

Harjot Ubhi CFD Report

by Harjot Singh Ubhi

FILE	7312_HARJOT_SINGH_UBHI_HARJOT_UBHI_CFD_REPORT_128450_154652543.DOCX (92)		
TIME SUBMITTED	28-JAN-2019 03:04AM (UTC+0000)	WORD COUNT	2028
SUBMISSION ID	99893879	CHARACTER COUNT	11758



Prifysgol Cymru
Y Drindod Dewi Sant
University of Wales
Trinity Saint David

ADVANCED COMPUTATIONAL METHODS

AERODYNAMIC VEHICLE TESTING AND ANALYSIS

Harjot Singh Ubhi P147965

M6X00408

January 28, 2019

Tim Tudor

Contents

1) Introduction	- 1 -
2) Domain Geometry	- 1 -
3) Mesh Quality	- 4 -
Surface Mesh Size	- 4 -
Prism Layers	- 5 -
Mesh Method	- 6 -
4) Turbulence Models	- 6 -
5) Vehicle Geometry	- 10 -
6) Conclusion	- 19 -
7) References	- 20 -

1) Introduction

This report will cover the investigative process of analysing the aerodynamics of the MIRA reference car geometry and introduction of a change in geometry that will affect the aerodynamics of the vehicle. A study was done into finding the most suitable settings for domain size and mesh quality, then the simulations for the alternate geometry were run using the most appropriate settings.

The initial setup was to create a domain around the supplied MIRA reference car geometry, as the car is symmetrical the domain and car were halved in order to decrease the time taken to compute. The mesh quality settings used before investigation began were based on the tutorial video, the only difference being the number of prism layers used was 3, because after a few test runs it was found that the simulations were not converging with 5 prism layers. (Tudor, 2019)

2) Domain Geometry

The domain size was increased in increments and the coefficients of drag and lift were recorded once the simulations had reached convergence. The height and width of the domain were kept constant throughout at 4.4m and 8m respectively. In table 1 below H3 is the length of the domain in front car measured from the back of the car, and H4 is the length of the domain behind the car measured from the back of the car. A general trend can be seen whereby as the size of the domain is increased the difference in coefficient of lift calculated and the MIRA reference values decreased. However as seen in table 1 the coefficient of lift that was achieved for each of the runs were off by large percentages, lift predictions in wind tunnel simulations are often inaccurate. (Palaskar, et al., 2016)

The last entry in table 1 is the recommended domain length proposed by Lanfrit, 3 car lengths in front of the vehicle and 5 car lengths behind the vehicle. (Lanfrit, 2005)

Domain Size Change							
				MIRA Ref			
H3(m)	H4(m)	CD	CL	CD	CL	%Diff CD	%Diff CL
6	2	0.3713	-0.3173	0.2963	-0.0845	25.32	275.49
8	4	0.3019	-0.2356	0.2963	-0.0845	1.87	178.76
10	6	0.3197	-0.1262	0.2963	-0.0845	7.90	49.39
12	8	0.2856	-0.2192	0.2963	-0.0845	3.60	159.41
14	10	0.3273	-0.1664	0.2963	-0.0845	10.45	96.98
16	12	0.3327	-0.1949	0.2963	-0.0845	12.28	130.61
18	14	0.2843	-0.2197	0.2963	-0.0845	4.05	159.98
20	16	0.2850	-0.1863	0.2963	-0.0845	3.81	120.50
22	18	0.2851	-0.2102	0.2963	-0.0845	3.77	148.72
24	20	0.2836	-0.1953	0.2963	-0.0845	4.30	131.09
26	22	0.2981	-0.1764	0.2963	-0.0845	0.60	108.75
16	20	0.2977	-0.1877	0.2963	-0.0845	0.47	122.18

Table 1: Change in domain size

Using the results tool in Ansys, analysis of the changes in domain size were done. Figure 1 below shows 3 different domain sizes with contour plots of pressure on the vehicle and velocity of the air on the symmetry plane. The top left domain is H3 at 6m and H4 at 2m, this shows that the domain is too small in both directions because it doesn't fully capture the velocity contour. The domain in the top right is H3 22m and H4 at 18m, the H3 size is greater than 3 car lengths in front of the domain but less than the 5 car lengths behind. The results show that there aren't any uncaptured contours in the front of the domain and in the back, however, the domain size suggested by Lanfrit yielded the lowest difference in coefficient of drag and also captured all contours, as shown in the bottom of figure 1.

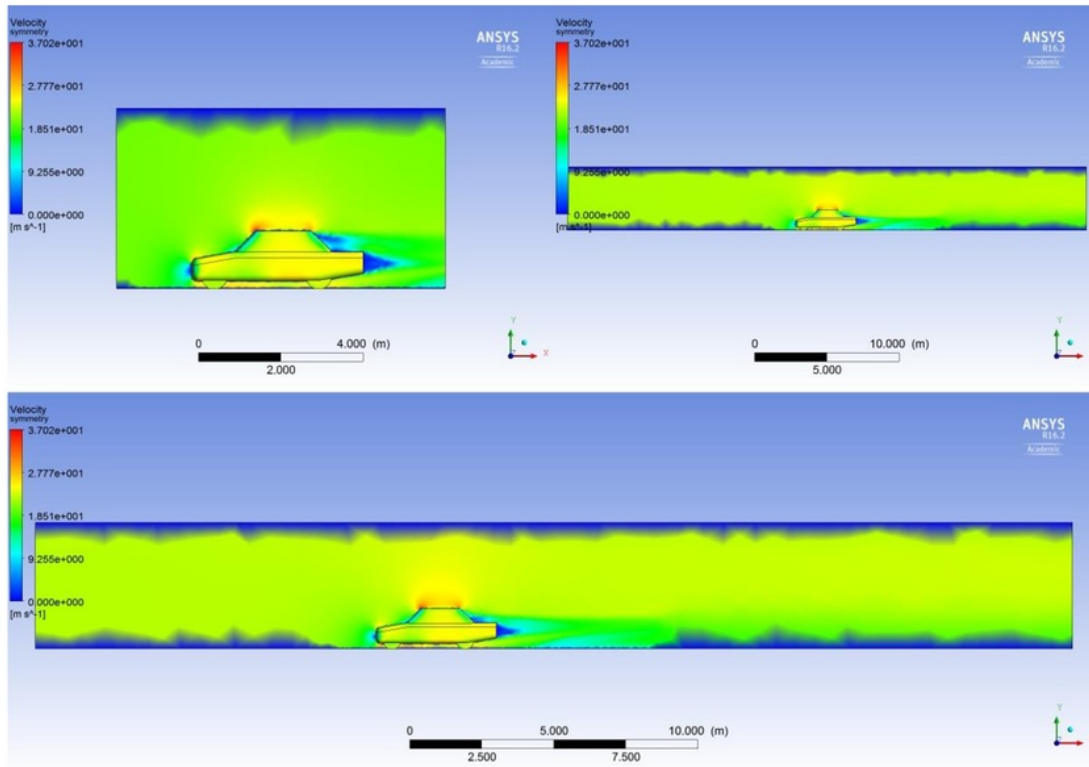


Figure 1: Change in domain size

Based on the data (table 1) and the analysis of the visual results (figure 1), the domain size of H3 at 16m and H4 at 20m was carried forward for the other investigations.

3) Mesh Quality

There are three different changes that were done to change the mesh quality, changes in surface mesh size, changes in number of prism layers and also changes to the mesh method.

Surface Mesh Size

The surface mesh size for the car was decreased and the coefficients of drag and lift were recorded. The first entry in table 2 below are the settings used during the study of domain size.

From table 2 a correlation can be seen between decreasing the surface mesh size and the difference in coefficients of drag and lift from the MIRA reference values. This shows that as the mesh quality gets finer, a more accurate solution can be achieved. There are two outliers than can be seen in table 2, at a mesh size of 0.06 and 0.05, this could be due to the mesh size being too large and the simulation would calculate and average between nodes, and those averages happened to give a value closer to the reference value. As mentioned earlier, the values achieved for the coefficient of lift can be inaccurate, so these two values were considered anomalies.

Change in Mesh Size								
Mesh Size of MIRA Parts			MIRA Ref					
Top	Bottom	Wheels	CD	CL	CD	CL	%Diff CD	%Diff CL
0.05	0.05	0.02	0.2977	-0.1877	0.2963	-0.0845	0.47	-122.18
0.06	0.06	0.06	0.3324	-0.0887	0.2963	-0.0845	12.19	4.93
0.05	0.05	0.05	0.3203	-0.0895	0.2963	-0.0845	8.09	5.91
0.04	0.04	0.04	0.3220	-0.1399	0.2963	-0.0845	8.66	65.56
0.03	0.03	0.03	0.2945	-0.1182	0.2963	-0.0845	0.61	39.85
0.02	0.02	0.02	0.2975	-0.1090	0.2963	-0.0845	0.39	28.95
0.01	0.01	0.01	0.2954	-0.0644	0.2963	-0.0845	0.29	23.79

Table 2: Change in mesh size

The surface mesh size used for the next part of the investigation was 0.02m. Although 0.01m yielded a closer coefficient of drag to the true value, the time to compute was longer so a compromise was made.

Prism Layers

The use of prism layers replaces the need for a fine mesh at boundaries, as a fine mesh would be computationally expensive. Turbulence models that incorporate wall functions work hand in hand with prism layers. (Kapoor, 2016)

Using the chosen surface mesh size from the previous investigation, 0.02m, the number of prism layers was changed. The greyed-out row in table 3 is the settings carried forward from the previous investigation.

A general trend can be drawn from table 3, whereby as the number of prism layers increases the difference from the true values of the coefficients of drag and lift decreases.

Change in number of prism layers						
			MIRA Ref			
Prism Layers	CD	CL	CD	CL	%Diff CD	%Diff CL
0	0.2761	-0.1977	0.2963	-0.0845	6.81	133.94
1	0.3079	-0.1358	0.2963	-0.0845	3.92	60.76
2	0.2992	-0.1413	0.2963	-0.0845	0.97	67.19
3	0.2975	-0.1090	0.2963	-0.0845	0.39	28.95
4	0.2943	-0.1037	0.2963	-0.0845	0.69	22.75
5	0.2910	-0.0976	0.2963	-0.0845	1.78	15.50
6	0.2942	-0.0768	0.2963	-0.0845	0.72	9.07
7	0.3055	-0.0896	0.2963	-0.0845	3.12	5.99

Table 3: Change in prism layers

The number of prism layers carried forward to the next investigation was 6, the reason being that the difference in the coefficient of drag remained low, while the difference in the coefficient of lift was brought down.

Mesh Method

Different mesh methods produce different shape and size cells in the mesh, Octree being more structured than Delaunay, which is more organic and non-structured. The polyhedral mesh reduced the mesh size by around a fifth by creating larger cells, this might be useful for obtaining a quick result but not for accurate ones. Patch independent and patch dependent mesh are similar, but patch dependent meshes are more suitable for situations where high mesh quality is required and time is not limited.

Table 4 shows the changes in the difference in coefficients of drag and lift as the mesh method is changed. The Octree patch independent method was used due to the fact in maintained a low difference in coefficients. Although the Octree patch dependent mesh yielded similar results, it is computationally more expensive and takes more time. The reason the Delaunay polyhedral mesh was dismissed as being the best choice was because the polyhedral mesh is not a suitable mesh due to the cell size being so large.

Change in Mesh Method								
					MIRA Ref			
Mesh Type	Mesh Method		CD	CL	CD	CL	%Diff CD	%Diff CL
Tetra/Mixed	Octree	All Tri/Patch Independent	0.2942	-0.0768	0.2963	-0.0845	0.72	9.07
Tetra/Mixed	Octree Polyhedral	All Tri/Patch Independent	0.2944	-0.0636	0.2963	-0.0845	0.65	24.68
Tetra/Mixed	Delaunay	All Tri/Patch Independent	0.2925	-0.0979	0.2963	-0.0845	1.28	15.90
Tetra/Mixed	Delaunay Polyhedral	All Tri/Patch Independent	0.3043	-0.0799	0.2963	-0.0845	2.70	5.46
Tetra/Mixed	Octree	All Tri/Patch Dependent	0.2942	-0.0765	0.2963	-0.0845	0.72	9.52

Table 4: Change in mesh method

4) Turbulence Models

The turbulence models investigated in this report are based off the Navier-stokes equations. The k-epsilon model solves for two variables k and epsilon, the turbulence kinetic energy and the rate of dissipation of k, respectively. This model is more appropriate for simple geometries, such as the MIRA car used in this report. The k-omega model is similar to k-epsilon, where omega is the specific rate of dissipation of kinetic energy. The k-omega model usually used where for flows with high curvature. (Frei, 2017)

Non-equilibrium wall functions are based on how the calculations are made at the boundaries, i.e the surfaces of the car.

Table 5 shows how the difference in the coefficients of drag and lift change as the turbulence model changes. The greyed-out row is the settings used in the previous simulations.

The 3 figures below, figures 2, 3 and 4, show how the turbulence models differ with regards to how the flow is calculated at the boundaries. Figure 2 shows the velocity contour, figure 2 shows the vectors of velocity and figure 3 shows the vortices created by each of the models.

Change in Turbulence Model					MIRA Ref			
Model			CD	CL	CD	CL	%Diff CD	%Diff CL
k-epsilon	Realizable	Non-Equilibrium Wall Functions	0.2942	-0.0768	0.2963	-0.0845	0.72	9.07
k-epsilon	Standard	Non-Equilibrium Wall Functions	0.3272	-0.1204	0.2963	-0.0845	10.42	42.51
k-epsilon	RNG	Non-Equilibrium Wall Functions	0.2887	-0.0421	0.2963	-0.0845	2.57	50.21
k-omega	Standard	-	0.3417	-0.0983	0.2963	-0.0845	15.31	16.39

Table 5: Change in turbulence model

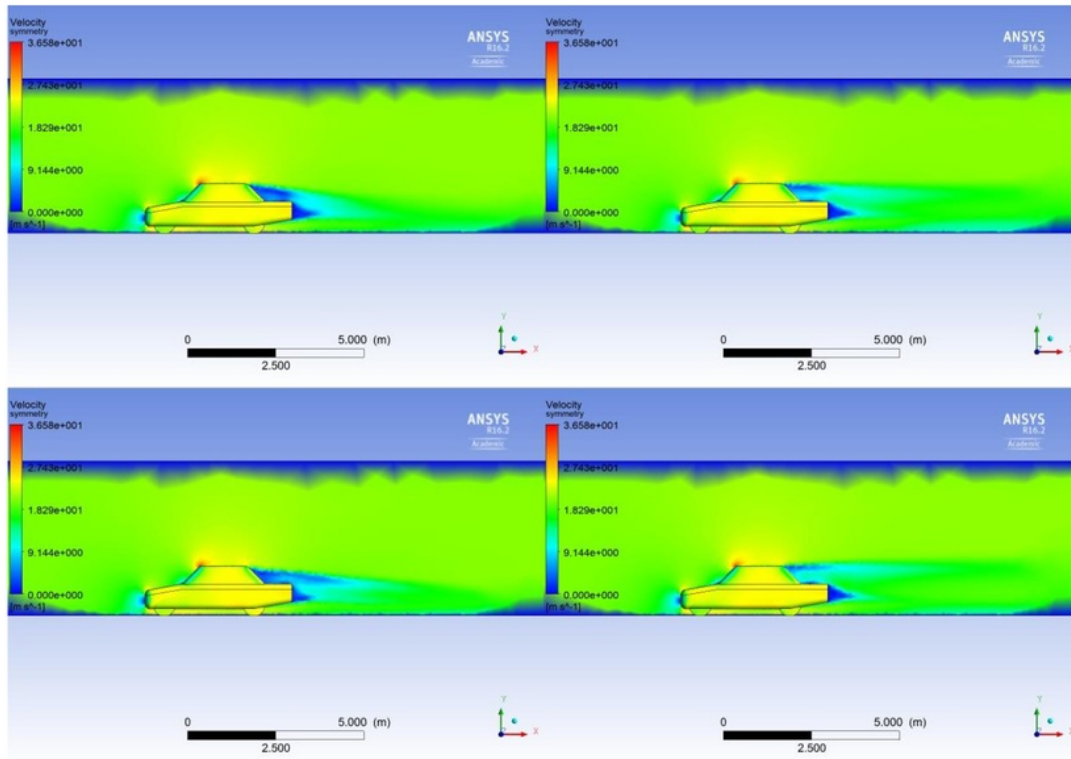


Figure 2: Velocity contours for change in turbulence model

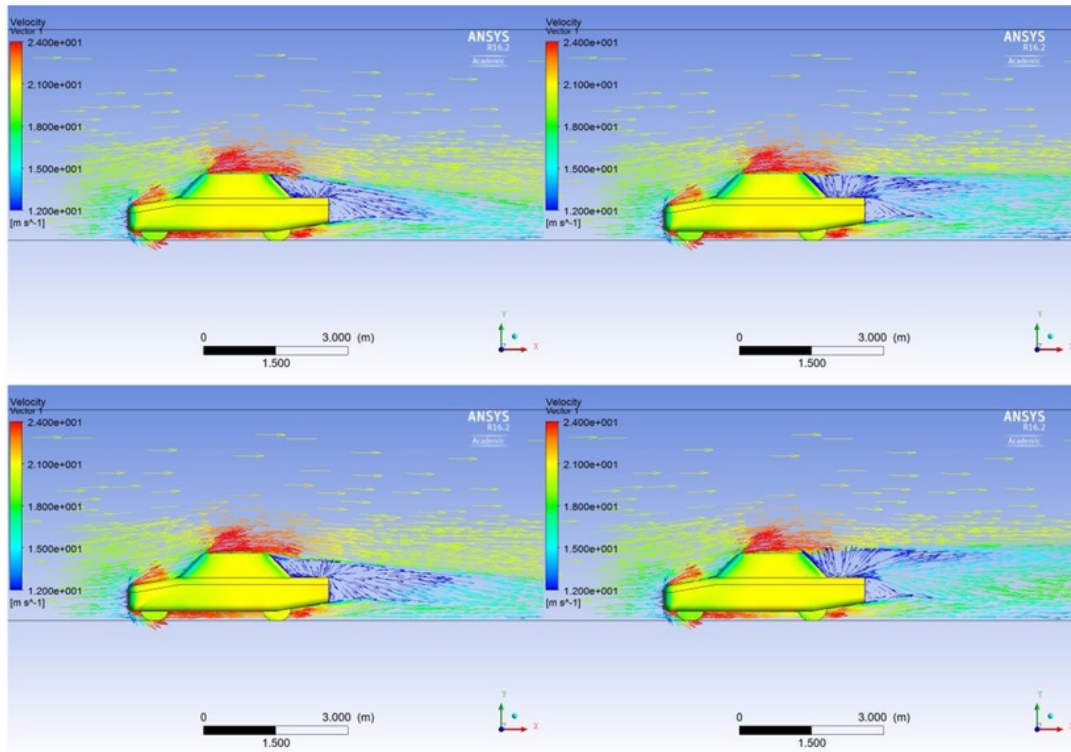


Figure 3: Velocity vectors for change in turbulence model

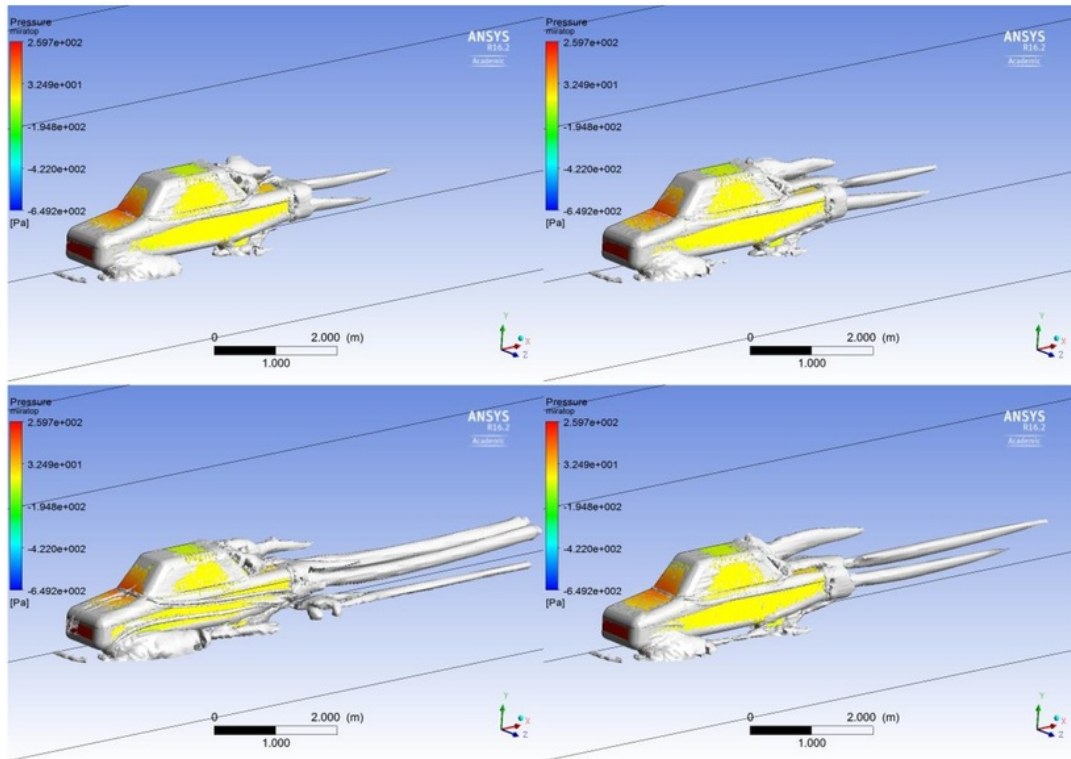


Figure 4: Vortex core region for change in turbulence model

5) Vehicle Geometry

Using all of the settings that were chosen in previous investigations, the vehicle geometry was altered in a way that would affect the aerodynamics of the vehicle. Two modifications were made to the original vehicle geometry; the first alteration was a roof sign added to the vehicle, similar to that of a driving instructor car or a taxi; the second was the original MIRA converted into a pickup truck.

The coefficients of lift and drag achieved were recorded and the difference from the MIRA reference values calculated (table 6).

Change in Vehicle Geometry					
MIRA with Roof Sign					
MIRA Ref					
CD	CL	CD	CL	%Diff CD	%Diff CL
0.4250	-0.0141	0.2963	-0.0845	43.44	83.37
MIRA Pickup					
MIRA Ref					
CD	CL	CD	CL	%Diff CD	%Diff CL
0.6378	-0.6328	0.2963	-0.0845	115.27	648.88

Table 6: Change in vehicle geometry

Once the simulations had converged, they were analysed in the results section of Ansys. Figure 5 shows the velocity contour plot on the symmetry plane. It also shows the separation of air and the wake region behind the cars. The reattachment of flow is quite different for the pickup truck because of the flow into the open cargo area.

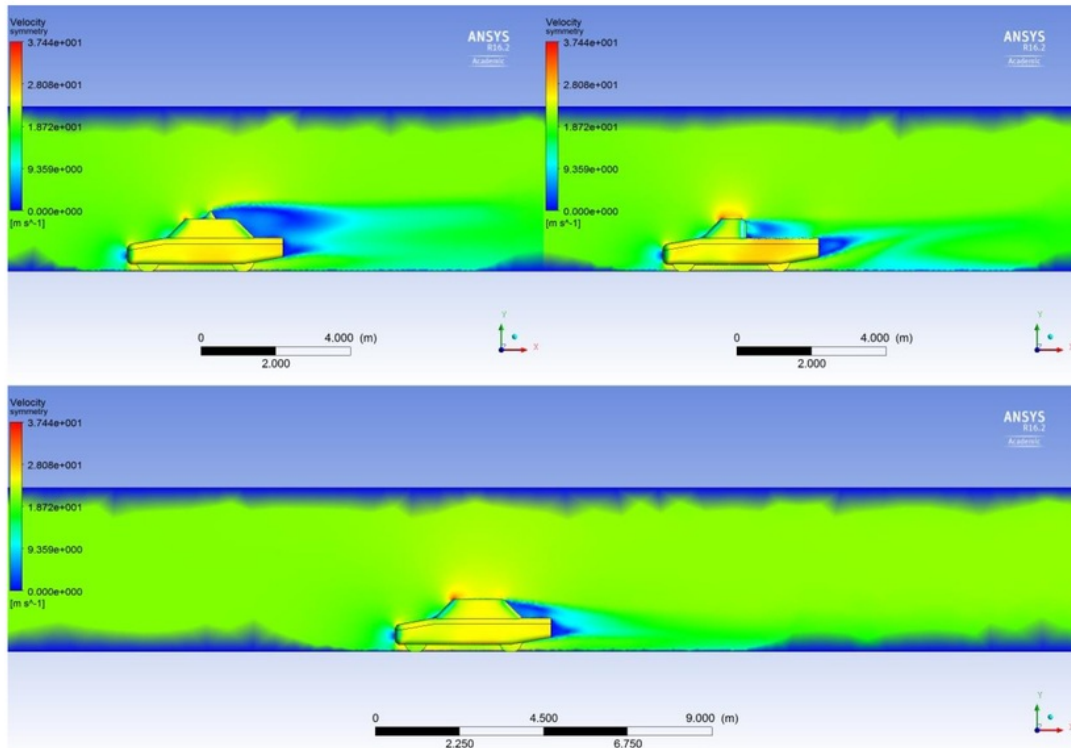


Figure 5: Velocity contour for change in geometry

Figure 6 show a better view of the pressure contour plot. It can be seen from figure 6 that the addition of the roof sign introduces another point of stagnation and separation of flow.

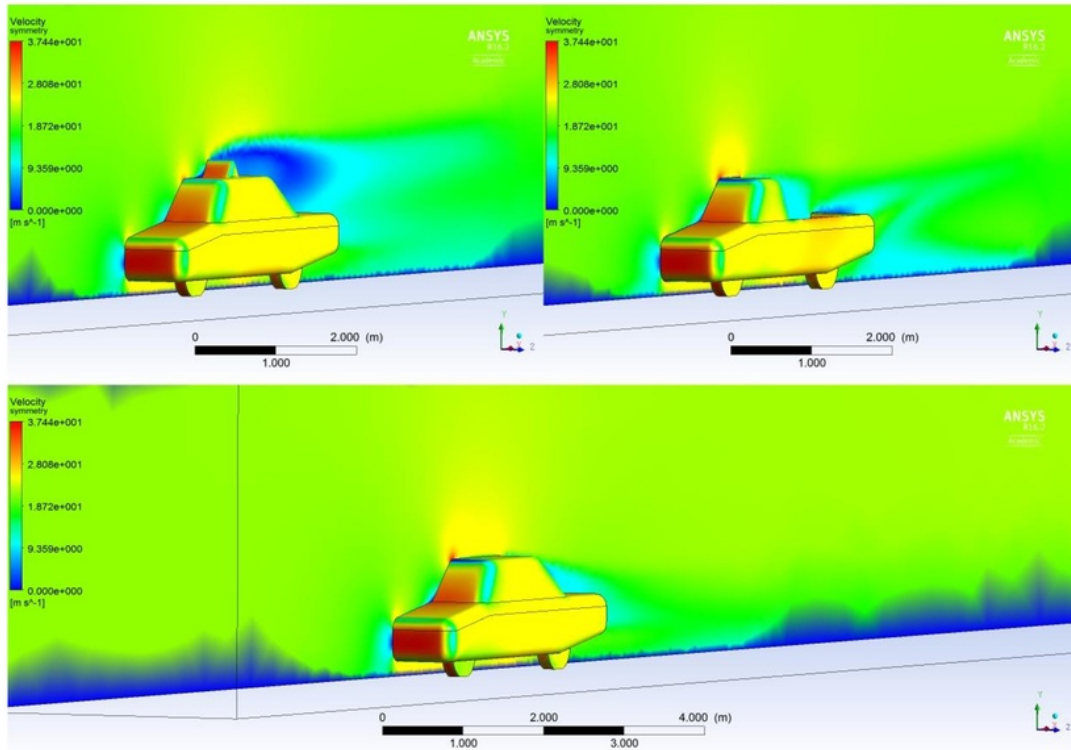


Figure 6: Pressure contour for change in geometry

Figures 7, 8 and 9 show velocity vectors, giving a better visual representation of the air flow around the vehicles. High velocity flow can be seen at the flat surfaces above and below the vehicles and at the points of separation.

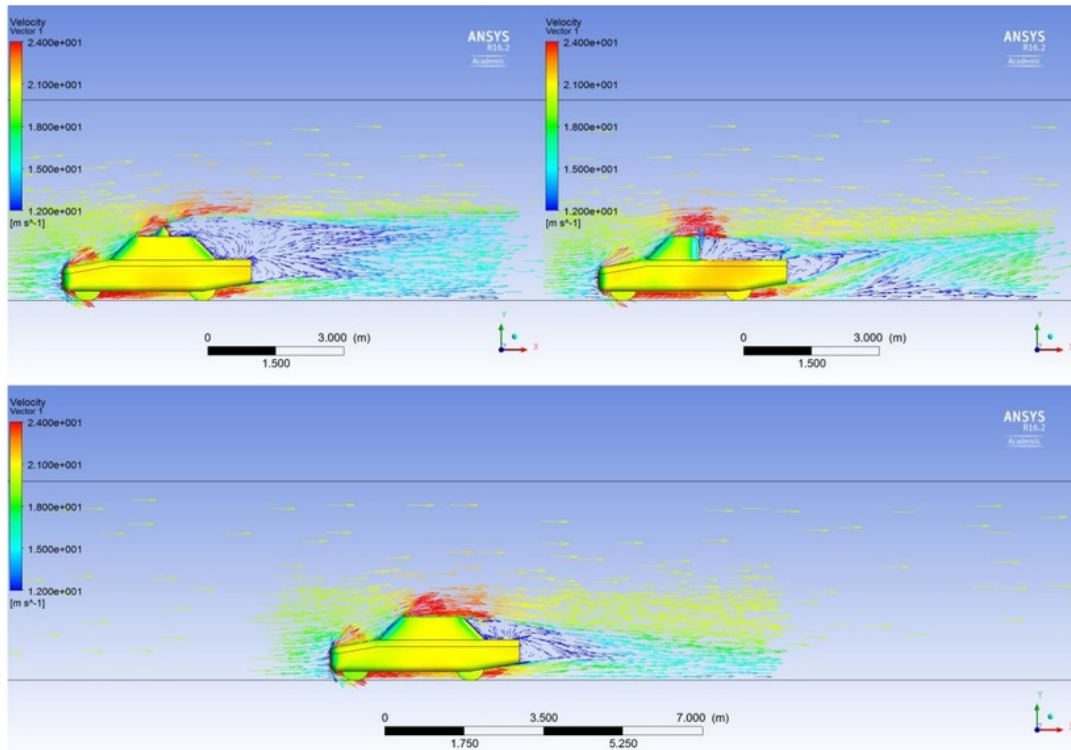


Figure 7: Velocity vectors for change in geometry

Figure 8 gives a closer look at the roof sign modification and how flow separates when hitting the roof sign. Eddy currents can also be seen in the wake region of the car.

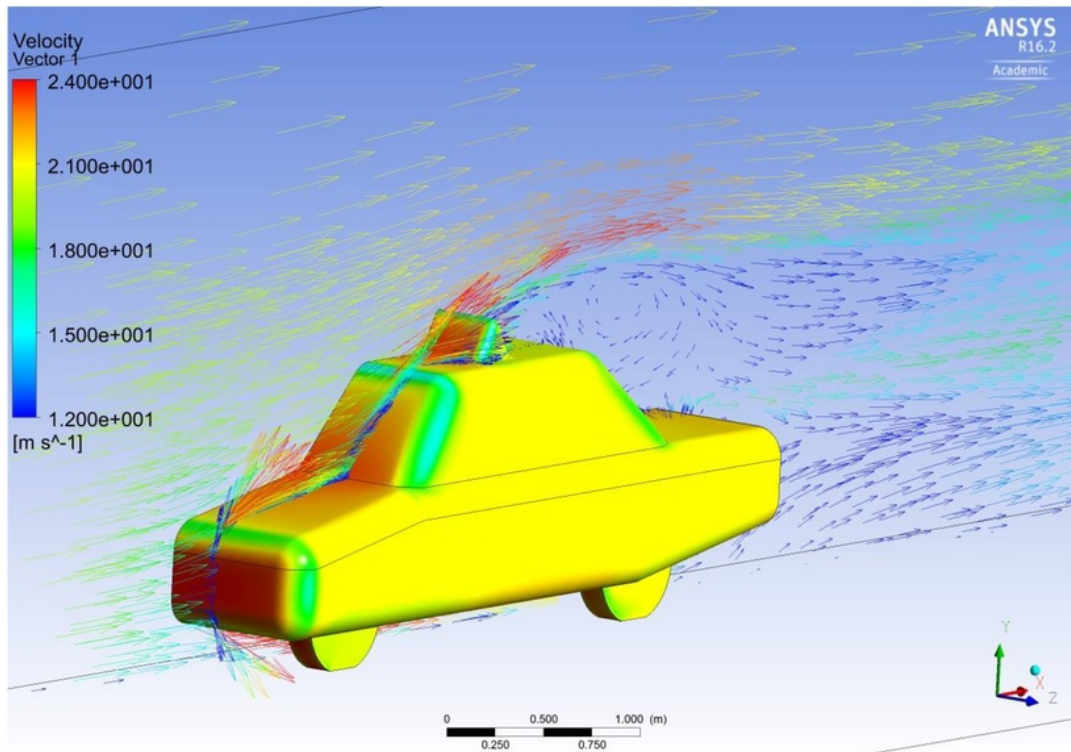


Figure 8: Velocity vectors for MIRA with a roof sign

Figure 9 shows how the flow of air at the edge of the open cargo area. There is high velocity flow at the corner of the cabin, and lower speed flow when the air is exiting the open cargo region.

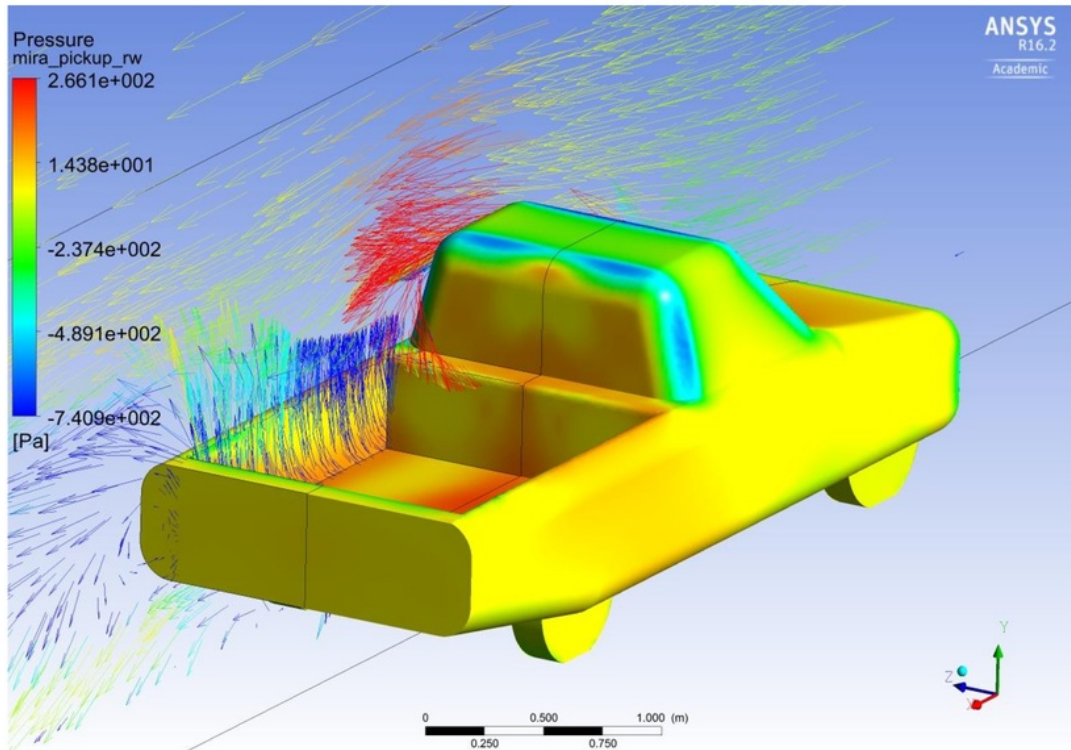


Figure 9: Velocity vectors for MIRA as a pickup truck

Figure 10 shows the vortices produced by the vehicles. Vortices occur when air flow turns into a swirling motion, and represent a source of drag. (Barnard , 2001) The vortices produced by the modifications are significantly larger than those produce by the MIRA car.

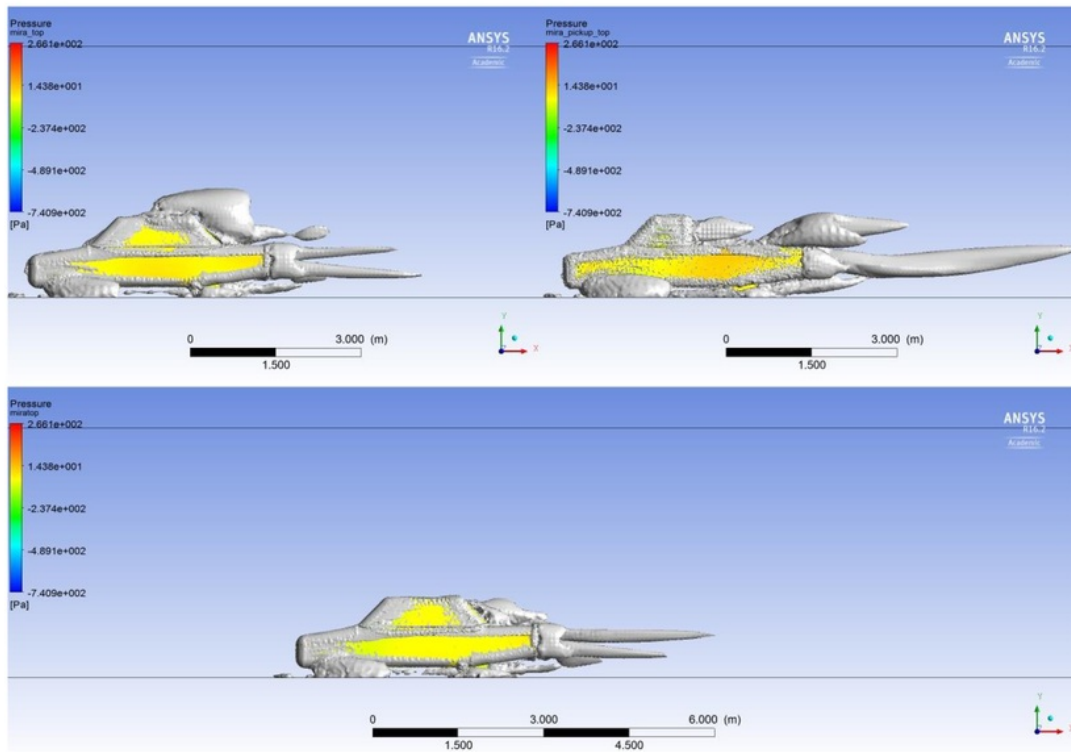


Figure 10: Vortex core region for change in geometry

Figure 11 shows the points where the vortices originate. There is a large vortex producing in the open cargo area in the pickup truck.

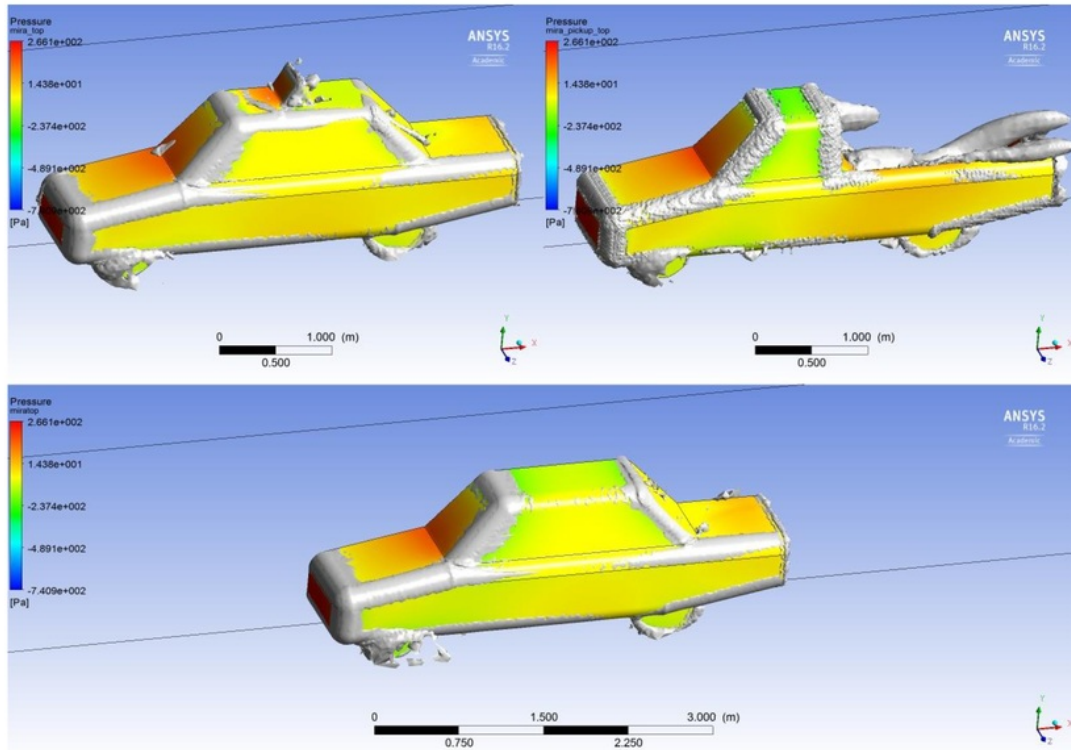


Figure 11: Initial points of vortex core regions for change in geometry

6) Conclusion

The process of finding the most suitable settings is time consuming and CPU expensive at times. The process was done by changing one setting at a time and reviewing the data and results in Ansys, then moving on to the next setting change.

The introduction of a geometry change of the MIRA car cause a great change in the aerodynamics in the car, in the two cases presented, for the worse, but different aerodynamic tools can be used to improve the performance of the vehicle.

7) References

Ahmad, N. E., Abo-Serie, E. & Gaylard, A., 2010. Mesh Optimization for Ground Vehicle. *CFD Letters*, 2(1), pp. 54-65.

Anon., n.d. *Non-Equilibrium Wall Functions*. [Online]
Available at: <https://www.sharcnet.ca/Software/Fluent6/html/ug/node513.htm>
[Accessed 25 January 2019].

Barnard , R. H., 2001. *Road Vehicle Aerodynamic Design*. 2nd ed. s.l.:Mechaero Publishing .

Fletcher, C. A., n.d. *Computational Techniques for Fluid Dynamics Volume 2*. 2nd ed. s.l.:s.n.

Frei, W., 2017. *Which Turbulence Model Should I Choose for My CFD Application?*. [Online]
Available at: <https://uk.comsol.com/blogs/which-turbulence-model-should-choose-cfd-application/>
[Accessed 25 January 2019].

Kapoor, A., 2016. *A Quick Approach to Boundary Layer meshing - For Beginners*. [Online]
Available at: <https://www.linkedin.com/pulse/quick-approach-boundary-layer-meshing-beginners-awadh-kapoor->
[Accessed 25 January 2019].

Lanfrit, M., 2005. *Best practice guidelines for handling Automotive External Aerodynamics with FLUENT*, s.l.: s.n.

Palaskar, P. M., Kumar, V. & Vaidya, R., 2016. *Methodology Development to Accurately Predict Aerodynamic Drag and Lift for Passenger Vehicles Using CFD*, s.l.: SAE Technical Paper.

Tudor, T., 2019. *CFD Tutorials*, s.l.: s.n.