

CFD_Assignment

by Zhaocong Sun

FILE	12271_ZHAOCONG_SUN_CFD_ASSIGNMENT_128450_73077762.DOCX (51)		
TIME SUBMITTED	28-JAN-2019 01:43AM (UTC+0000)	WORD COUNT	2737
SUBMISSION ID	99892227	CHARACTER COUNT	15188



School of Engineering

Aerodynamic Vehicle Testing and analysis

Zhaocong Sun

1805164

Automotive Engineering Beng

27/01/2019

Advanced Computational Methods M6X00408

Contents

1. Introduction	3
2. Applied mathematics	3
3. Standard model	3
3.1 Introduction to MIRA Reference Car	3
3.2 Geometry and domain.	4
3.3 Mesh using ICEM CFD	5
3.4 Fluent Calculation	7
3.4 Results& Post-processing	11
4. Changes to the model& Analysis	17
4.1 Changes to the domains	17
4.2 Changes to the mesh strategy	17
4.3 Changes to the geometry	21
5. Conclusion	23
6. Reference	24
7. Appendix(MATLAB CODE)	24

1. Introduction

Theoretical and computational methods are both used to solve the problems of vehicle aerodynamics. Computational fluid dynamics is especially used to access aerodynamic performance. ANSYS and FLUENT is used as simulation software.

In this report, the MIRA vehicle geometry is used as an aerodynamic tool to investigate into the performance of a common aerodynamic device using computational fluid dynamics.

After finishing geometry, meshing, boundary conditions, fluent, post processing and data analysis, some changes of the settings and the geometry should be made to see how it improves the aerodynamic performance. Then, the validated simulation methodology is developed using the baseline MIRA Reference car geometry and supplied wind tunnel data.

After that, some changes to the geometry should be made for deeper investigation. Then, this report will analyze how the geometry affect aerodynamic performance

2. Applied mathematics

To calculate the pressure coefficient, the below formula is used

$$\text{Pressure} = \frac{1}{2} \rho C_p V^2 \quad (1)$$

Which is:

$$C_p = \frac{2 \times \text{Pressure}}{\rho V^2} \quad (2)$$

3. Standard model

3.1 Introduction to MIRA Reference Car

This geometry is created by automobile manufacturers when the MIRA wind tunnel was commissioned. The geometry is simple to keep our mesh count down. It could allow us to do some reasonably quick investigation.

From the video, the size of the test wind tunnel area is of 15m length, 7.9m wide and 4.4m high with a cross sectional frontal area of 35m^2 .

3.2 Geometry and domain.

Assume that the vehicle is on a static floor for the first simulation. The geometry is symmetrical and the flow is from to the straight-ahead direction. We can simplify our model by assuming a symmetrical boundary condition down the longitudinal plane through the vehicle. And the flow should be the same on the left or the right hand side of the vehicle. That allows us to create a domain which is half the size of the vehicle.

Lanfrit's paper suggest that this domain should be 3 car lengths in front of the vehicle, 5 car lengths back of the vehicle, 3 car heights above the vehicle and 3 car width beside the vehicle. Assume that this MIRA vehicle is tested in a wind tunnel, so the tunnel size is used to inform our standard domain size.

Actually, the car was on pneumatic tires so the domain should be a little bit higher from the normal floor in order to make the vehicle intersect this domain. When creating the domain, we need to use 'add frozen' not 'add material' to let the two solid not combined. The width of the domain is set as 3.95 meters which is half of the tunnel width. After setting the boundary condition such as inlet, outlet, symmetry and topside, the car is broke up into smaller sections in order to investigate the pressure distributions on upper and lower surfaces. The following figure shows the domain and the geometry.

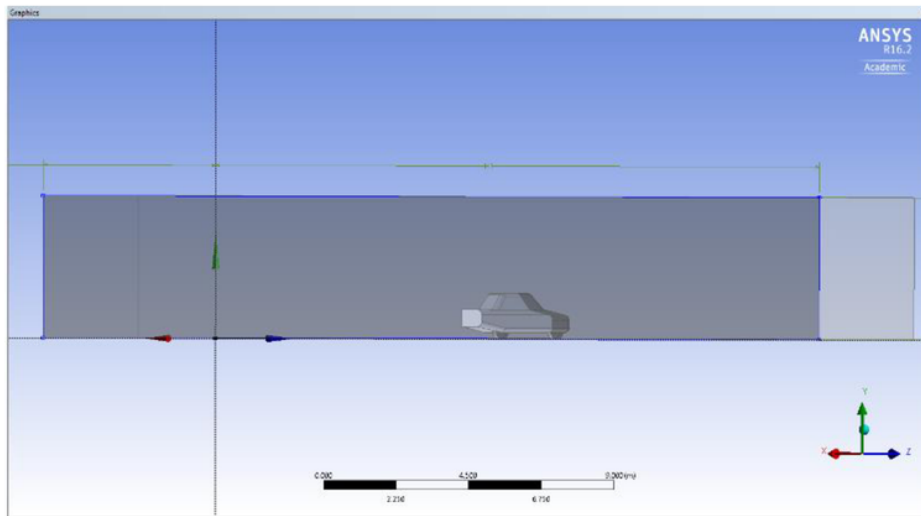


Figure 1 Domain and geometry of the standard simulation

3.3 Mesh using ICEM CFD

The ICAM CFD component is used to create better mesh for this simulation.

The general view of the MIRA car, symmetry and floor is created for meshing. In part of general model setup, the global element max element size is set as 1 meter. The parameters of the surface mesh are set to be all triangles while the mesh method is set as patch independent.

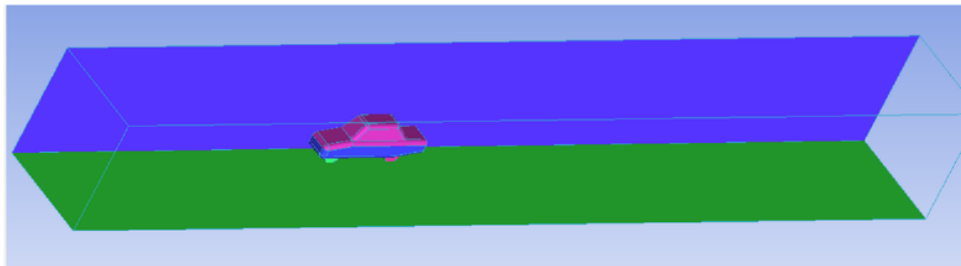


Figure 2 The schematic diagram for the mesh component

Actually, the general size can't meet the requirements of the mesh quality. One of the main tasks of this report and simulation is to find a better mesh quality. For areas far from the vehicle, the density could be large. But for the elements around the vehicle, we need more dense mesh to analyze the flows, pressure and the velocity. To make mesh represent the geometry very well, some element sizes on the car is specified.

For example, the maximum size of all the vehicle surfaces is set as 0.1 meters to make the mesh smaller and smaller.

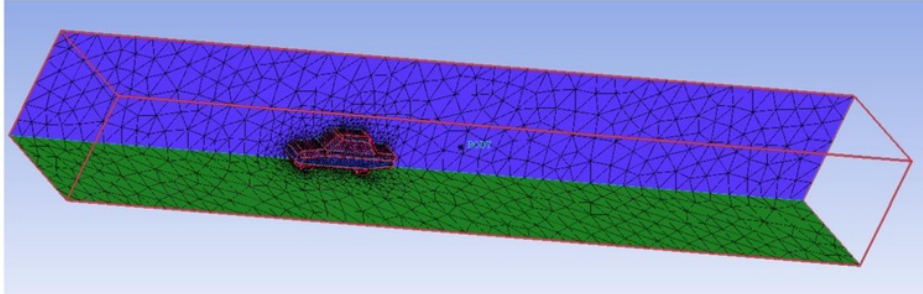


Figure 3 The area around the vehicle

Instead the vehicle itself, the domain around it is essential as well. To get a better definition of the mesh around the vehicle, the density box is created to refine a domain around the car where we expect significant flow separation. And the density box should give more space to the rear part of the car and less space to the front.

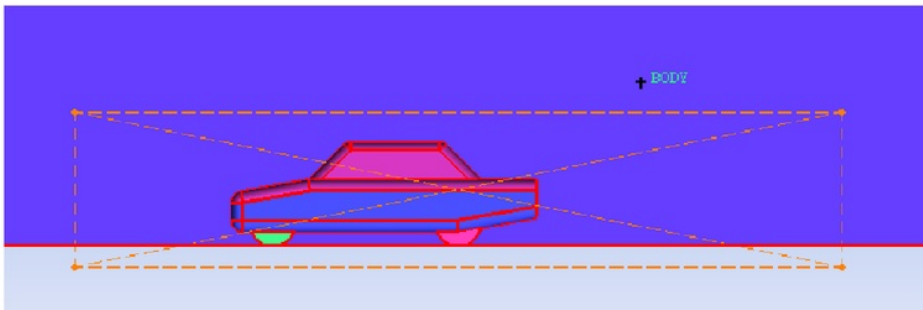


Figure 4 Density box of the mesh component

In addition, areas that are very close to the vehicle need to be further subdivided. The prism layers, where there are a number of elements close to the surface, are set as 5 elements to catch the boundary layer. The 'octree' mesh is set and very good for poor quality mesh, and the middle of the domain isn't good for CFD analysis.

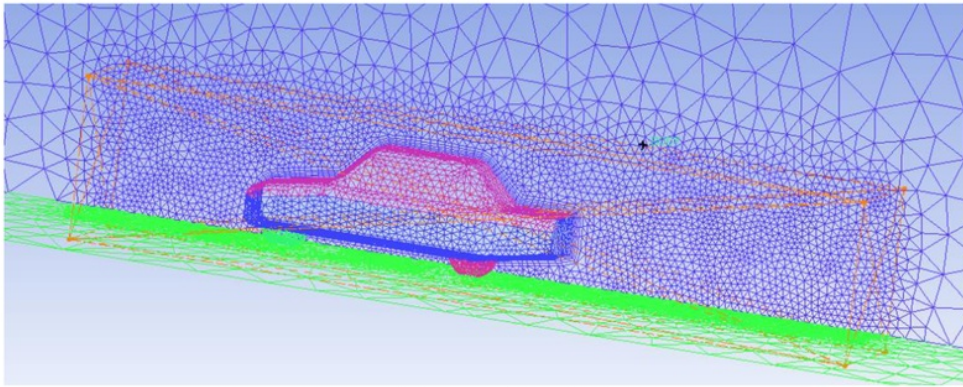


Figure 5 Overall figure of the Mesh domain

We can see the general distribution of the mesh from the above figure. And the lecturer introduced a different method to create mesh, which is much more convenient. After setting the density box, we just need to use Surface Mesh, Volume Mesh and Prism Mesh.

3.4 Fluent Calculation

The Fluent product is integrated into the software ANSYS. Firstly, every five elements is joined into polyhedral to make the calculation speed five times quicker, which is shown as below figure

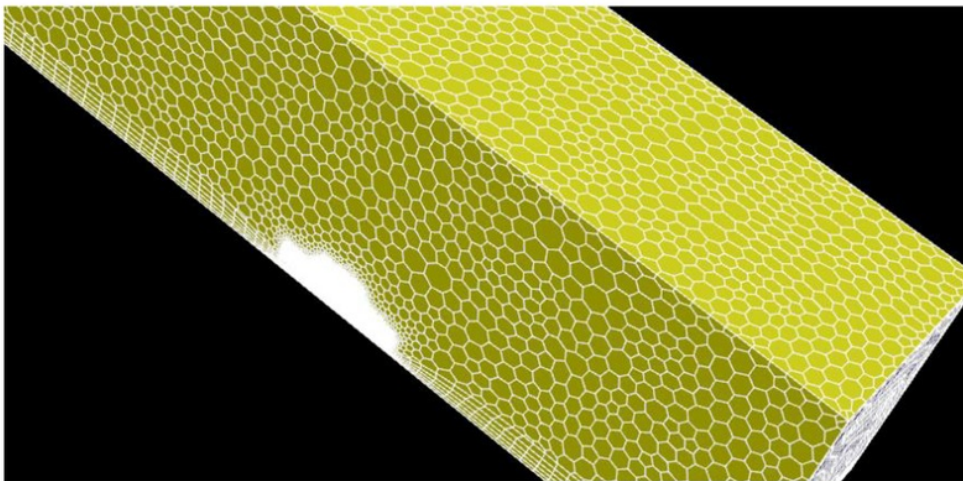


Figure 6 Polyhedral elements

In order to get more accurate results, the function of polyhedral is not used. The simulation type is set as pressure-based model because the vehicle is at subsonic speed. In viscous model, a transport equation is used to see how quickly the energy is

getting dissipated and predict how much energy these points have. Lanfrit[1] suggest that we use a k-epsilon realizable and non-equilibrium wall functions, which is useful and supported by multiple sources. Then, the material is checked to be air and the body is specified as the fluid.

The next setting is boundary conditions, such as the inlet velocity to be 20m/s and the outlet to be a pressure outlet. The reference values is always set as inlet, and area of the MIRA car is set as the frontal area of the inlet

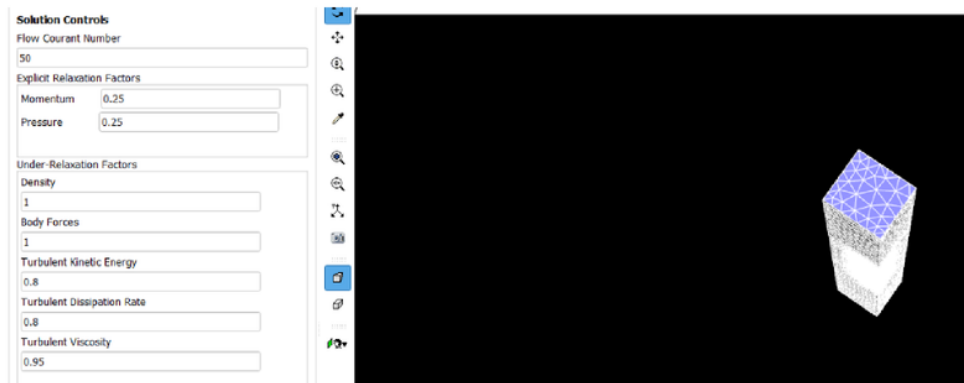


Figure 7 Solution controls

Lift and drag is monitored, which is shown below as figure 8. We hope to see the residuals slowly drop and the coefficient of lift and drag they might wave initially and stabilize at a value. And the residuals have got down. And we normally aim to multiply it together. The drag and lift would converge on some values.

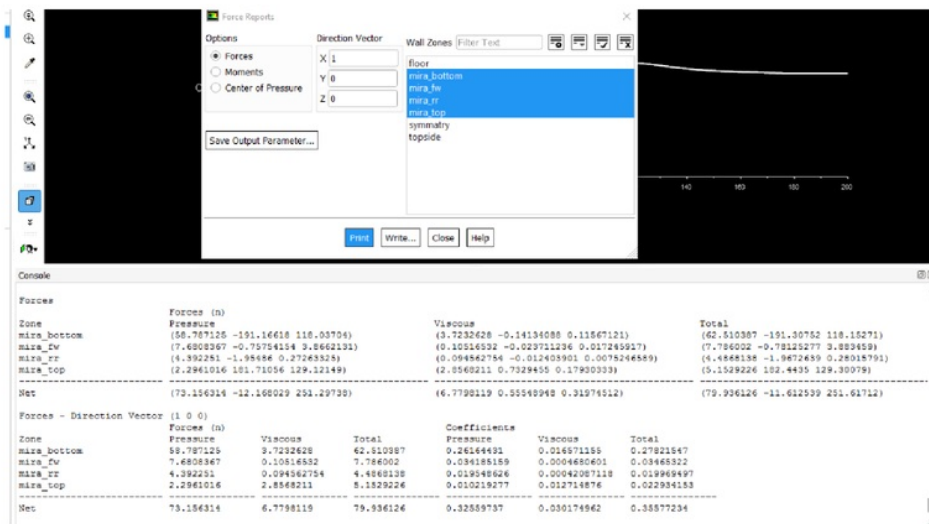


Figure 8 Print the lift force and coefficient

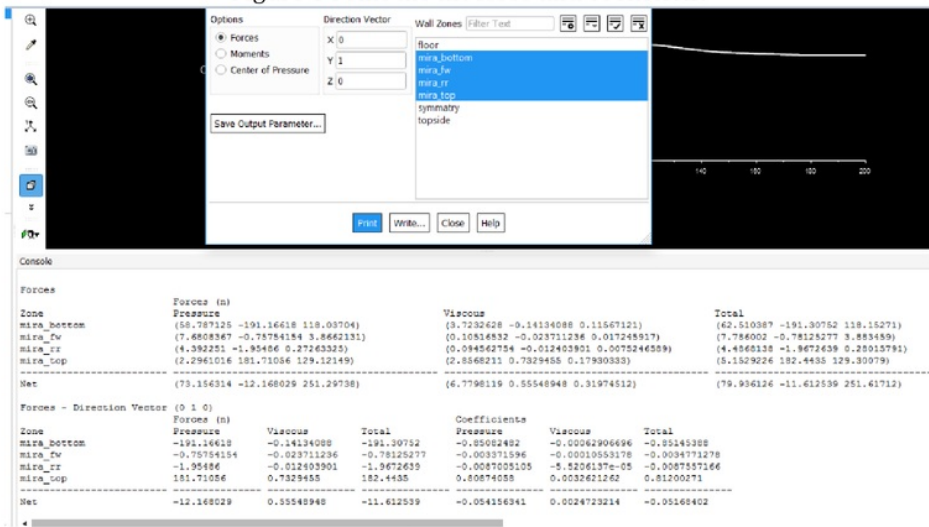


Figure 9 Print the drag force and the coefficient

From report, the above figure tells us about the drag of the MIRA car in the X direction and the lift of the MIRA car in the Y direction. And after 315 iterations, the C_L and the C_D becomes more stable. The values for drag for the actual experimental version of this car is that drag was about 0.29, and in this simulation the value is about 0.35, which is a little bit higher.

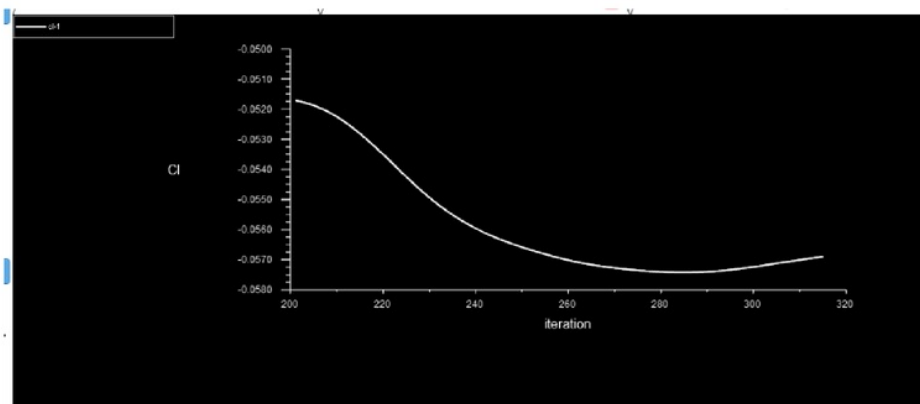


Figure 10 The trend of lift coefficient

The lift coefficient is a little bit negative. After 200 iterations, the drag or the lift will not be converged but it's certainly on the way toward converging on a solution. We could calculate more iteration to wait for the convergence.

The contours in graphics could be used to have a quick view of the calculating results. The pressure of the MIRA car and the symmetry surface is shown as follows.

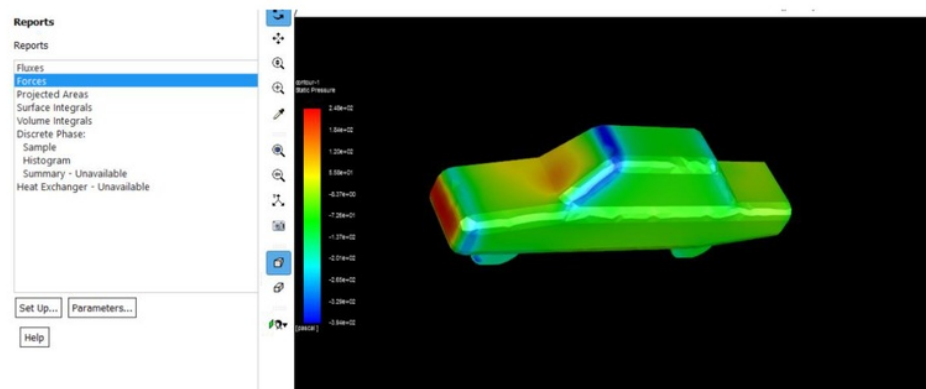


Figure 11 The pressure of the contours in graphics in Fluent simulator

We've got a pressure distribution basically like we'd expect, which kind of validates our boundary conditions. The flows probably come in the right direction where we've got a high stagnation pressure in front. The highest pressure is at the front of the vehicle and the windshield.

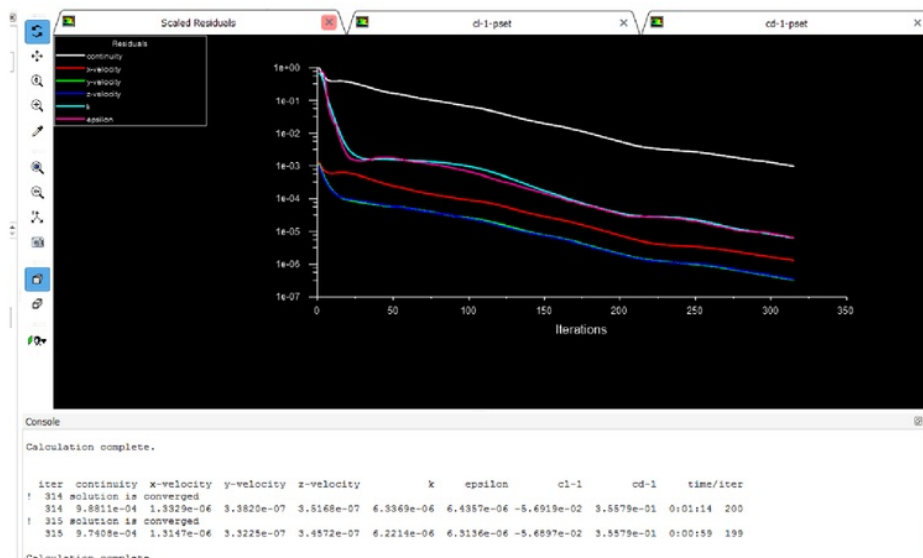


Figure 12 The residuals of x-y-z velocity

We wish to look at not just pressure contours but velocity contours. We can see we've got separation off the back window and behind the vehicle. We've got low speed at the front of the vehicle and the stagnation areas where we'd expect.

3.4 Results& Post-processing

This module could view the results of the simulation better. We could see the pressure through clicking the surface as follows.

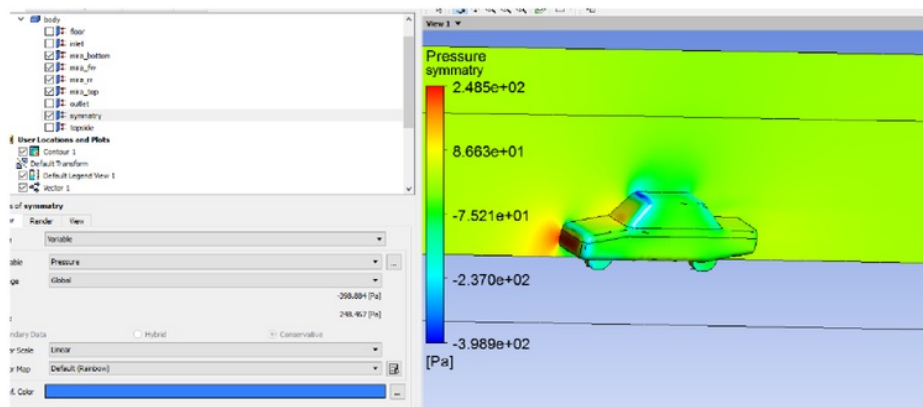


Figure 13 The pressure distribution of the vehicle

We have some higher velocity underneath specifically at the front. At the kick point, there are also some high velocity points. We could certainly see there's a very concentrated high velocity over the top edge of the roof where the flow is ripping over

the leading edge, which gives us a good indication straightaway of the general flow characteristics.

To have the symmetry plane, we only used the half-car model because the left and right sides of the vehicle are symmetrical. In order to restore it, instance transform domain is formed through XY plane.

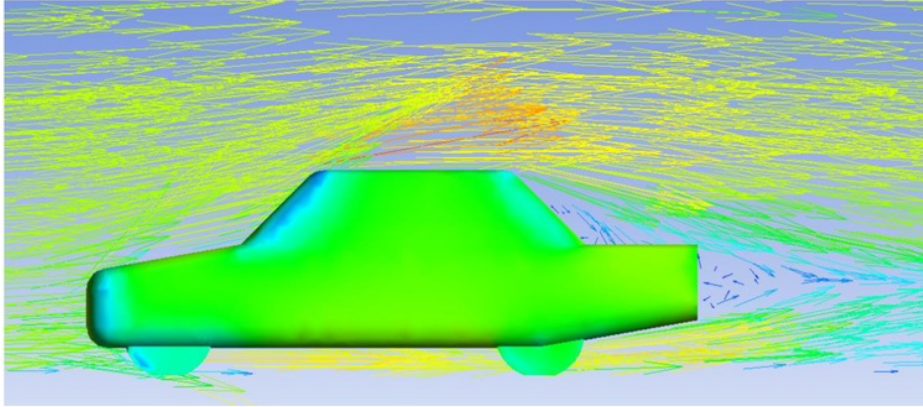


Figure 14 Velocity vector with direction

A XY plane with the 0.22m length of Z-axis is created to use contour to see the pressure or the velocity. The above vector is created to see the direction of the velocity. They are actually three dimensional vectors if you look at the plane along in line. With the X-Y plane you can see those vectors deviate out and in as that plane giving us a good idea of what the flow trajectory is at any of those points.

The domain around the vehicle is quite messy because the meshes here are quite small. The flow reversal and the vortices along the back have been got. This is because the difference of the pressure between vehicle back pushes the air to form a vortex. This gives us a much better idea of what's actually happening)

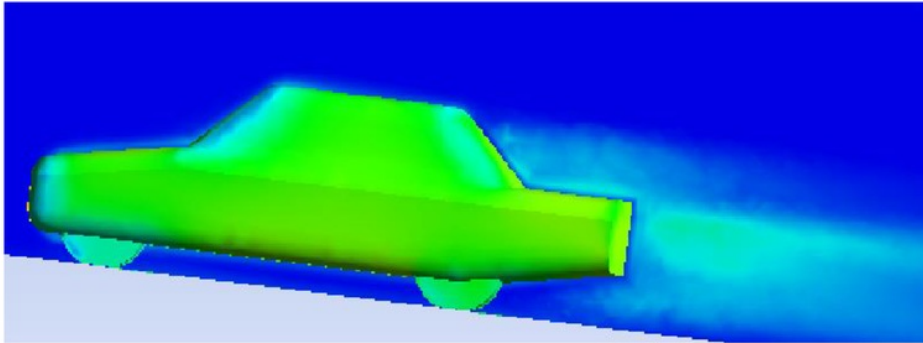


Figure 15 The kinetic energy of turbulence at XY plane

The XY plane also shows the turbulence kinetic energy, which is mostly concentrated at the back of the vehicle.

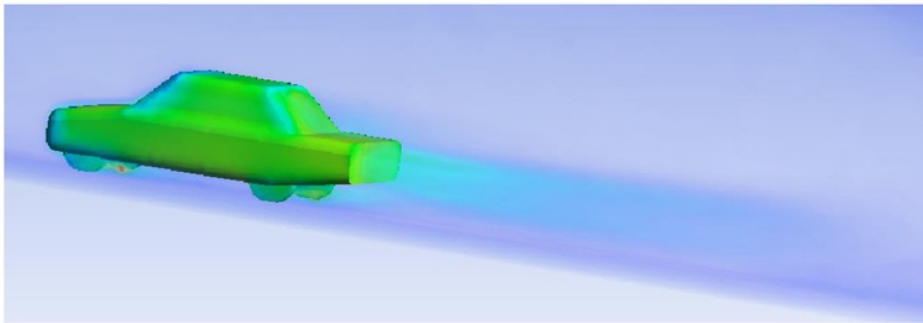


Figure 16 The domain to show turbulence kinetic energy.

And the volume render could give us a 3-D domain which shows turbulence kinetic energy. From the figure, the kinetic energy is around the edges and behind the back.

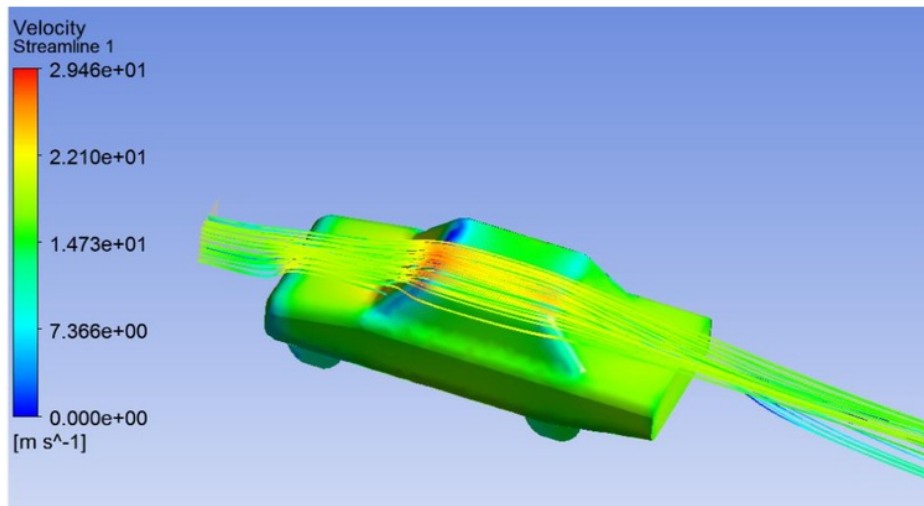


Figure 17 The streamline of the velocity

The streamline is usually used for sort of PR images. After that, a rectangle in front of the vehicle is created to show the streamline. If I move the rectangle a little bit lower, it will look like figure 18.

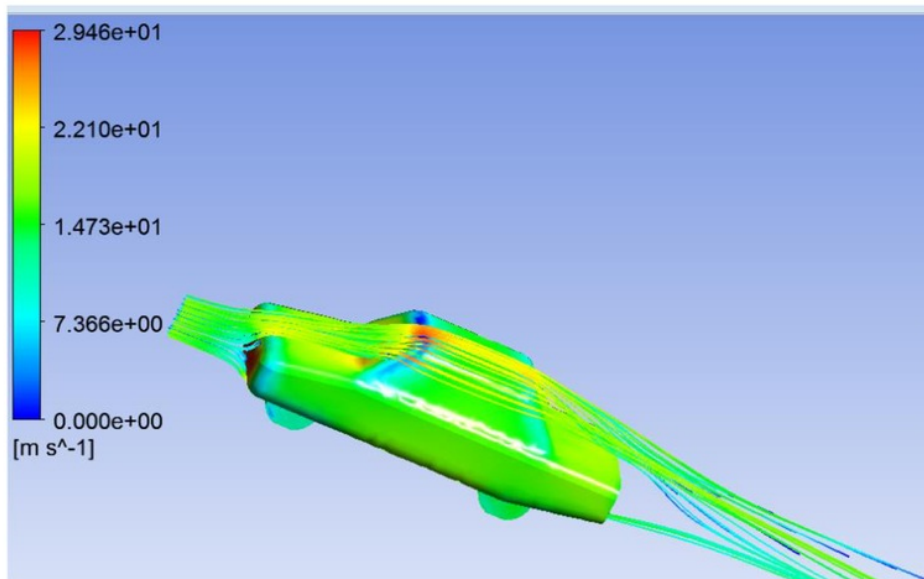


Figure 18 The streamline of the velocity

There are some streams going through the bottom of the vehicle because some flows have touched the edges of the front of the vehicle. If I move the rectangle a little bit lower, it will look like figure 19.

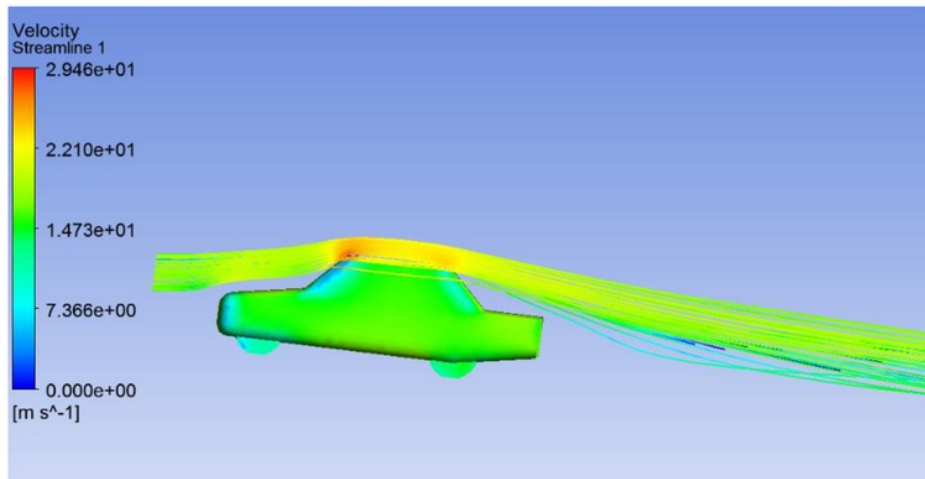


Figure 19 The streamline of the velocity

There will be no changes in flow boundary conditions, and the turbulence goes downstream and the turbulent eddies get smaller. After that, the polyline is created between the top & bottom and the symmetry plane to plot the relationship of pressure and X-axis geometry.

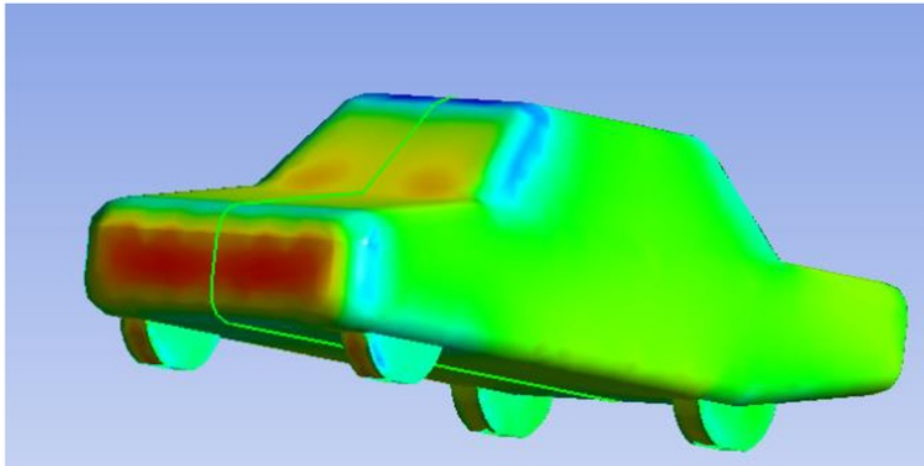


Figure 20 Polyline of the vehicle

After that, the pressure along the vehicle could be extracted out to MATLAB and be compared to the experimental data.



Figure 21 Upper: standard simulation comparison data

And

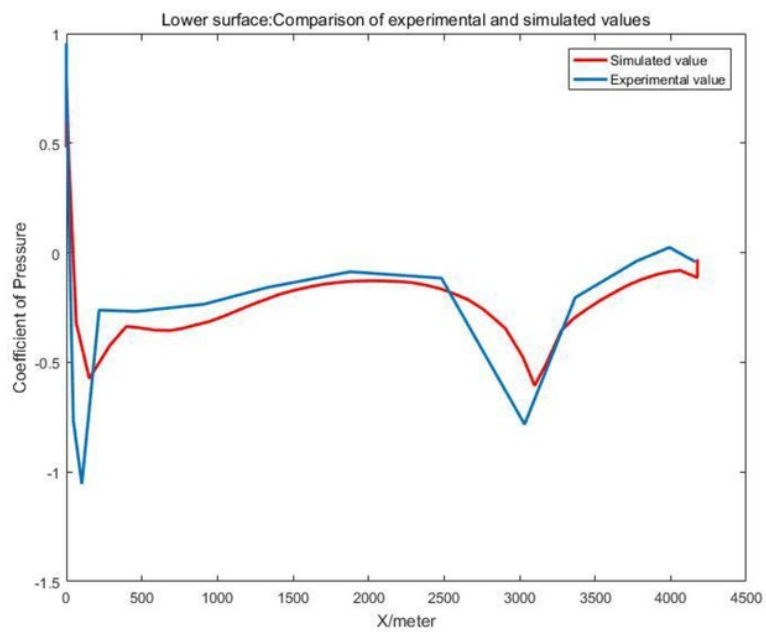


Figure 22 Lower: standard simulation comparison data

The standard simulation is roughly the same as the experimental value. The shape is almost the same. The deeper simulation will be discussed later

4. Changes to the model& Analysis

4.1 Changes to the domains

As is defined in 3, the inlet area is 12(3 times of vehicle length) meters in front of the MIRA car. I changed the distance to 4m, 8m and 12m to see how the C_L and C_D changes.

Table 1 The C_L and C_D of different front domain size

Front distance	C_L	C_D
8m	-0.219	0.327
12m	-0.058	0.362
16m	-0.0516	0.3557

Actually, I think the front size of the domain has nothing to do with the simulation results. When I am doing the standard simulation, I move the rectangle a little bit far away from the vehicle, the streamline is just a straight line, which means the domain far away from the front of the vehicle will not influence the stream and the aerodynamic performance.

I have change the rectangle to left and right side of the vehicle to see the turbulence shape. In my opinion, the domain edge which is a little bit far away from the left or the right of the vehicle will not affect the vehicle performance, so I didn't do further tests about the domain.

4.2 Changes to the mesh strategy

In standard simulation, the number of the prism layer is 5. And I changed that to 3 and 7 to see the results

Table 2 The C_L and C_D of different prism layers

prism layers	C_L	C_D
3	-0.0698	0.345
5	-0.058	0.362
7	0.01099	0.3514

From table, the C_L becomes bigger and it is getting closer to 0. With the growing number of the layers, the surface around the vehicle have smaller and better meshes.

The following figures are the simulation result data compared to the experimental data with 3, 5, 8 prism layers.

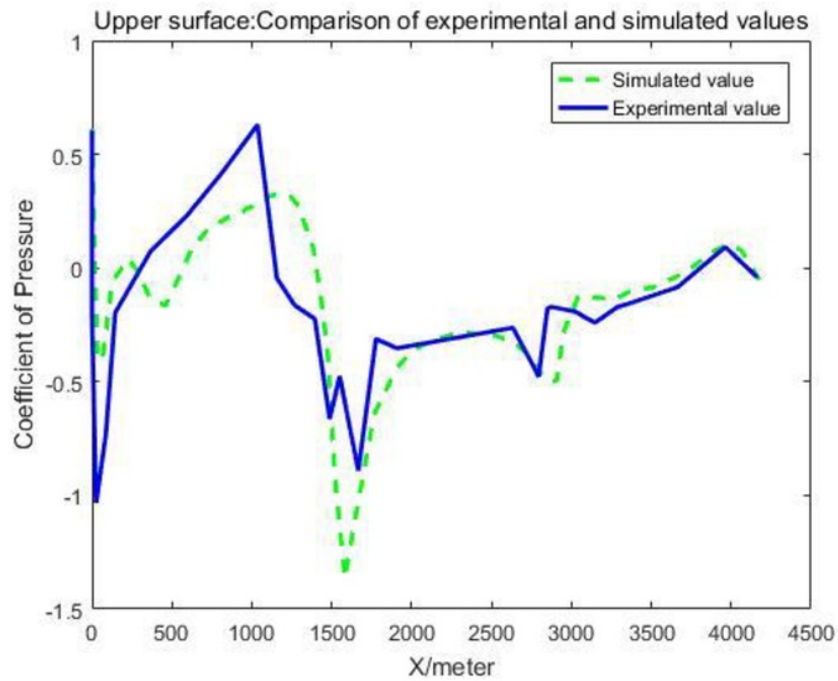


Figure 23 Upper: simulation with 3 prism layers comparison data

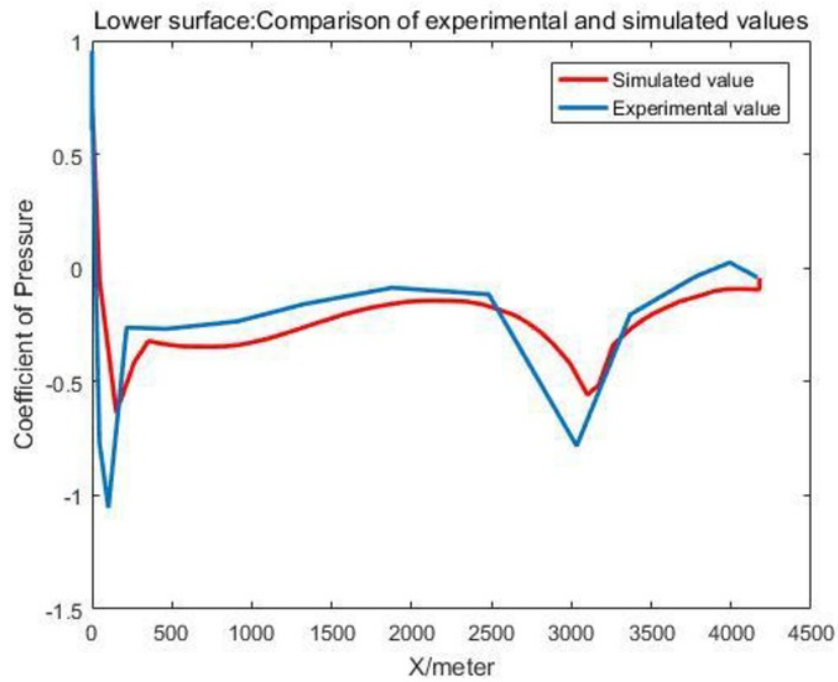


Figure 24 Lower: simulation with 3 prism layers comparison data

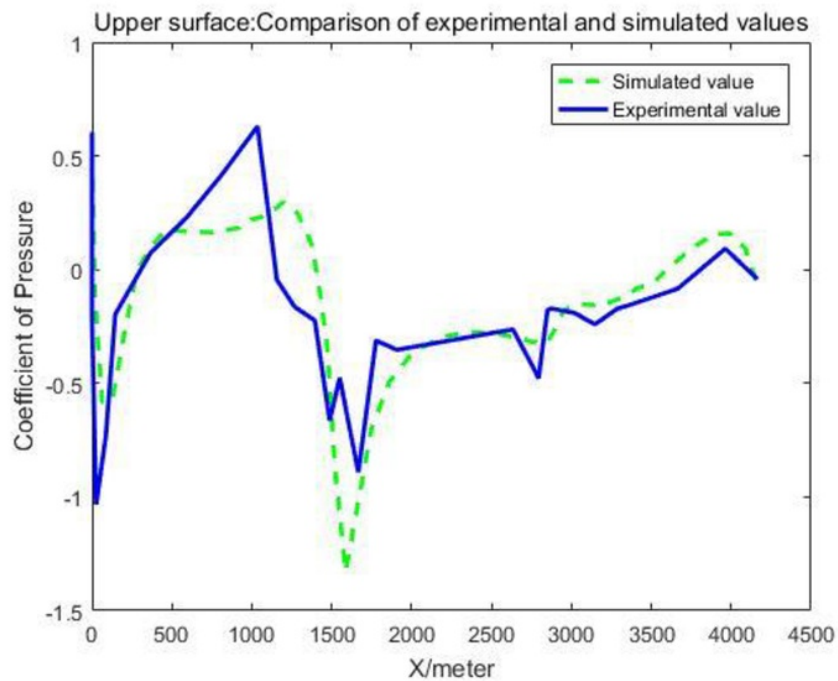


Figure 25 Upper: simulation with 5 prism layers comparison data

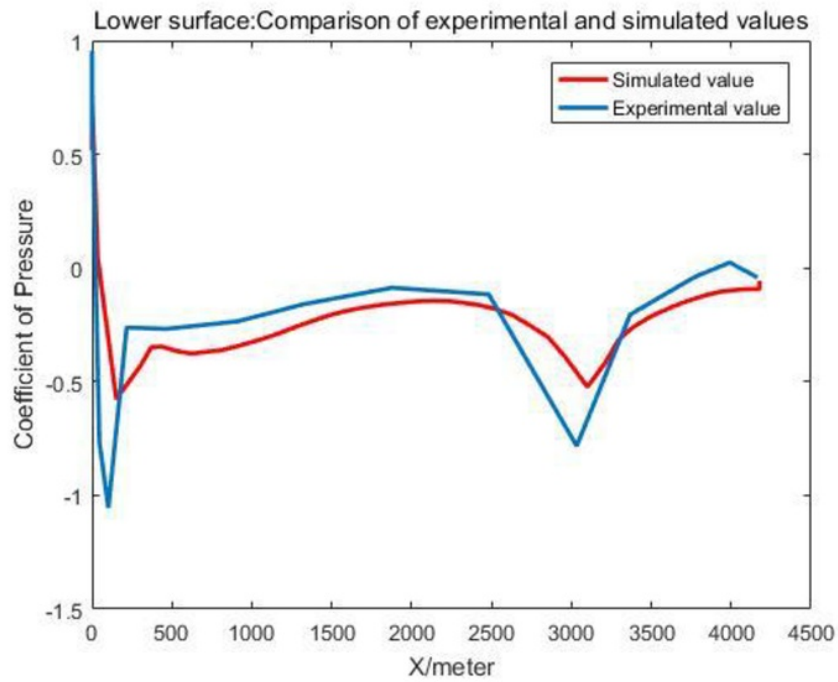


Figure 26 Lower: simulation with 5 prism layers comparison data

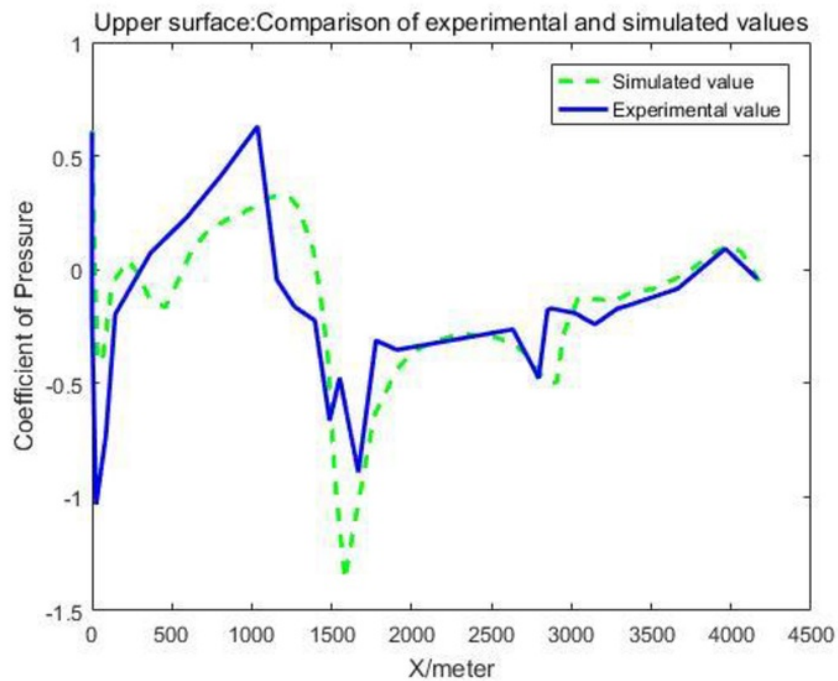


Figure 27 Upper: simulation with 8 prism layers comparison data

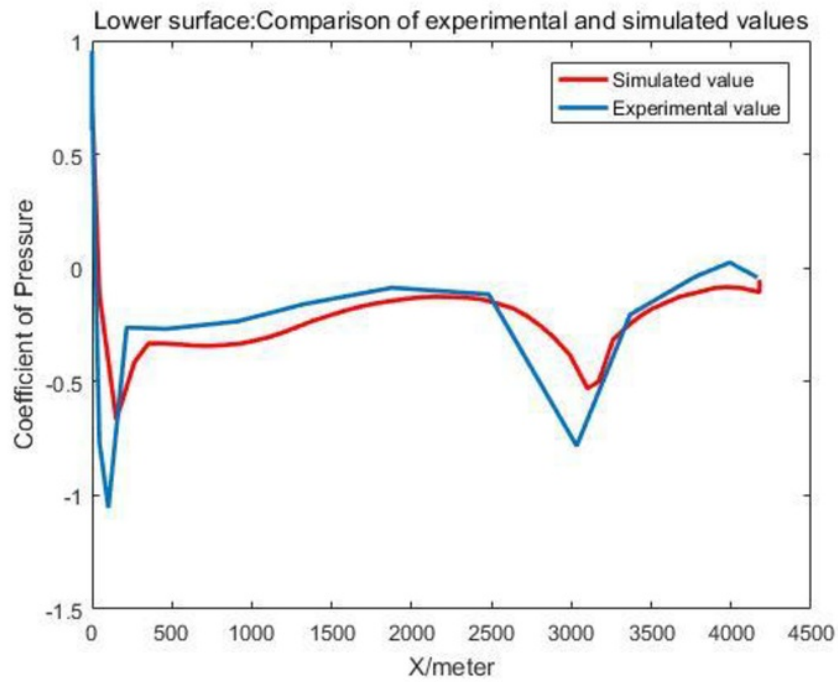


Figure 28 Lower: simulation with 8 prism layers comparison data

The difference between these test is not very big, and the curves are close to each other.

4.3 Changes to the geometry.

I have added a tail to the vehicle, which has the angle of 10 degrees and 20 degrees and are shown below.

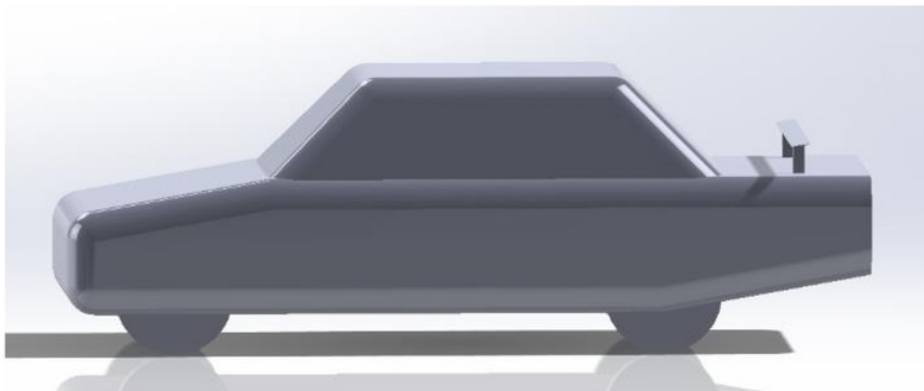


Figure 29 10 degrees MIRA model

And

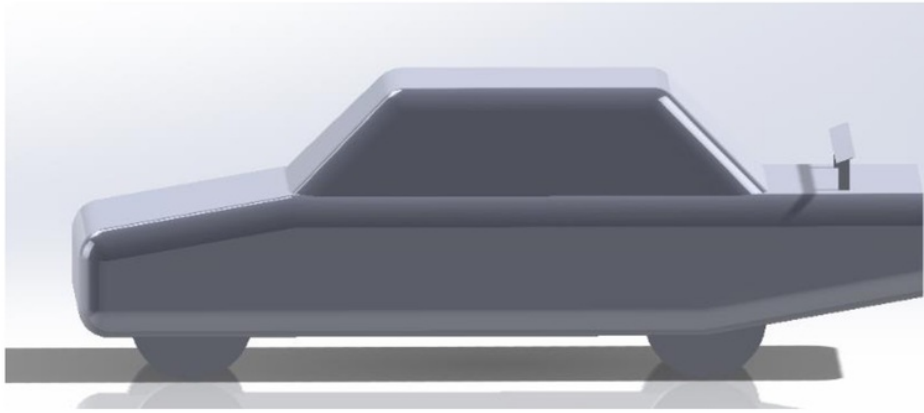


Figure 30 20 degrees MIRA model

When I was creating the mesh, I noticed something interesting about the tail.

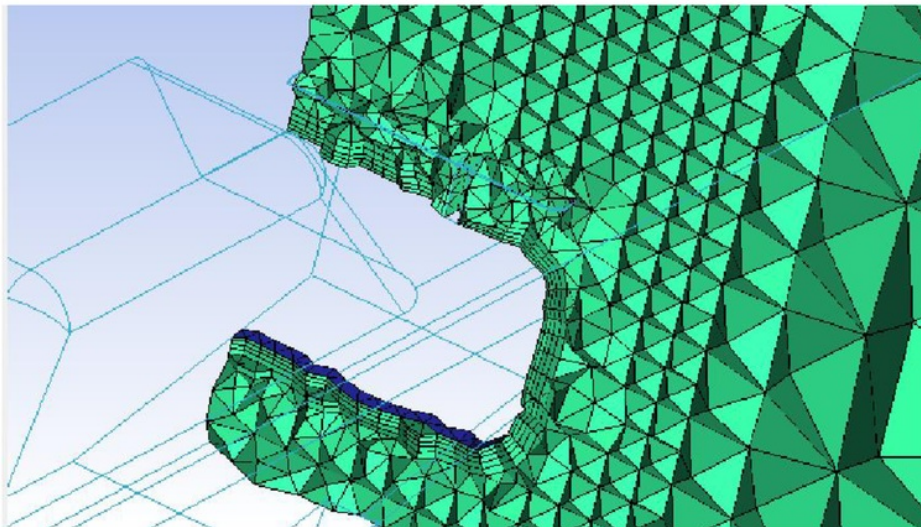


Figure 31 The mesh volume about the tail

As you can see, the mesh at the tail is different from other parts. Actually, the mesh at the tail sag into other domain, because I just set the tail as a part of 'MIRA_top'. Next time I will set the tail separately. The simulation results are as follows:

Table 3 The C_L and C_D of different tail degrees

Tail degrees/ $^{\circ}$	C_L	C_D
10	-0.213	0.312
20	-0.2164	0.3085

Compared to the experimental results, the C_D is close to the data 0.2963. But the coefficient of lift is not accurate enough. The down force has been added due to the tail angle, thus the C_L will be more negative.

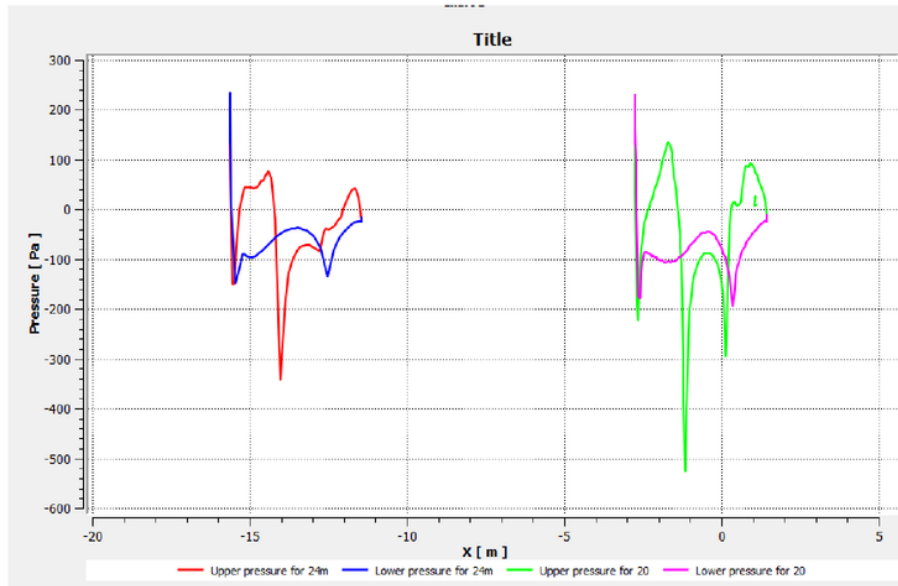


Figure 32 The comparison between the 20 degrees tail and no tail.

From figure 26, the height of the shape shows that the upper body with the tail would endure more negative pressure. This is because the tail gives huge down force to the vehicle. We don't need to look at the X-axis, because the difference is from the different coordinates of the geometry.

5. Conclusion

This report has investigated what parameters in the simulation will affect the mesh quality and the aerodynamic performance. First, the mesh should be small around the aerodynamic devices. Second, the geometry should be as realistic as possible. Third, the prism layers and the parameters of the solver controls should be suitable to the problem.

What is interesting is that there are many features in ANSYS that will allow you to analyze your results very well.

6. Reference

Marco Lanfrit. Best practice guidelines for handling Automotive External Aerodynamic with Fluent. University of Southampton.

7. Appendix(MATLAB CODE)

```
com_up_x1=(com_up_x-com_up_x(102))*1000;
com_down_x1=(com_down_x-com_down_x(1))*1000;
den=1.293;
com_down_y1=2.*com_down_y/den/20/20;
com_up_y1=2.*com_up_y/den/20/20;
figure
plot(com_up_x1,com_up_y1,'g--',ori_up_x,ori_up_y,'b','LineWidth',2);
xlabel('X/meter');
ylabel('Coefficient of Pressure');
title('Upper surface:Comparison of experimental and simulated values')
legend('Simulated value','Experimental value')
figure
plot(com_down_x1,com_down_y1,'r',ori_down_x,ori_down_y,'LineWidth',2)
;
xlabel('X/meter');
ylabel('Coefficient of Pressure');
title('Lower surface:Comparison of experimental and simulated values')
legend('Simulated value','Experimental value')
```