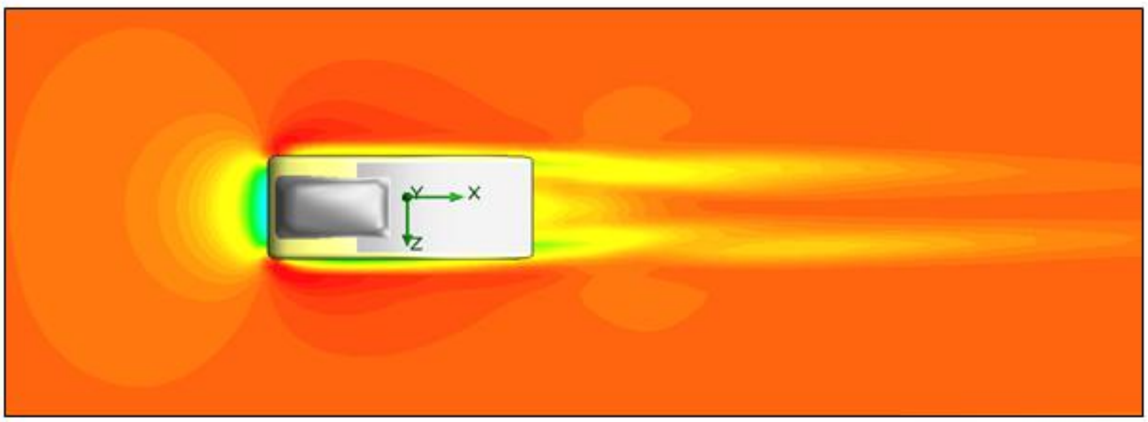


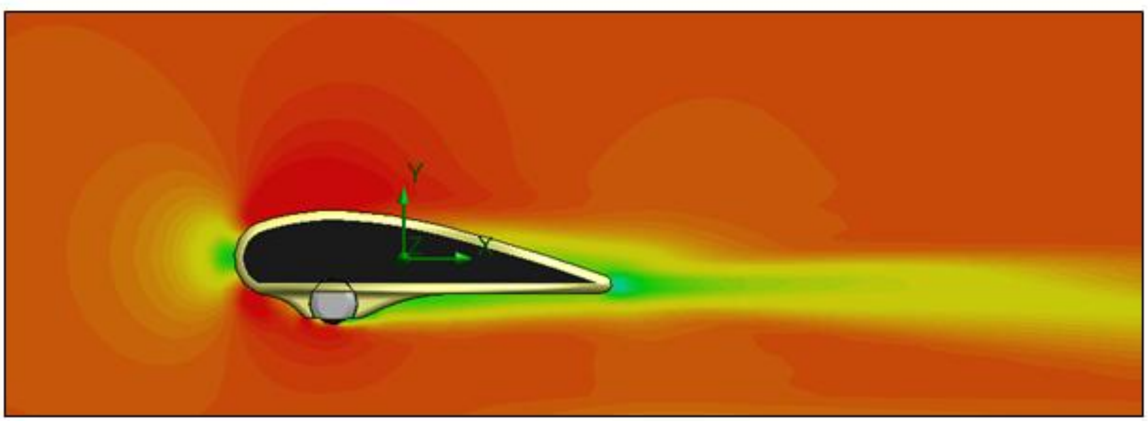
Figure 1



Velocity on the Cut Plot

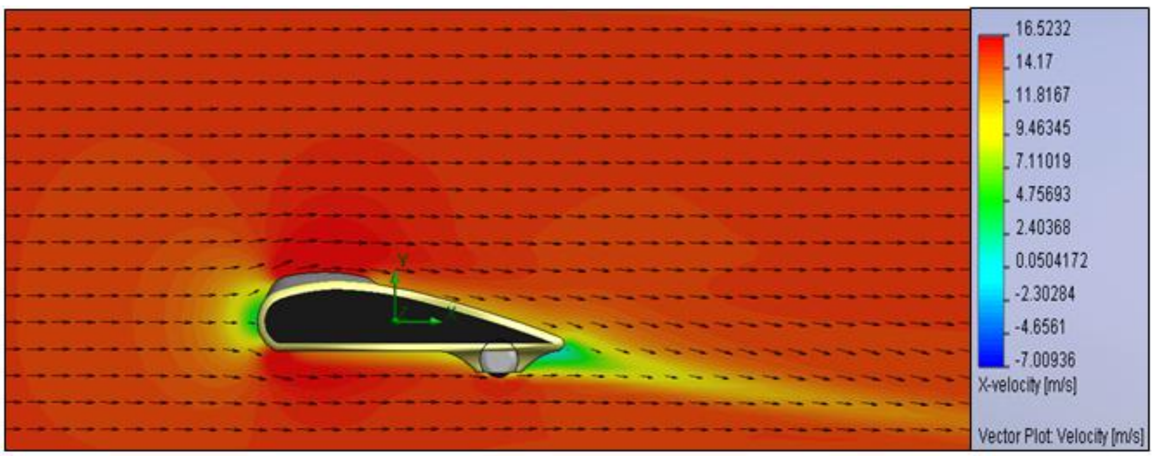


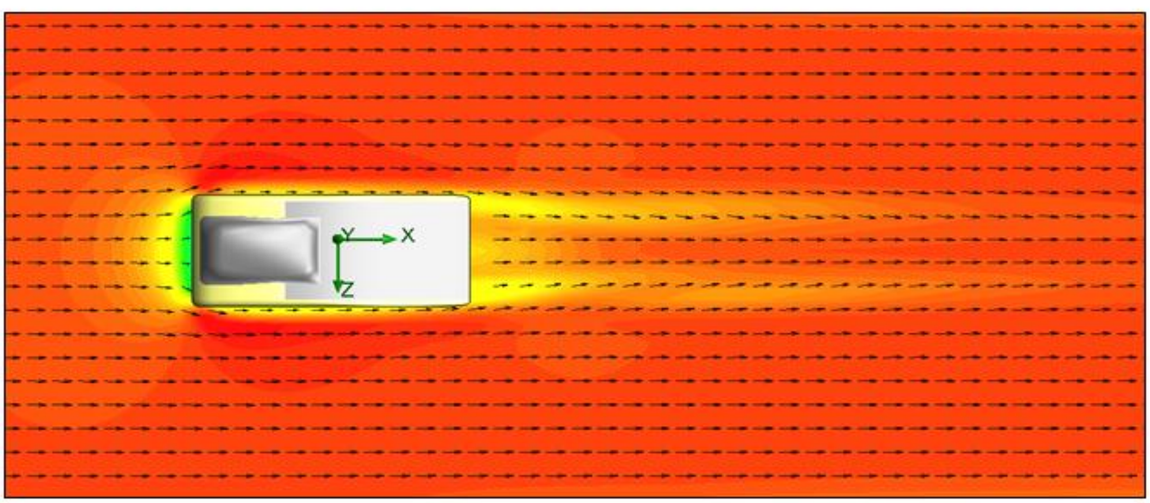




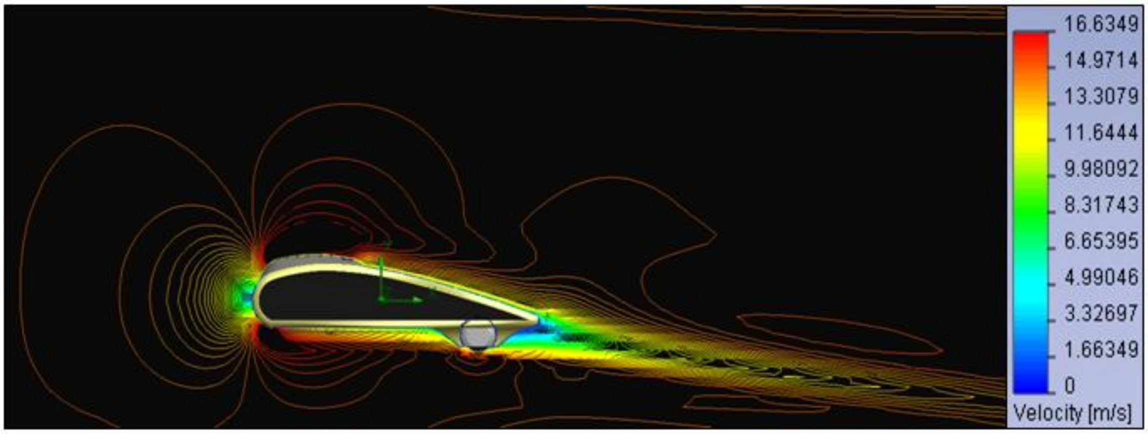
Total Pressure on the Cut Plot

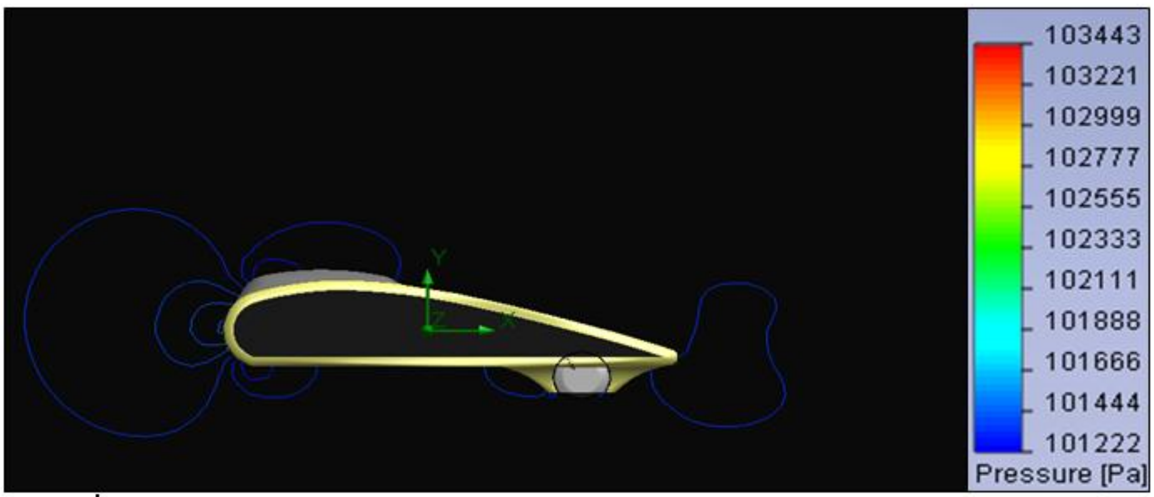
Vector Plot





Isoline Plot



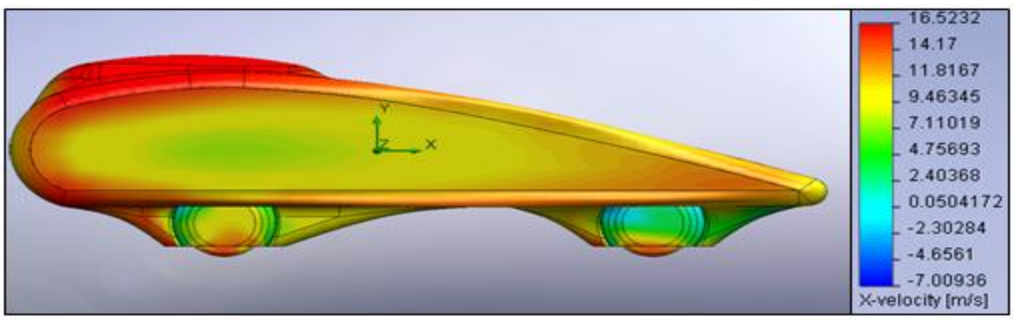


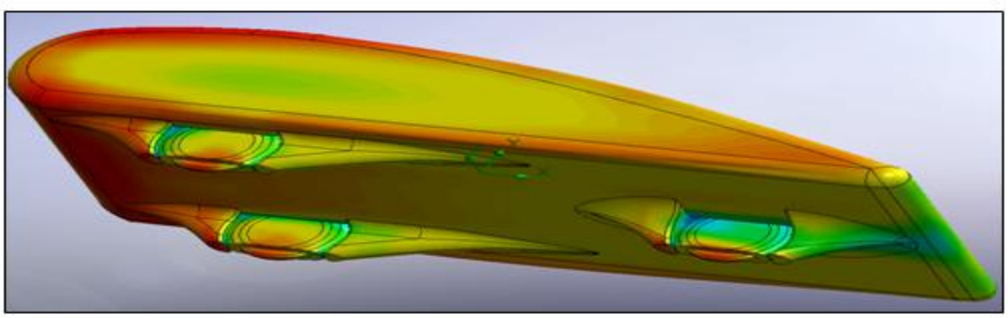
Appendix 2

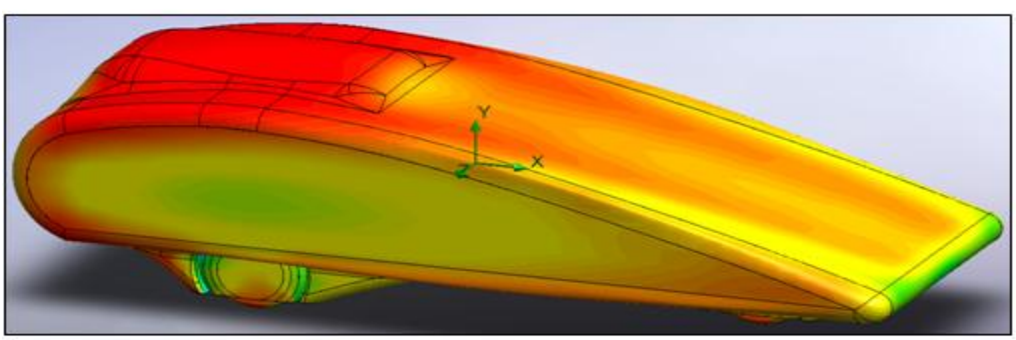
Vector Plot and Isolines on the Surface Plot for the following parameters

1. *X-Velocity*
2. *Velocity*
3. *Total Pressure*
4. *Pressure*
5. *Temperature*

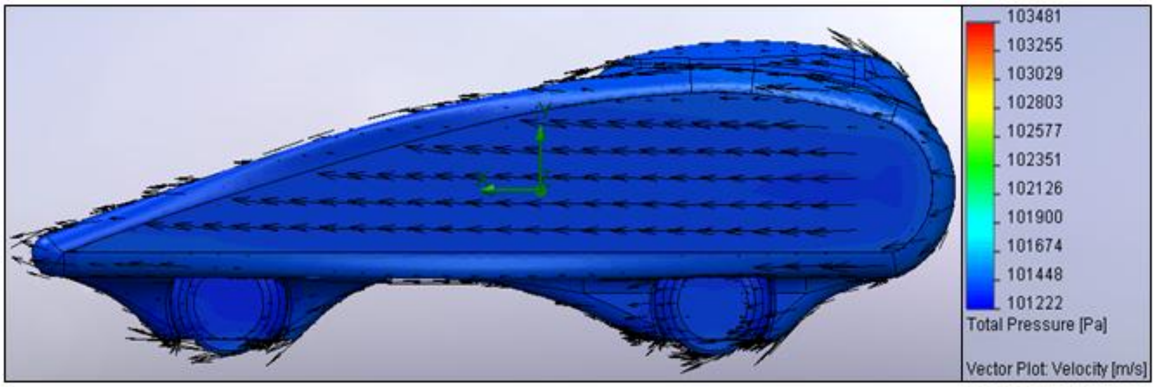
X-Velocity on the Surface Plot

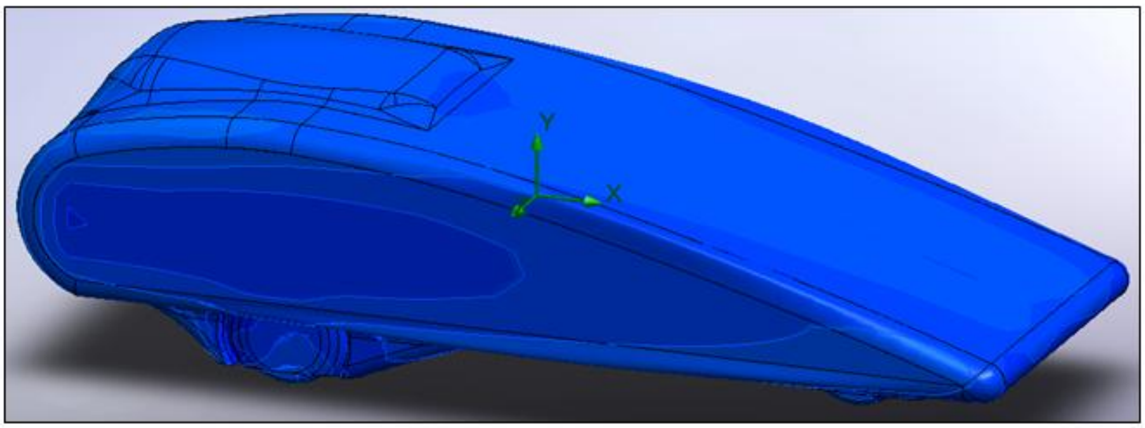




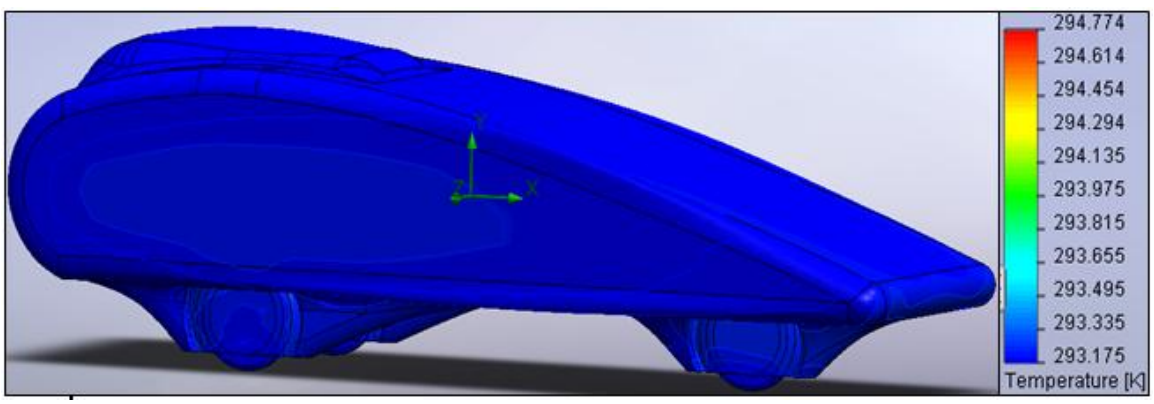


Total pressure on the Surface Plot with Vector Plot and Isolines

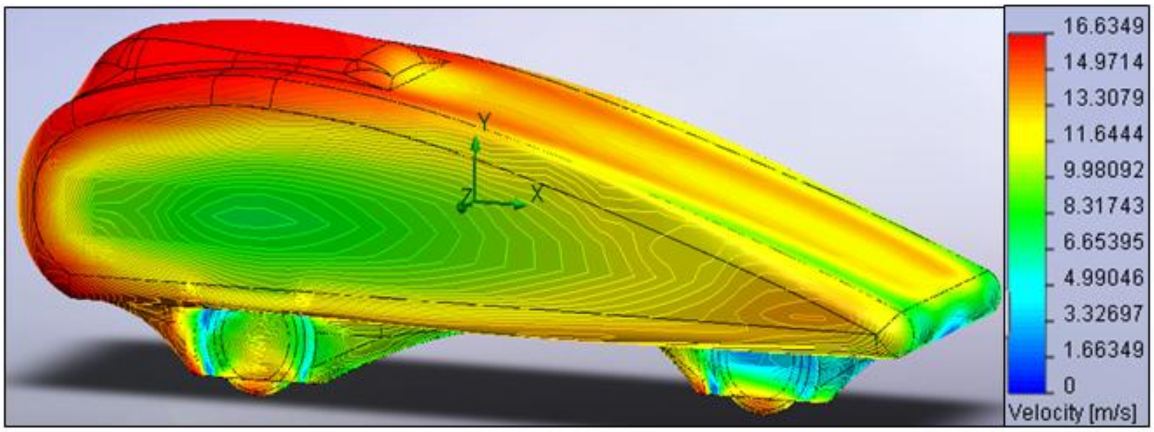


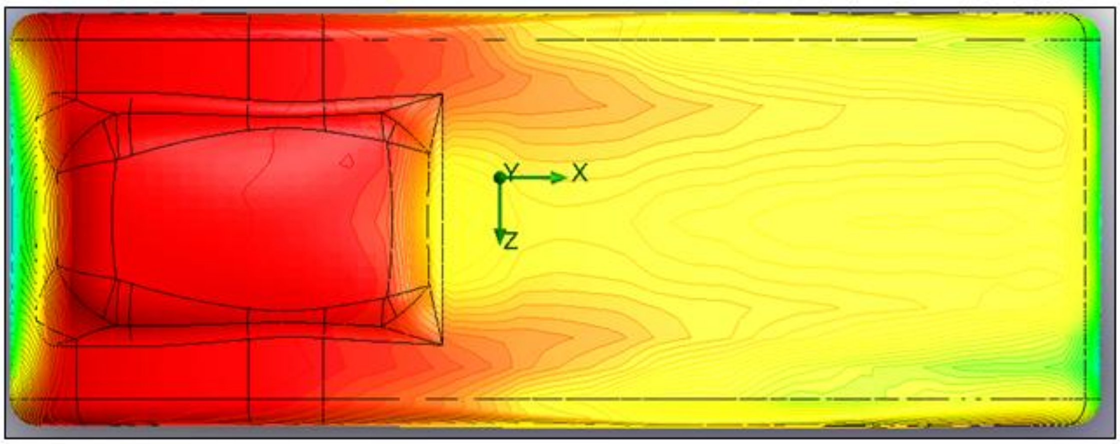


Pressure on the Surface Plot with Isolines



Velocity on the surface plot with Isolines





Appendix 3

Flow trajectories