

University of Wales Trinity Saint David

School of Engineering

Computational Fluid Dynamics

Robert Ogden

P140899

Simulation and Modelling (M7X01380)

The aim of the work carried out in the process of this assignment was to create a three dimensional simulation model of the simplified external geometry of a car on which an investigation into the car geometries aerodynamics could be carried out. Although the assignment brief stated that the simulation model should use an Ahmed body, a simplified car model developed for aerodynamic investigation, the simplified car geometry on which this investigation is based is the MIRAESTATE geometry provided as one of the alternative geometries for study. The aerodynamic simulation was to be carried out using Ansys 16.2 computer software to produce a study, of multiple run throughs, with iterative improvement if necessary to produce the most accurate results possible. It would be difficult to verify the results of the study for the MIRAESTATE geometry used as there are no previous studies available for this geometry which could be referenced, however the simulation model could be replicated using other geometry, such as one of the Ahmed bodies which have been extensively studied and so the simulation can be repeated using these geometries to verify whether the study should theoretically provide accurate results.

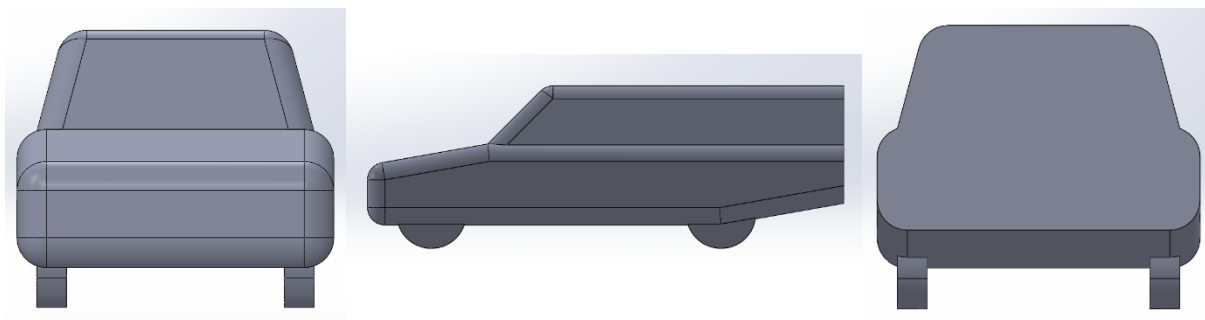


Figure 1 – MIRAESTATE Geometry

The first part in the development of the three dimensional simulation model for the aerodynamic study is to create the geometry which will be studied. This involves the importing of the MIRAESTATE model for study and the creation of a domain for the study around the model. Due to the MIRAESTATE being a simplified car model, travelling straight ahead, such that neither wheel is turned, the model is symmetrical and so therefore the study model and domain can effectively be cut in half in order to reduce the overall domain size, allowing the subsequent mesh to have a much higher element density without compromising the effectiveness of the study or the overall ability of the computer used to process the study. The MIRAESTATE geometry could therefore be subtracted from the overall domain to produce a subtracted negative of the MIRAESTATE vehicle within the domain for study.

The domain itself must be much larger than the MIRAESTATE geometry on which the study is based, such that it can encapsulate all of the interference of the MIRAESTATE geometry on the airflow around it. Due to the movement of the air around the MIRAESTATE geometry there are several particular phenomena, which are also present in most aerodynamic studies on vehicles or associated bodies such as the Ahmed body, which can be observed. These aerodynamic phenomena are principally the area of increased pressure and compression of air at the front of the vehicle, the compression across the top of the vehicle, the area of low pressure and turbulence in the wake of the vehicle and the areas of turbulence created by the wheels of the vehicle.

It is important that the domain fully encapsulates these phenomena as these are the principal causes of lift and drag on the vehicle, therefore the size of the domain is critical. The inlet of the domain must be far enough ahead of the vehicle that it starts before the compression of the air caused by the vehicle and it must be tall enough that it is clear of the interference caused by the air passing over the top of the vehicle. The floor of the domain is in contact with the wheels of the

MIRAESTATE geometry, as it would be when the vehicle is travelling on a road, lastly the domain must be long enough that the area of low pressure and turbulence in the wake of the vehicle has fully normalised before the outlet at the end of the domain.

The various external faces of the domain geometry are also given as named selections within the geometry. Doing this when creating the geometry for the study defines the velocity inlet and outlet for the airflow as well as defining the floor as a solid boundary, the outer face and topside as areas of ambient pressure and the inner face with the MIRAESTATE geometry as being the face of symmetry, for use later when the mesh for the study is generated.

The next part in the development of the three dimensional simulation for the study is to generate a mesh with which to perform the study. In terms of computer simulations, a mesh is a series of tetrahedrons, which fill the volume of the domain, dividing the domain up into numerous smaller elements, with each point of the tetrahedron referred to as a node. The division of the domain into smaller elements allows the movement of the fluid through the domain to be monitored as each element records where on the tetrahedron a fluid particle enters and exits, allowing the movement of the particle through the domain to be tracked. The density of the mesh is important to the accuracy of the results of the study being carried out as the denser the mesh, the more accurately the flow of the fluid can be followed as there will be many more data points per unit area when compared to a less densely populated mesh.

However the denser the mesh, the more elements which need to be processed and so therefore the more computational processing power and time would be required to run the study, meaning that the density of the mesh will always be a trade-off of mesh density and accuracy to available computational power and time. It is therefore necessary to prioritise areas where a higher density mesh would be advantageous to the study, for example in the case of the study carried out here, a denser mesh is advantageous in the areas around the wheels and in the wake of the MIRAESTATE vehicle geometry where strong vortices are generated, which can affect the output values of the study. Whereas towards the outer edges of the domain, such as the area immediately before the outlet vent, the flow of the fluid has almost normalised from the interference of the vehicle geometry and so to place a dense mesh in this area wouldn't be an effective use of processing power as the fluid is mostly moving in a straight line and so multiple elements wouldn't be required to follow the movement of the fluid particles. It is for this reason that for the study carried out on the MIRAESTATE geometry, the volume meshing sizing function was set to be related to 'proximity and curvature' relative to the MIRAESTATE geometry such that the mesh density is higher close to the MIRAESTATE geometry and particularly the curved areas of the geometry. Within the ICFM CFD programme used for the meshing of the domain, there is the option to place boxes to increase the density of the mesh within the volume of the box, either by adjusting the element sizing options within that box or by specifying a number of elements to be fitted within the volume of the box. These boxes of increased density allow for the mesh to be refined to study certain areas of the overall study domain in increased depth. For the study of the MIRAESTATE geometry there were three such boxes placed, one at the front of the vehicle, to monitor the compressive effect of the vehicle geometry on the fluid, one at the rear of the vehicle geometry to monitor the vortices generated in the wake of the vehicle geometry and one underneath the length of the vehicle geometry to observe the effects of the wheels and the general airflow under the vehicle. The increased density boxes under the vehicle geometry and in the wake of the vehicle will feature a small overlap due to the area of turbulence emerging from under the car extending into the wake of the vehicle geometry therefore it would be necessary for the density box to be overlapping with the box monitoring the vortices in the area of turbulence in the wake of the vehicle. There is also an area of overlap between the density box monitoring compression at the front of the vehicle and the box

underneath, however due to their positions relative to the effects they monitor, the overlap of the boxes is more a case of coincidence than design as their overlap does not aid the results of the study in the same way that the overlap of the box under the vehicle and the box in the wake of the vehicle does due to their effects not affecting the phenomena observed in the other box in quite the same way.

Before proceeding to the carrying out of the study, the final step in the mesh development process was to check the skewness, orthogonal quality and aspect ratio of the 4.02 million elements in the mesh. The skewness of the mesh is defined as being the average how far from being an equilateral triangle each element side is, with the answer being ideally as close to 0 as possible, meaning that every side of the tetrahedrons throughout the mesh is an equilateral triangle, however given the requirements of the mesh (different element sizes, geometric transitions etc) and the computing power available this is often difficult or impossible to achieve, the final mesh used in the MIRAESTATE study had a skewness of 0.16. The orthogonal quality of the mesh is the average across the mesh of the cos angles between the centre of the cell and the normal to each face divided by the cos angle of the centre of the cell and the adjacent cell, with the ideal answer being as close to 1 as possible, in the case of the final mesh used the orthogonal quality was 0.89. The aspect ratio of the mesh is the ratio of the face length of the largest element divided by the face length of smallest element. Ideally this would be 1, however due to computing restrictions, it is necessary to refine the mesh to prioritise areas of importance to the study, so as such the aspect ratio in the study was 3.96.

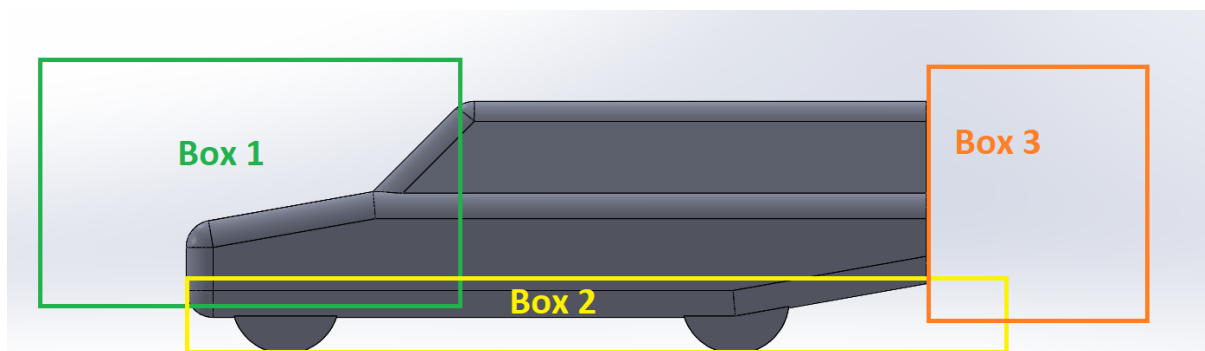


Figure 2 – Density box placements

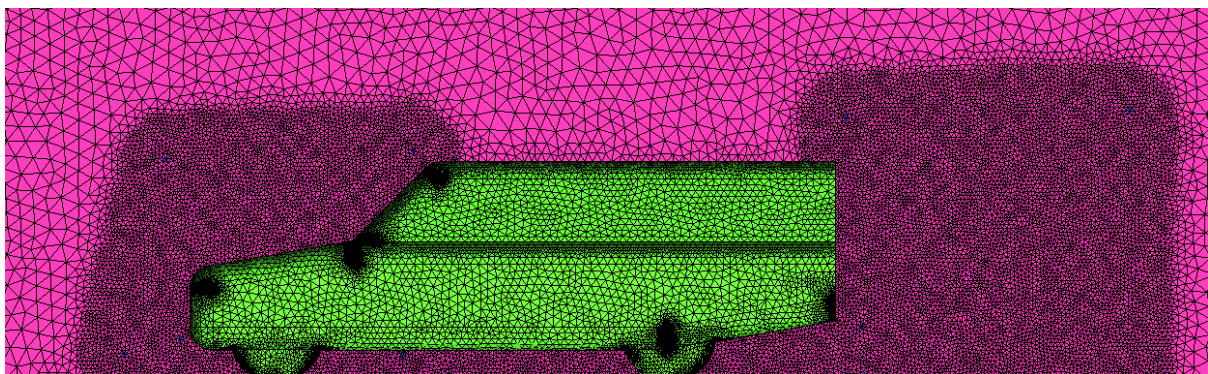


Figure 3 – Final mesh surrounding MIRAESTATE geometry

Once the mesh for the study was finalised, the final part of the study was to use the Fluent programme within Ansys to process the study and compute the fluid flow around the MIRAESTATE geometry. Firstly the materials were defined such that the solid, the MIRAESTATE geometry, is made of aluminium and the fluid which will be passing through the volume of the domain is air. The specification states that the vehicle being studied is travelling at a velocity of 40 metres per second,

therefore under the velocity inlet settings the fluid velocity was set as 40m/s in the positive X direction. It is also important that the outlet vent is set to ambient pressure, which is done by setting the pressure at the outlet vent to 0 pascal (gauge) such that the flow is consistent. It is also important to set all walls, topside, symmetry face, outer face and floor to being no slip surfaces such that if the flow meets the wall, it is not slowed by the friction of the face, due to these faces being purely theoretical, with the exception of the floor, however this also helps to aid the adherence to reality as the simulation is to simulate the vehicle moving through the air as opposed to the air moving past the vehicle as in the simulation. The final set up variable used in fluent was the viscous or turbulence model used. There are many options available however from other studies on automotive aerodynamics and studies on the Ahmed body, K Epsilon Realizable and SST K Omega are the two which are most used, with the K Epsilon Realizable turbulence model being the most widely used for this application, being good for bodies made up of both flat and rounded surfaces, such as the Ahmed body and the MIRAESTATE geometry in this study. However the SST K Omega turbulence model is according to other studies, better than the K Epsilon Realizable turbulence model for near wall interactions, which in the case of the Ahmed body and the MIRAESTATE geometry is useful for the area around the wheels between the bottom of the vehicle and the floor as well as for simulations where strong vortices are present, such as the area in the wake of the vehicle, where a large vortex appeared from the underside of the vehicle. From a previous study performed on the effect of the rear slant angle of the Ahmed body, the SST K Omega turbulence model provided the investigation with the closest values obtained to those of experimental data.

When the study was run using the fluent solver, set to coupled and the K Epsilon Realizable turbulence model, the values obtained for the coefficients of lift and drag were -0.29 and 0.41 respectively. However, when the simulation model was used for the 25 degree slant Ahmed model, the results were not concordant with those obtained from experimental data. Therefore, both the simulations using the MIRAESTATE and Ahmed bodies were run again using the SST K Omega turbulence model, with the MIRAESTATE returning coefficients of lift and drag of -0.22 and 0.28 respectively and the Ahmed body returning concordant results to the experimental data provided (Cl 0.36 and Cd 0.28). With the Ahmed body simulation providing concordant results to experimental data, it would therefore be most likely to say that the simulation of the MIRAESTATE geometry using the SST K Omega turbulence model is the more accurate one, and that the coefficient of lift for the MIRAESTATE geometry is approximately -0.22 and the coefficient of drag 0.28.

The results and meshing decisions are further supported when post processing is used to analyse the data obtained. With the areas of compressive pressure at the front of the vehicle geometry and the vortices produced at the rear of the vehicle being clear.

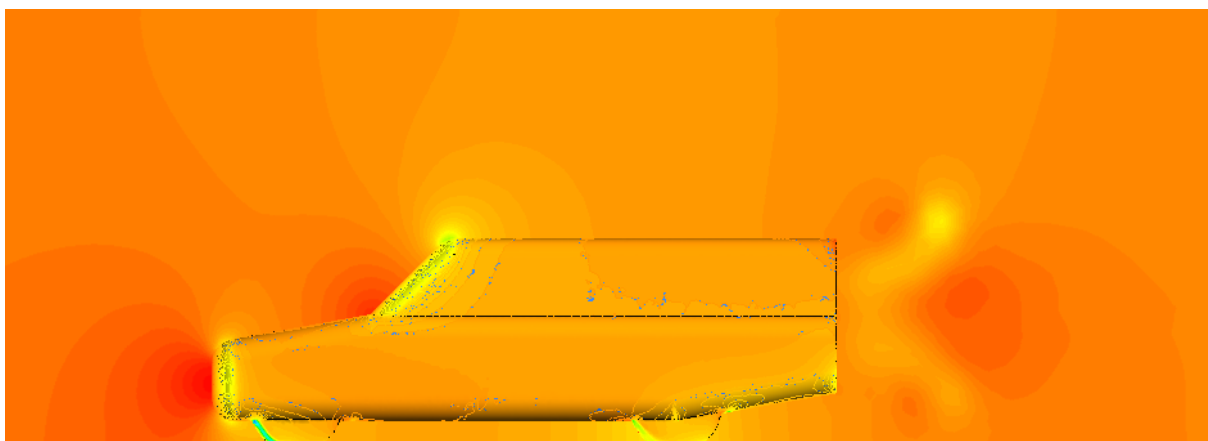


Figure 4 – Contour plot of static pressure

Figure 4 shows a contour plot of static pressure around the MIRAESTATE geometry, with the expected area of high pressure caused by the flow hitting the front of the vehicle geometry clearly visible, as well as the area of low pressure, followed by high pressure in the wake of the vehicle where the vortices caused by the vehicles interference are occurring.

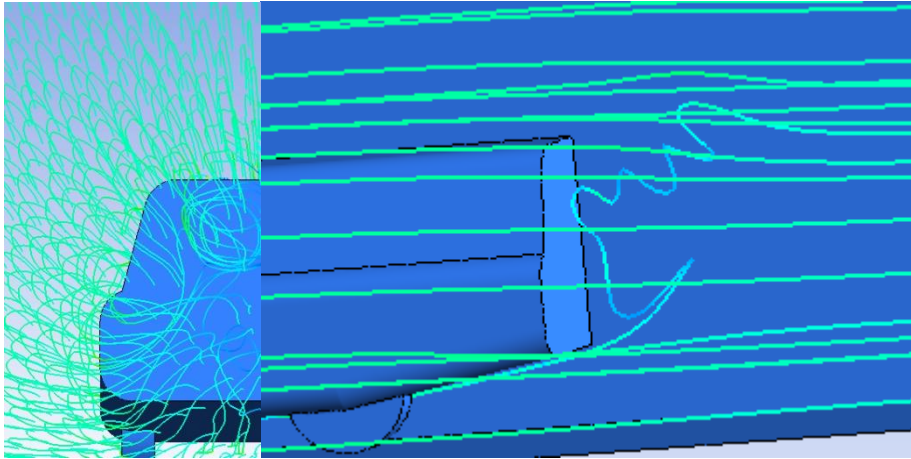


Figure 5 – Flow lines showing vortices

Figure 6 shows a plot of the flow lines around the MIRAESTATE geometry and in particular the rear of the vehicle, with large vortices emerging from the slanted rear side, due to the area of low pressure caused by the vehicle geometry. The vortices caused by the slanted underside of the vehicle geometry and are similar to a reversal of those caused by the top edge of the 25 degree Ahmed body, these vortices emanate upwards to affect the region above the vehicle geometry.

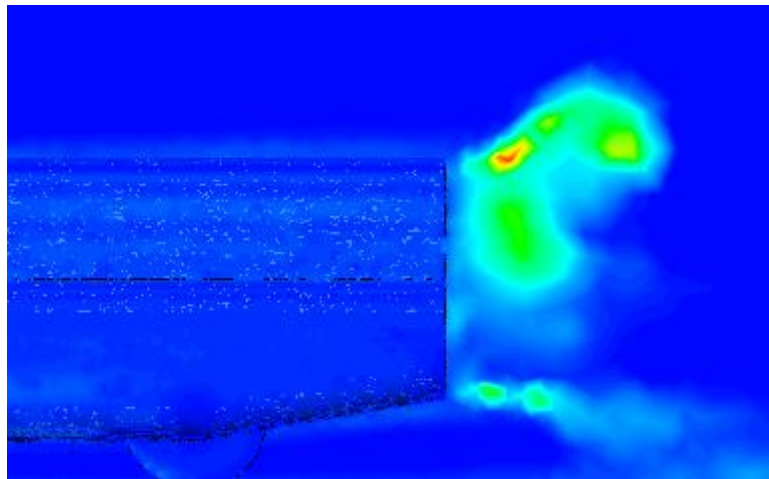


Figure 6 – Contour plot showing eddy current viscosity

Figure 6 shows a contour plot of the eddy current viscosity to the rear of the vehicle, with the largest area emerging from the top edge of the vehicle geometry, which concurs with the strong vortices seen in figure 5.

Table of figures:

Figure 2 – Density box placements, Page 4

Figure 3 – Final mesh surrounding MIRAESTATE geometry, Page 4

Figure 4 – Contour plot of static pressure, Page 5

Figure 5 – Flow lines showing vortices, Page 5

Figure 6 – Contour plot showing eddy current viscosity, Page 7

Sources:

RBF Morph - Shape optimization of the MIRA Reference CAR, 2010, (Accessed 19/4/19 at:

<http://www.rbf-morph.com/shape-optimization-of-the-mira-reference-car/>)

X.W. Wang, Y. Zhou, Y.F. Pin, T.L. Chan - Turbulent near wake of an Ahmed vehicle model, Exp. Fluids 54, P1490-1508, 2013

H. Ahmed, S. Chacko – Computational Optimization of Vehicle Aerodynamics, Annals & Proceedings of DAAAM International 2012, Volume 23, P313-318, 2012

H. Lienhart, C. Stoots, S. Becker - Flow and Turbulence Structures in the Wake of a Simplified Car Model (Ahmed Model), 2002

H. K. Versteeg, W. Malalasekera - An Introduction to Computational Fluid Dynamics: The Finite Volume Method. Pearson Education Limited. ISBN 9780131274983, 2007

S.R. Ahmed, G. Ramm - Some Salient Features of the Time-Averaged Ground Vehicle Wake, SAE-Paper 840300, 1984

R. Ogden – Assignment submission for Advanced Computational Methods (M6X00408), 2018