

Brett Nethell P151067

by Greison Brett Nethell

FILE	3668_GREISON_BRETT_NETHELL_BRETT_NETHELL_P151067_128450_426009652.DOCX (35)		
TIME SUBMITTED	27-JAN-2019 06:24 PM (UTC+0000)	WORD COUNT	880
SUBMISSION ID	99873243	CHARACTER COUNT	4175

Aerodynamic Vehicle Testing and analysis

By Brett Nethell

P151067

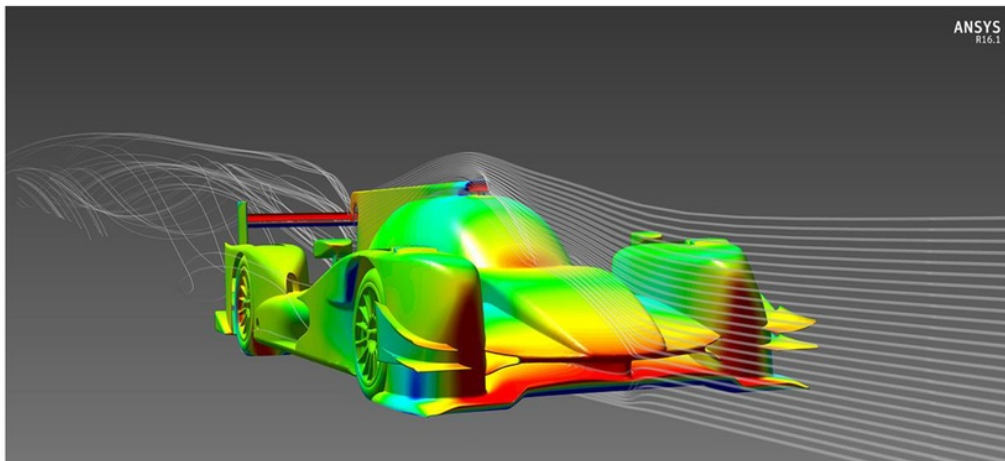


Figure 1

Introduction

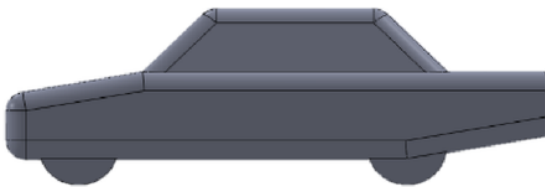
This assignment will be looking at how altering the MIRA aerodynamic car model with a Spoiler/ rear wing will affect its different areas of drag, turbulence and more. Firstly the basic model will be used to set up a method which will be used on the altered MIRA model. Once a set method has been developed, the next stage will be to design and add on a spoiler/rear wing to the MIRA model, this will likely affect the drag on the MIRA car, but this rear wing's main purpose would be to provide downforce not to reduce drag.

MIRA Car Model

The MIRA car model has come from a full scale wind tunnel testing experiment, which is where all the data comes from. Taking the MIRA model and using it on ANSYS, different data can be compared to real life and computer generated data. Depending on the different parameters used such as mesh quality, wind speed and other variables the results obtained should always be similar to that from real life, however the real life data may never be achieved through ANSYS due to factors that the computer cannot add in/measure.

The simulation will be done by simulating the car travelling at 20 m/s, this is because that when the real life data was collected it was simulated air at 20 m/s.

The MIRA car model is shown below, projected as a side profile the MIRA car model is a vision of one side of a real car.



Adaptation (REAR WING/ SPOILER)

For this assignment the MIRA car model must be altered by some means and then measured with the alteration, the alteration decided for this assignment was to add on a rear wing or spoiler, whether this be a lip spoiler or a proper racing rear wing. The reason this was chosen is because the rear of the car is very straight and angled rather than being curved or include a lip spoiler. It is known that the rear wing/spoiler might not reduce the overall drag of the car but it will help to provide downforce to car. Stability at top speeds will also be improved however we will not be looking at this in this assignment.

Methodology

- Same method for both standard geometry and altered geometry.
- 20 m/s wind speed/simulated travelling speed.

Task 1

Importing MIRA geometry: The standard MIRA geometry was already attained through the university and downloaded from a USB stick, using this and lectures on using CFD software ANSYS the investigation followed.

Task 2

Create mesh: The next step was to set the mesh quality and specific regions, in the first simulation the specific region was just situated around the MIRA model and not concentrated directly in any area. However, moving forward, the mesh will be more concentrated /saturated towards the rear was the MIRA car model.

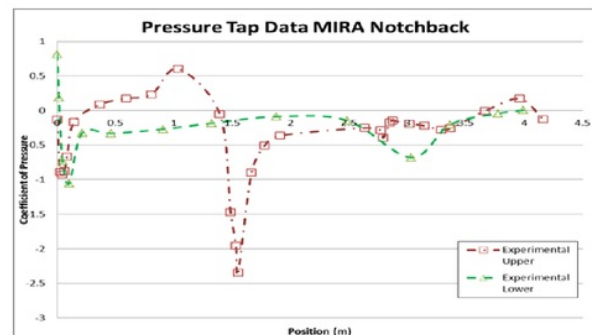
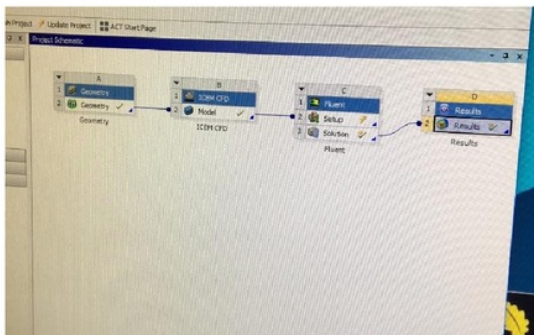
Task 3

Solve and analyse: Running the simulation to achieve results and following this, investigate different flow regions and numbers.

Task 4

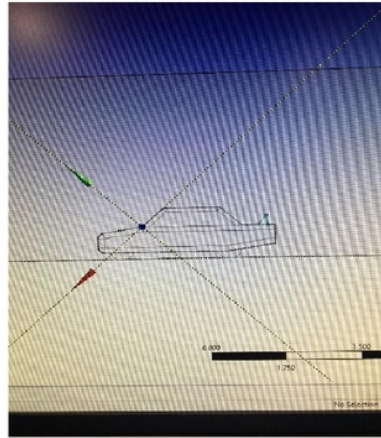
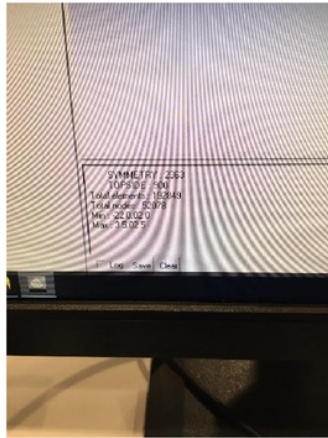
Refine Simulation: As mentioned above the mesh will be changed just to investigate how the flow around the rear of the MIRA car model, if need be the standard MIRA can have the exact same mesh applied to see how the adaptation to the MIRA model changes how the airflow moves around the added component.

The below picture shows the ANSYS workbench which is an overview panel in which all the different stages can be accessed and view, ticks signify the stage is complete. The second picture shows the pressure tap data from the MIRA.

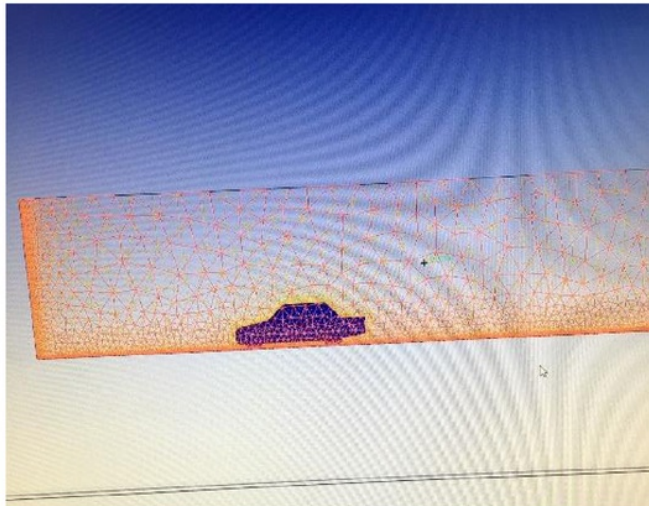


Results

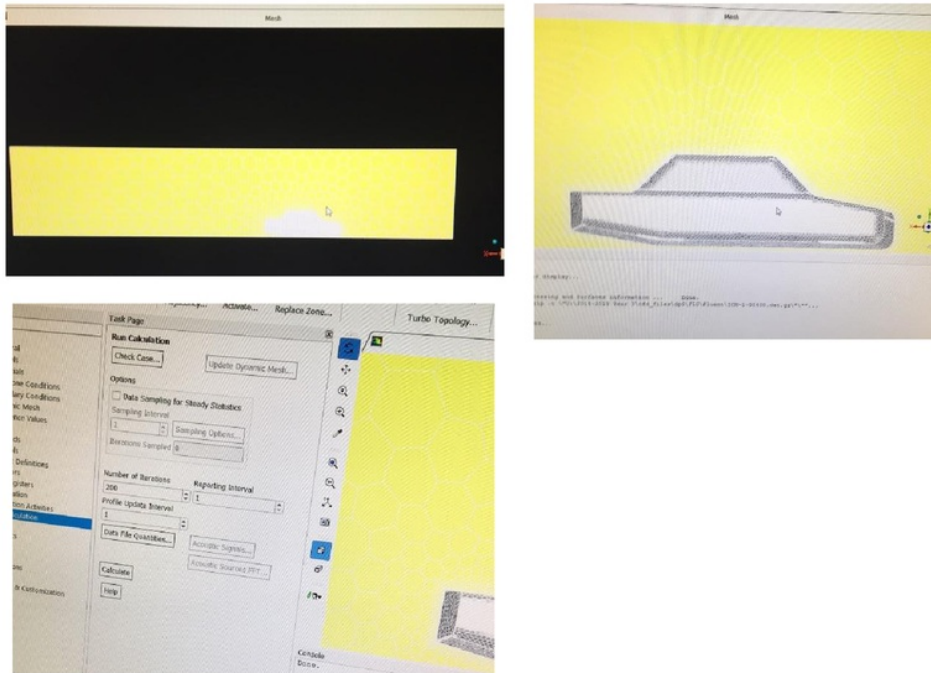
1. The below pictures of mesh details and the new adaptation. The new adaptation was chosen to be a small 'lip' spoiler. This lip spoiler is taken from real world examples and can help provide downforce and stability and high speeds to cars. It was chosen over a rear wing as the rear wing wouldn't direct airflow away from the sharp rear section.



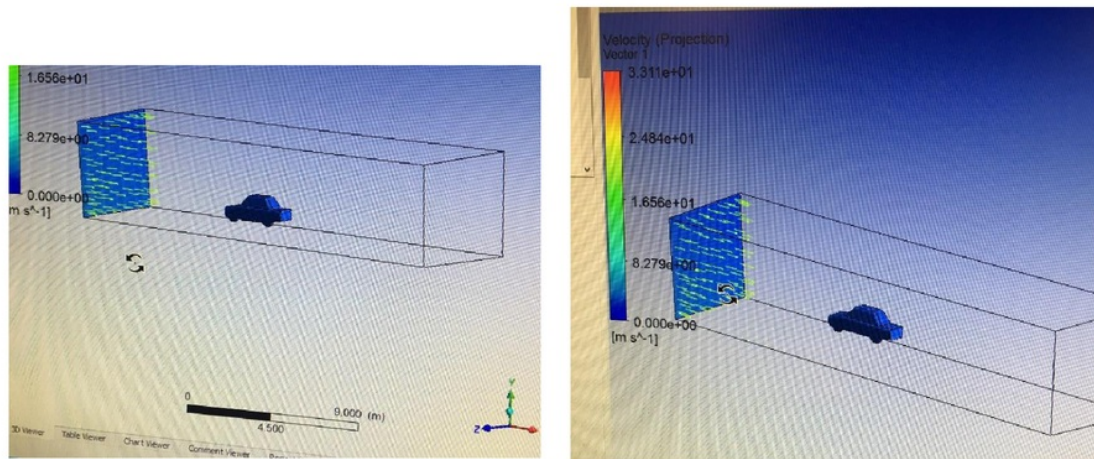
2. The picture below shows the calculated mesh around the MIRA car model. As seen below the mesh is very dense around the MIRA car model.



3. The collection of pictures below shows the MIRA car model in Fluent (a program part of ANSYS) that will run the simulation on the MIRA car model. The settings are displayed below, and the next stage will be looking at the results of this simulation.



4. The below pictures show the results from fluent. This package shows the data and air flow over and around the MIRA car model.



Conclusions

Even though the investigation is unfinished, a suitable testing method was established, and some data was collected along with important information. The MIRA car model behaved as expected with the adaptation, making a big change to the airflow at the rear of the MIRA model. Due to the square rear shape before, lots of turbulence and build up was created as the edge cut off, with the lip spoiler sending airflow up and over the square rear section.

Given more time a deeper analysis of the results and possibly more simulations run the investigation could be more informative and reflect the work carried out during this assignment.

References

Figure 1 - <https://www.ansys.com/about-ansys/news-center>

All pictures taken from ANSYS 19.2 and 16.2