

A Parametric Investigation to Vehicle Geometry Features on External Aerodynamic Performance

LUKE CRIPPS

Abstract

In the world of today, fuel economy is becoming a more important factor through the development of new vehicles. Increase drag force acting on the front on vehicles is one massive area in which improvements can be made. The different vehicle shapes available for the designers of today, including; notchback; fastback; pickup; and estate, will be tested against their aerodynamic characteristics that are produced. As well as this, the effectiveness of aerodynamic devices: an Air Dam and a rear spoiler will be investigated. Using computational fluid dynamics, a range of tests will be designed and investigated to evaluate the different vehicle shapes and aerodynamic devices, starting with a base setup case study, investigating the effects of using alternative mesh sizes, different turbulence models, as well as using different dimensions for the size of the domain. Each of the vehicle shapes produced different values for coefficient of drag and coefficient of lift. Attaching an Air Dam to the front of the different vehicles proved to have mixed results, with only the largest size of Air Dam reducing the coefficient of drag. When performing the investigations for the spoiler, it was clear that the main role of the device was to amplify the coefficient of lift, therefore, improving the downforce of the vehicle. Consequently meaning the coefficient of drag of the vehicle would increase. With the results of the testing verified, the fastback shape provided the lowest coefficient of drag out of the vehicle shapes, with both of the aerodynamic devices showing not to be very effective at lowering the coefficient of drag.

Table of Contents

Abstract.....	1
1.0 Introduction.....	5
2.0 Research	6
2.1 Background.....	6
2.1.1 Relevance to Industry	6
2.1.2 Aerodynamic Fundamentals	7
2.2 Current Studies.....	9
2.2.1 Aerodynamic Studies and Vehicles	9
2.2.2 MIRA Reference Car Studies	12
2.3 Methods.....	12
2.3.1 CFD Process.....	12
2.3.2 CFD Methods for External Aerodynamics.....	13
2.3.3 Validation Case.....	13
3.0 MIRA Reference Geometry with Validation.....	14
3.1 Base Car Setup	14
3.2 Investigations	20
3.2.1 Domain Size.....	20
3.2.2 Mesh Size	21
3.2.3 Turbulence Model.....	23
3.3 Fully Defined Model.....	25
4.0 Vehicle Geometry Investigations.....	25
4.1 Vehicle Shape.....	25
4.2 Air Dam	28
4.2.1 Lift	29
4.2.2 Drag.....	31
4.2.3 Centre of Pressure	32
4.3 Spoiler	33
4.3.1 Lift	34
4.3.2 Drag.....	36
4.3.3 Centre of Pressure	37
5.0 Conclusion	37
6.0 Evaluation.....	37
7.0 Bibliography.....	38
8.0 Appendix.....	40

Figure 1	A graph showing the correlation between the coefficient of drag and fuel economy for the new generations of the Honda Jazz	6
Figure 2	A graph showing the correlation between fuel economy and carbon dioxide production	6
Figure 3	The Ahmed geometry model	9
Figure 4	The NRCC/SAE geometry model	10
Figure 5	The Rover reference geometry	10
Figure 6	The Pininfarina reference geometry	11
Figure 7	Two configurations of the MIRA reference car	11
Figure 8	A flow chart illustrating the process of using CFD	12
Figure 9	Various graphs illustrating how splitter sizes affects coefficient of drag and lift values	13
Figure 10	A Graph illustrating coefficient of drag values against different assemblies	13
Figure 11	All four sections needed for the CFD process	14
Figure 12	The domain used for the CFD process, with the dimensions highlighted in black	14
Figure 13	Inserting the bolleen into the geometry	15
Figure 14	Selecting the faces of the model	15
Figure 15	The various settings used through the meshing process	16
Figure 16	The settings used to generate prism layers around intended areas	16
Figure 17	The volume mesh generated throughout the geometry	17
Figure 18	The various settings that need to be set up before starting the fluent process	18
Figure 19	Data gathered by Angelina I. Heft	19
Figure 20	Data gathered by V. Naveen Kumar	19
Figure 21	A visual representation of the air flow around the model	19
Figure 22	The coefficient of drag and lift results for the tests changing the domain size	20
Figure 23	The turbulent air flow visible in domain size test 1	21
Figure 24	The turbulent air flow visible in domain size test 12	21
Figure 25	The coefficient of drag and lift results for the different mesh sizes	22
Figure 26	A visual representation of changing the mesh sizes	22
Figure 27	The coefficient of drag and lift results for changing the turbulence model	23
Figure 28	A visual representation of the change in velocity of the air flow around the model	23
Figure 29	A visual representation of the change in turbulent air flow around the model; top left illustrates the K-Epsilon turbulence model; top right illustrates the K-Omega turbulence model; with the bottom ones representing the Transition SST turbulence model	24
Figure 30	Cad model of the estate model variant	25
Figure 31	Cad model of the notchback model variant	25
Figure 32	Cad model of the fastback model variant	25
Figure 33	Cad model of the pickup model variant	25
Figure 34	The coefficient of drag and lift results for the different vehicle shapes	26
Figure 35	A visual representation of the wake zone created at the rear of the model; top left notchback; top right fastback; bottom left pickup; bottom right estate	26
Figure 36	A visual representation of the turbulent air flow around the vehicle; top left notchback; top right pickup; bottom left fastback; bottom right estate	27
Figure 37	An example of an Air Dam fitted to the notchback model	28
Figure 38	A graph illustrating the change in coefficient of lift against the four model variants with changing length of air dam	29

Figure 39	A visual representation of the wake zone created by attaching a 200mm air dam to the front of the vehicle. Fastback model shown	29
Figure 40	A visual representation of the wake zone created by attaching a 300mm air dam to the front of the vehicle. Fastback model shown	29
Figure 41	A visual representation of the turbulent air flow created by attaching a 200mm air dam to the front of the vehicle. Fastback model shown	30
Figure 42	A visual representation of the turbulent air flow created by attaching a 300mm air dam to the front of the vehicle. Fastback model shown	30
Figure 43	A visual representation of the velocity of the air flow around the vehicle when a 200mm air dam is attached. Fastback model shown	30
Figure 44	A visual representation of the velocity of the air flow around the vehicle when a 300mm air dam is attached. Fastback model shown	30
Figure 45	A graph showing the change in coefficient of drag for the different vehicle shapes with changing air dam lengths	31
Figure 46	A visual representation of the turbulent air flow around the estate vehicle with 0mm air dam	31
Figure 47	A visual representation of the turbulent air flow around the estate vehicle with 100mm air dam	31
Figure 48	A visual representation of the turbulent air flow around the estate vehicle with 200mm air dam	32
Figure 49	A visual representation of the turbulent air flow around the estate vehicle with 300mm air dam	32
Figure 50	Measuring the distance between the rear of the vehicle to the rear axle. Pickup model shown	32
Figure 51	An example of the spoiler used fitted to the notchback model	33
Figure 52	A graph showing the change in coefficient of lift for changing height of spoiler for the different vehicle shapes	34
Figure 53	A visual representation of high pressure forming in front of the spoiler. Estate model shown	35
Figure 54	A visual representation of the pressure distribution around the fastback model with 50mm spoiler	35
Figure 55	A visual representation of the pressure distribution around the fastback model with 100mm spoiler	35
Figure 56	A graph showing the change in coefficient of drag for changing heights of spoiler for different vehicle shapes	36
Figure 57	A visual representation of the direction of the air flow around the pickup model, fitted with 150mm spoiler	36
Figure 58	A visual representation of the direction of the air flow around the estate model, fitted with 150mm spoiler	36

1.0 Introduction

Vehicle shapes have a massive effect on the drag and lift characteristics of a car, due to the external aerodynamic forces that are acting on it. These characteristics are becoming more crucial in the design of vehicles because of the stricter measures placed on carbon dioxide emissions, and other toxic compounds produced by the internal combustion engine, through the Euro six and the fore coming Euro 7 standards, as described by RAC, 2018. This is due to the drag being a factor in a vehicles fuel economy, meaning that an improvement in fuel economy would reduce the emissions produced as less fuel would be needed to be used in the combustion process, documented by Keith Douglass, N.D.

The most basic car shapes used in the automotive industry would be; estate; fastback; notch and pickup. Each shape influences the coefficient of drag produced by the vehicle. Below are some examples of coefficient of drag values for different cars, using a variety of vehicle shapes, provided by Angelina I. Heft, 2012:

	Estate Back	Notchback	Fastback
Audi	0.288	0.260	0.259 ²
BMW	0.257	0.248	- ³
DrivAer	0.319	0.277	0.275

Table 1 - Coefficient of drag values for a variety of vehicle shapes

Vehicle shape is not the only feature that affects the drag and lift of a vehicle. Aerodynamic devices also play a role in the overall effect of the drag and lift characteristics. Splitters and rear wing spoilers, just to name a few, are some of these devices that can affect these characteristics, as stated by Renan Francisco Soares, 2018.

The overall aims looking ahead are:

- To investigate how the difference in vehicle shape affects the coefficient of drag and coefficient of lift.
- To investigate the impact, positive or negative, of adding aerodynamic devices to the vehicle in response to the aerodynamic characteristics, including coefficient of lift, coefficient of drag and centre of pressure.

The aims will be investigated using computational fluid dynamics (CFD). Graphs for the coefficient of drag and lift will be produced for each variant of the vehicle.

2.0 Research

2.1 Background

2.1.1 Relevance to Industry

One major factor why the automotive industry is interested into looking at reducing the drag acting on a vehicle is that drag has an influence on fuel economy. If a vehicle has a high coefficient of drag, then the fuel economy of that vehicle will be worse when compared to another car with a lower coefficient of drag. An example of this would be the Honda Jazz. The graph below shows how when the coefficient of drag is improved, so does the fuel economy. Figure produced by Kentaro Machida, 2015:

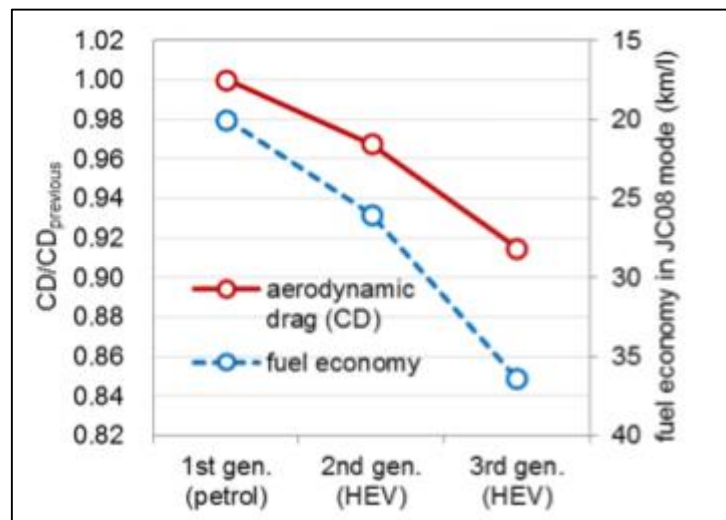


Figure 1 - A graph showing the correlation between the coefficient of drag and fuel economy for the new generations of the Honda Jazz

This increase in fuel economy has a link to the carbon dioxide emissions produced by a vehicle. With an increase in fuel economy, a reduction in the carbon dioxide produced is the result, as illustrated in the following graph, produced by Keith Douglass, N.D.

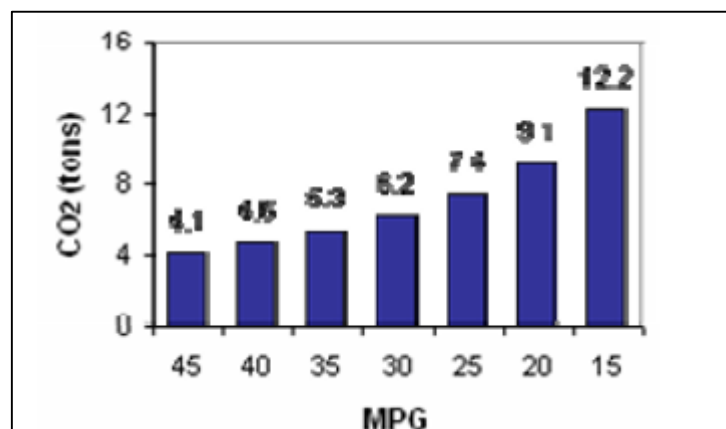


Figure 2 - A graph showing the correlation between fuel economy and carbon dioxide production

2.1.2 Aerodynamic Fundamentals

Bernoulli's Equation, as stated by Princeton, N.D:

$$p + \frac{1}{2}\rho V^2 + \rho gh = \text{constant}$$

p = Pressure

ρ = Density

V^2 = Velocity

g = Gravitational Constant

h = Elevation

Some assumptions are made, according to Katz, 2016:

- The points where the pressure is acting lie on a streamline
- The viscosity of the fluid used is constant
- Friction is not observable in the system

Reynolds number, as stated by Nuclear Power, N.D, is the ratio of inertial forces to viscous forces. It is an adequate parameter for predicting if a flow condition will be laminar or turbulent.

$$RE = \frac{\rho \times V \times D}{\mu}$$

RE = Reynolds Number

ρ = Density of fluid

V = Velocity of fluid

D = Diameter of fluid

μ = Viscosity of fluid

Engineering Toolbox, 2003

Boundary Layers

As a vehicle moves through a fluid, the molecules of air near the surface of the vehicle start to move around as they have been disturbed. Aerodynamic forces are therefore generated between the fluid and the surface of the vehicle, as declared by Hall, 2015. The fluid closest to the surface of the vehicle will have no velocity. As the layers of fluid travel further away from the surface, the velocity of the fluid will increase, as there will be less collisions between the molecules, as clarified by Technical F1 Dictionary, N.D. Once there are no collisions between the molecules, this is classed as the boundary layer. The viscosity and compressibility of the fluid are the two main factors in the position of the boundary layer.

Boundary layers can either be laminar or turbulent, dependent on the value for Reynolds number. A lower value for Reynolds number illustrates a laminar flow, where a higher Reynolds number would mean the opposite, a turbulent flow, as mentioned by Mclean, 2012.

Separation

Flow separation occurs when the boundary layer of a vehicle has travelled against an adverse pressure gradient for long enough that the velocity of the boundary layer when compared to the vehicle retards to zero, as defined by Clancy, 1975. The flow of the fluid detaches from the vehicle's surface and inherits the forms of eddies and vortices. This type of separation leads to increased drag affecting the vehicle's performance.

Boundary layer separation occurs when a boundary layer becomes separate from the surface of the vehicle, and becomes a broad wake, as explained by White, 2011. This type of separation occurs when the boundary layer closest to the surface of the vehicle has its direction of motion reversed, Wilcox, 2000.

Lift and Drag

Drag is a force that an object, vehicle is subjected to that provides a resistance to the direction of motion, as defined by Electropaedia, 2005. The drag force acts parallel to the motion of the fluid, meaning it will operate in the same direction.

Lift is a force that acts perpendicular to the flow of fluid that the vehicle is passing through, explained by Hall, 2015. As this lift force can act perpendicular, being a vector quantity, the force will either try to lift the vehicle up and work against gravity, or its magnitude will be in the other direction, acting as downforce to push the vehicle into the ground.

2.2 Current Studies

2.2.1 Aerodynamic Studies and Vehicles

Reference shapes

With stricter measures being placed on carbon dioxide production and having the link between the coefficient of drag and better fuel economy, as mentioned on page 3, the use of reference shapes to help improve the coefficient of drag for vehicles is becoming more vital. Using these reference shapes will mean the aerodynamic characteristics that either benefit or hinder gains in fuel economy and therefore emissions can be tested, and verified, as expressed by Angelina I. Heft, 2012.

There are numerous different vehicle geometry models that have been created to help in the investigation into reducing the coefficient of drag:

- Ahmed Model
 - NRCC/SAE Model
 - Rover Bluff Body Model
 - Pininfarina Reference Car
 - MIRA Reference Car
- Geoffrey M. Le Good, 2004

The Ahmed geometry was originally developed for vehicle wake investigations and was designed in mind so when an experiment was conducted, there would be only one aerodynamic feature, as seen in the diagram below, that would be assessed, justified by Strachan R.K, 2007:

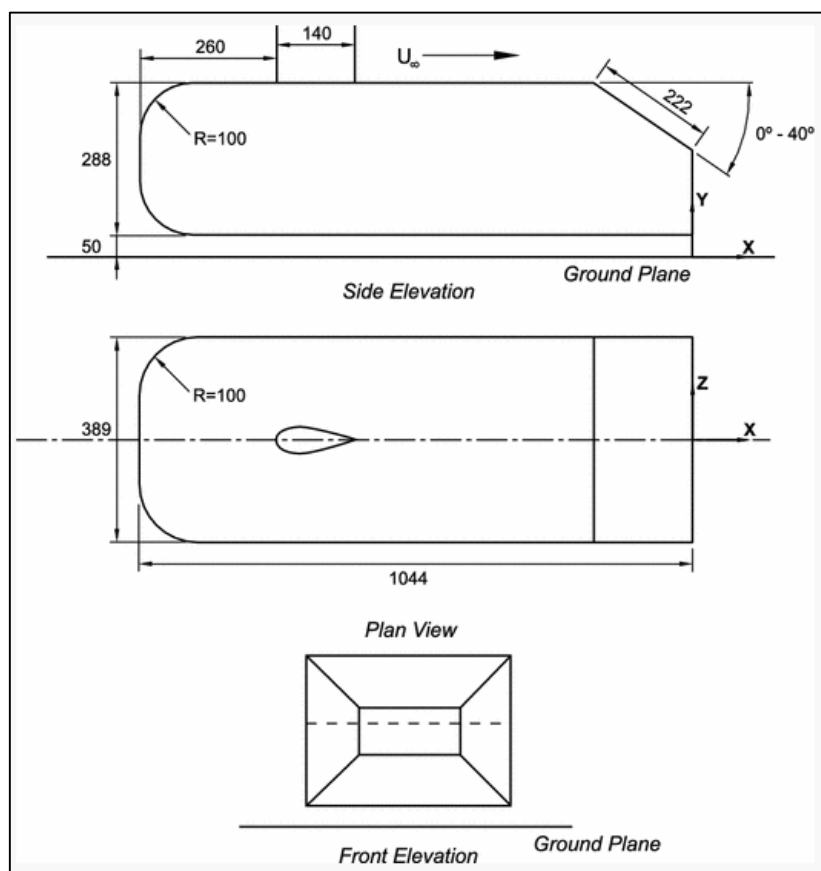


Figure 3 - The Ahmed geometry model

NRCC geometry was configured to exhibit attached or separated flows on American cars and trucks. This geometry, amongst others, was used to investigate the effect of aerodynamic drag against longitudinal position inside a wind tunnel, tests carried out by Garry K.P, 1994. Below is a diagram of the NRCC reference geometry, provided by D Wood, 2014:

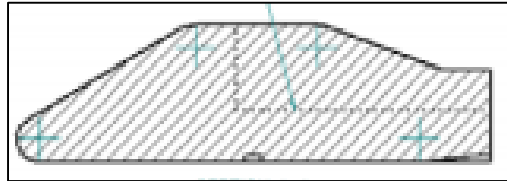


Figure 4 - The NRCC/SAE geometry model

The rover model was designed to assist in investigations on how changing vehicle shapes would affect the aerodynamic forces, as explained by Geoffrey M. Le Good, 2004. The Rover geometry has interchangeable slits at the rear so that the angle of slope can be changed. Below is a diagram of the Rover reference geometry, shown by Howell, 1994:

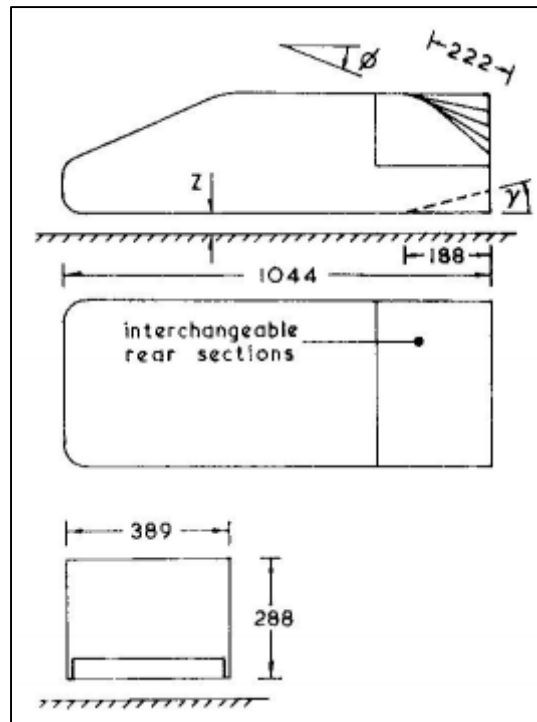


Figure 5 - The Rover reference geometry

The Pininfarina reference geometry, is the first of the models to have wheels included in the design. This means that results taken from using this reference geometry would be more accurate, due to interaction between the underside of the car being referenced as well, as detailed by Geoffrey M. Le Good, 2004. A illustration of the reference geometry is shown below, also provided by Geoffrey M. Le Good, 2004:

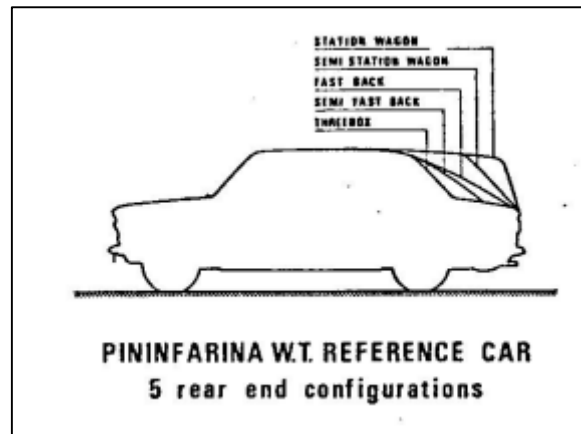


Figure 6 - The Pininfarina reference geometry

The MIRA reference car, the most well-known reference geometry in the automotive industry. It is used to calibrate the MIRA wind-tunnel every three months. It has been used for numerous investigations with relevance to wind-tunnel testing, and CFD investigations, as used by Yingchao Zhang, 2014. Below is a diagram of the MIRA reference car used in two configurations, created by Pruthviraj Mohanrao Palaskar, 2016:

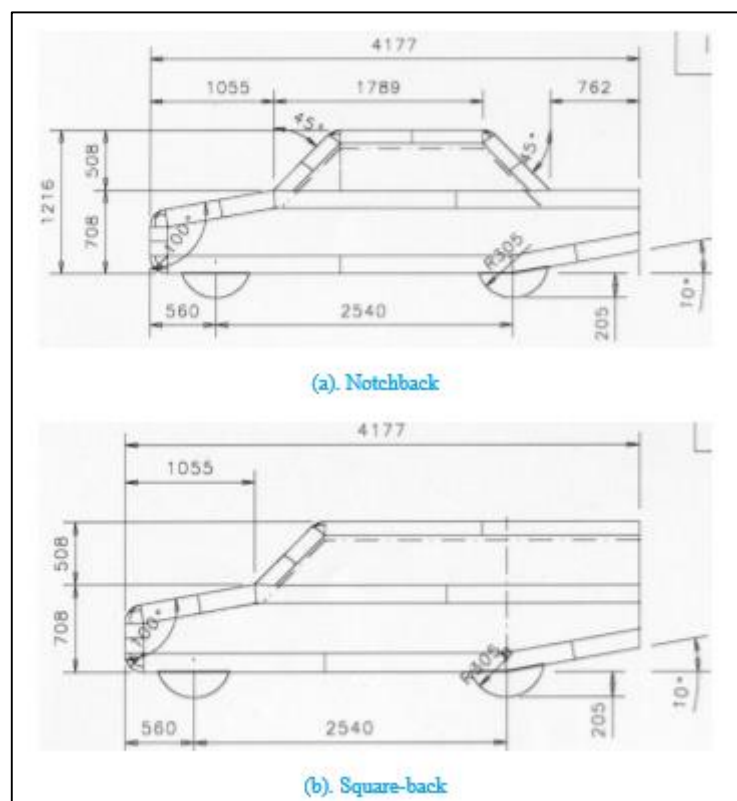


Figure 7 - Two configurations of the MIRA reference car

2.2.2 MIRA Reference Car Studies

As the MIRA reference car is the most used car geometry out of the ones talked about above, there are some studies out in the world where the need for the geometry is being used. Jeffrey Hoffman, 2001, has used the MIRA reference car to see if the coefficient of drag changes when different vehicle shapes are used. While this is not the primary aim of investigations completed by Hoffman, data has been gathered that will be useful during investigations into the aims mentioned above.

Another study carried out by Yingchao Zhang, 2014, investigates the optimum aerodynamic shape for a vehicle using the MIRA reference car as its reference geometry. All of the components of the vehicle were looked at and tested, to compare where most of the resistance, drag was created. Both papers produce data which can be used as a validation method for the aims of this investigation.

2.3 Methods

2.3.1 CFD Process

Computational fluid dynamics, CFD, has been used for many years in designing stage of a vehicle's development to evaluate its aerodynamic performance. This then leads to the design of the vehicle changing to minimize the aerodynamic drag resisting the car's movement, as expressed by Yingchao Zhang, 2014. The equations of fluid dynamics are the conservation laws of mass, momentum and energy. When using CFD, these conservation laws are called the Navier-Stokes equations, justified by Kronbichler, 2010. The programme needed to calculate the results for investigations is called Ansys. The following flow chart illustrates the general process for using the software to generate value for coefficient of lift and drag, inspiration provided from Francesco Mariani, 2016 and Stephane Cyr, 2011:

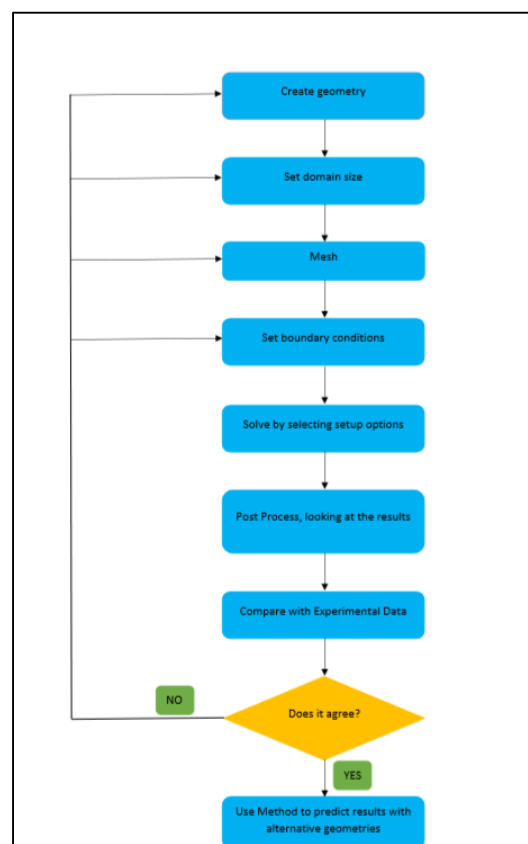


Figure 8 - A flow chart illustrating the process of using CFD

2.3.2 CFD Methods for External Aerodynamics

When looking at external aerodynamic features; diffusers; splitters; rear wings, a similar process is used to calculate results for the coefficient of lift and drag. Once the baseline study has been verified, the geometry is changed, to include one of the features stated above. For example, when testing a front splitter, the size of the splitter can make a difference to the drag generated, as shown in the graph below, created by Renan Francisco Soares, 2018:

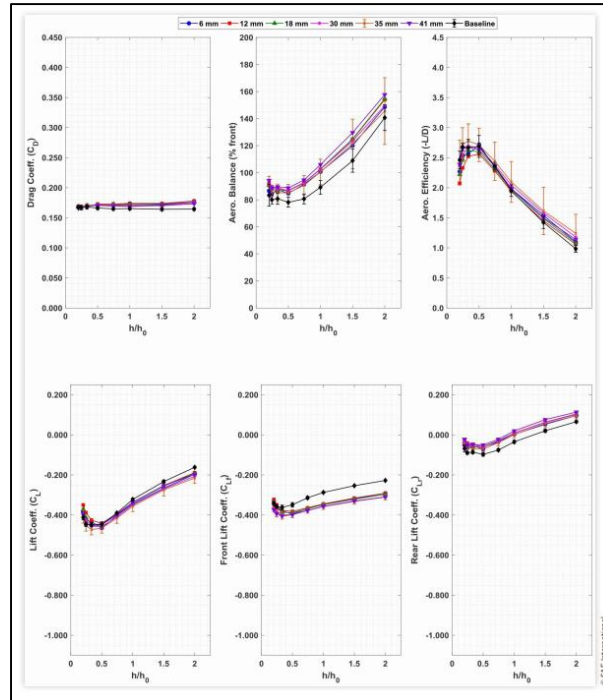


Figure 9 - Various graphs illustrating how splitter sizes affects coefficient of drag and lift values

2.3.3 Validation Case

In order to generate results, a validated baseline model needs to be generated. If the baseline study cannot be validated, then results cannot be used. The validation can be made using existing studies, or through experimental data. Once a baseline study has been made, so that the data can be verified, the results should be similar to ones shown below, produced by Renan Francisco Soares, 2015:

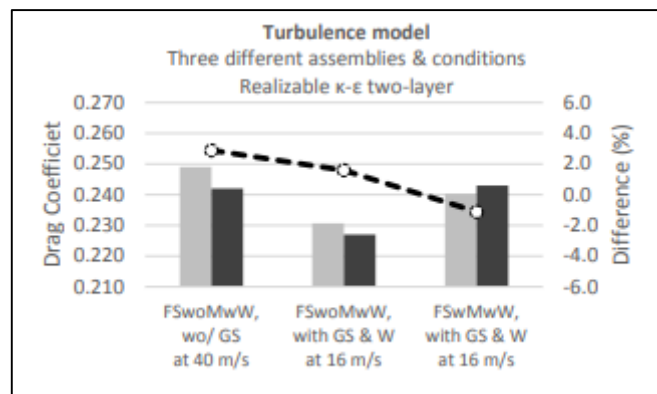


Figure 10 - A Graph illustrating coefficient of drag values against different assemblies

3.0 MIRA Reference Geometry with Validation

3.1 Base Car Setup

The first stage in the process is create the base setup for the car. The computer software Ansys is required to perform the CFD process. Once the software has been loaded up, Design Modeller, ICEM, Fluent and Results tab needs to be inserted into the worksheet, as seen in the diagram below:

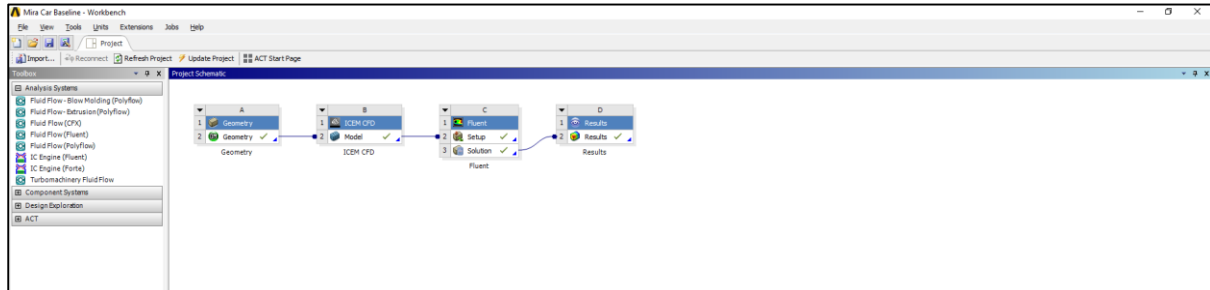


Figure 11 - All four sections needed for the CFD process

The first component needed to be configured is the geometry. Once the geometry tab has been opened, the first step is to import the geometry file into the document. Under the options tree that loads in with the geometry, the geometry needs to be simplified. Select the generate key to load the geometry into the work area. The next stage is to draw a rectangle around the geometry so that dimensions can be inserted to illustrate the size of the domain, using the modelling tool. To insert the dimensions, the dimension tab needs to be selected, with two horizontal, and two vertical dimensions placed in the positions shown in the diagram below, using the following measurements:

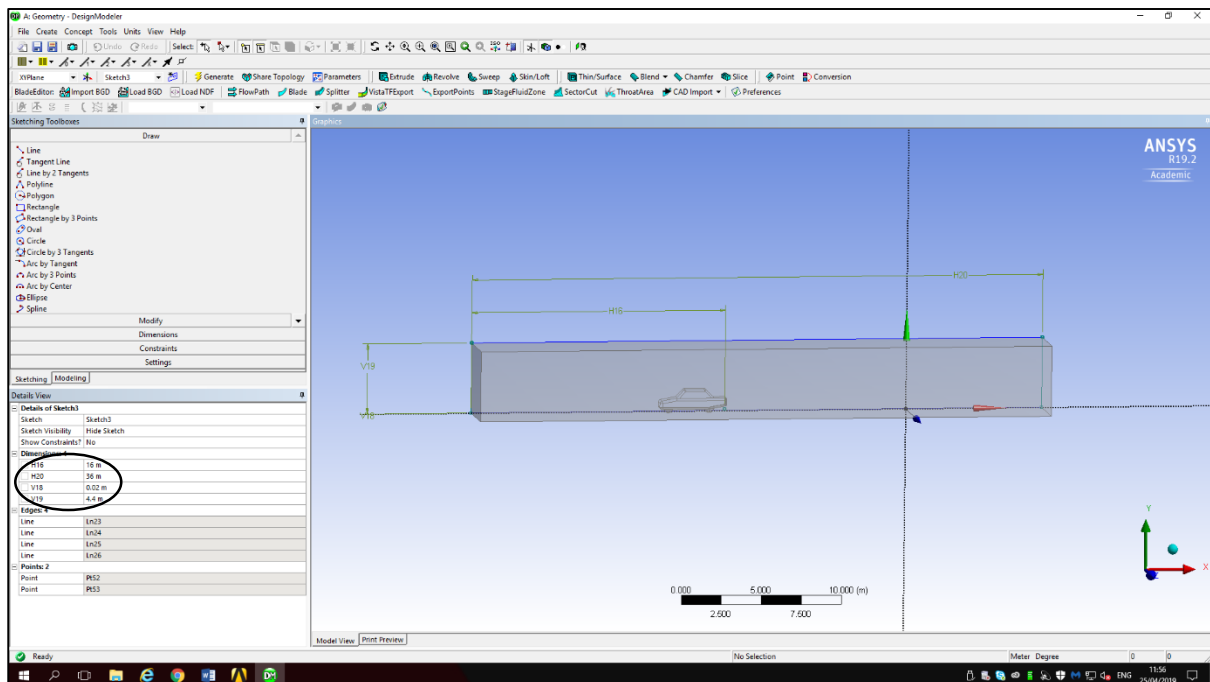


Figure 12 - The domain used for the CFD process, with the dimensions highlighted in black

The rectangle needs to be extruded, four meters in length, with the setting being changed to add frozen. Once all the settings for the extruding processing have been clarified, the generate function needs to be used to finish the process. Now a bolleen needs to be inserted between the geometry model and the extruded shape. The diagram below shows how the bolleen is created:

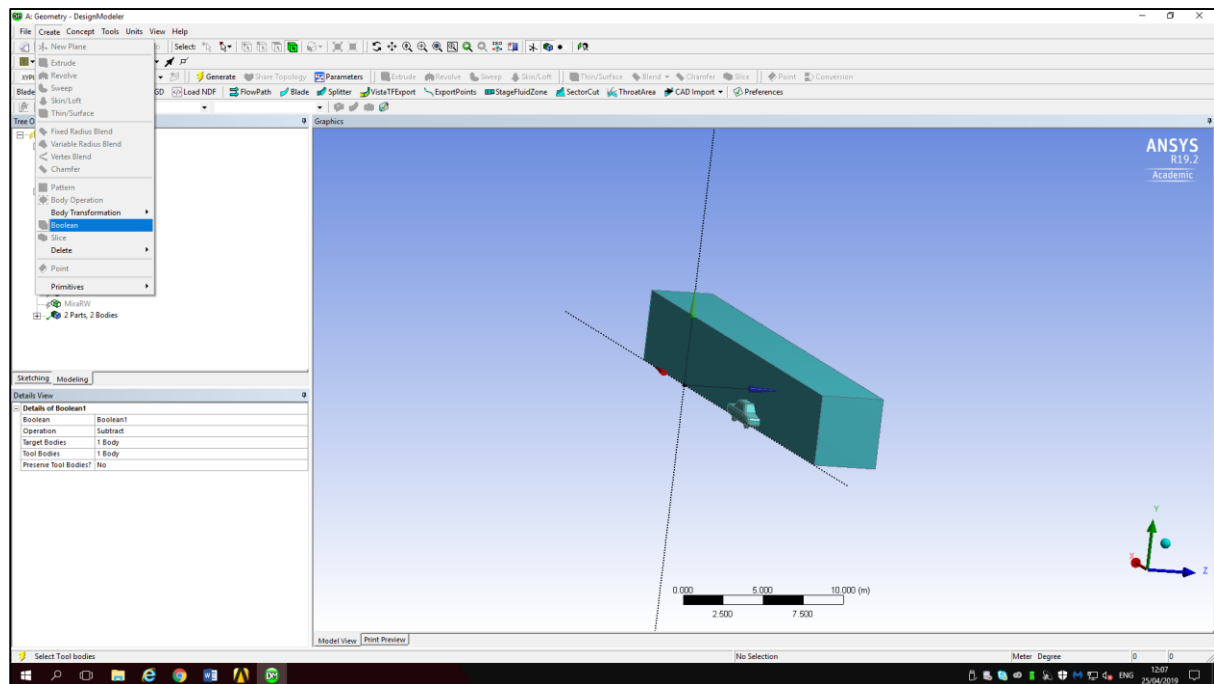


Figure 13 - Inserting the bolleen into the geometry

Once the Bolleen has been made, the individual faces of the model need to be assigned, ranging from velocity inlet, to identifying individual parts of the car itself. The diagram below shows the process used for one of the faces:

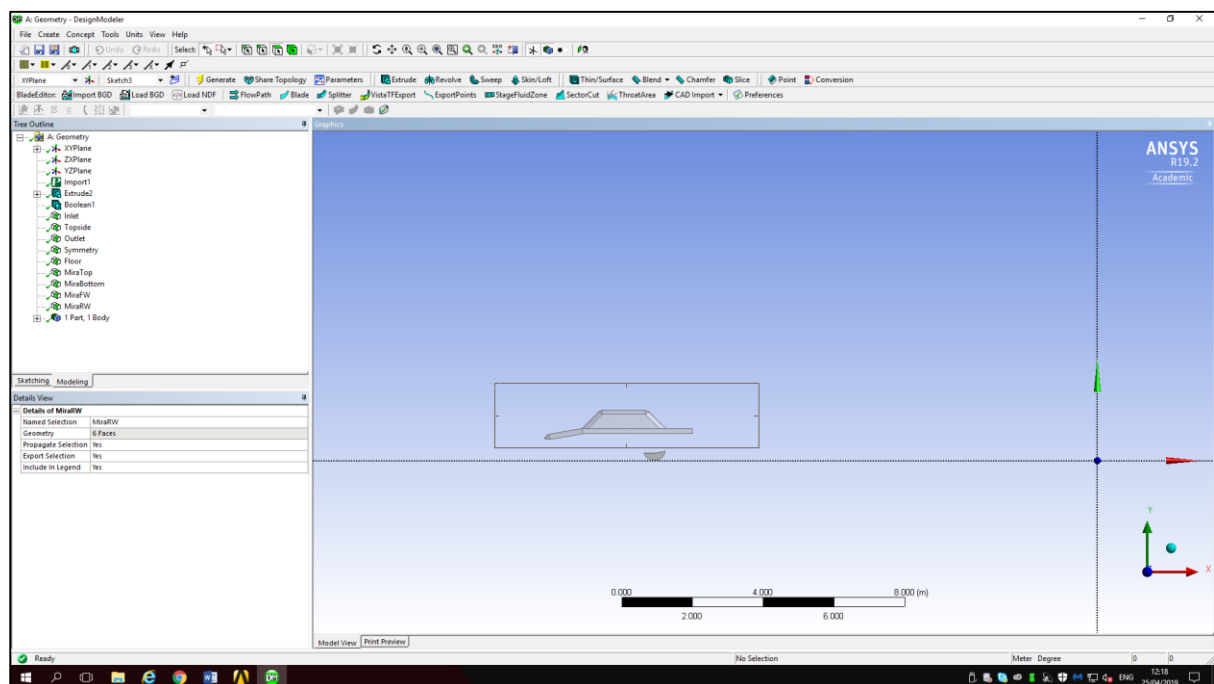


Figure 14 - Selecting the faces of the model

Now that the geometry section has been completed, the ICEM section can be loaded to start the meshing process. Once ICEM has loaded, the first step is to repair the geometry. Once this has been done, the various settings for the meshing need to be applied. The diagram below shows the various settings used for the mesh for the base setup:

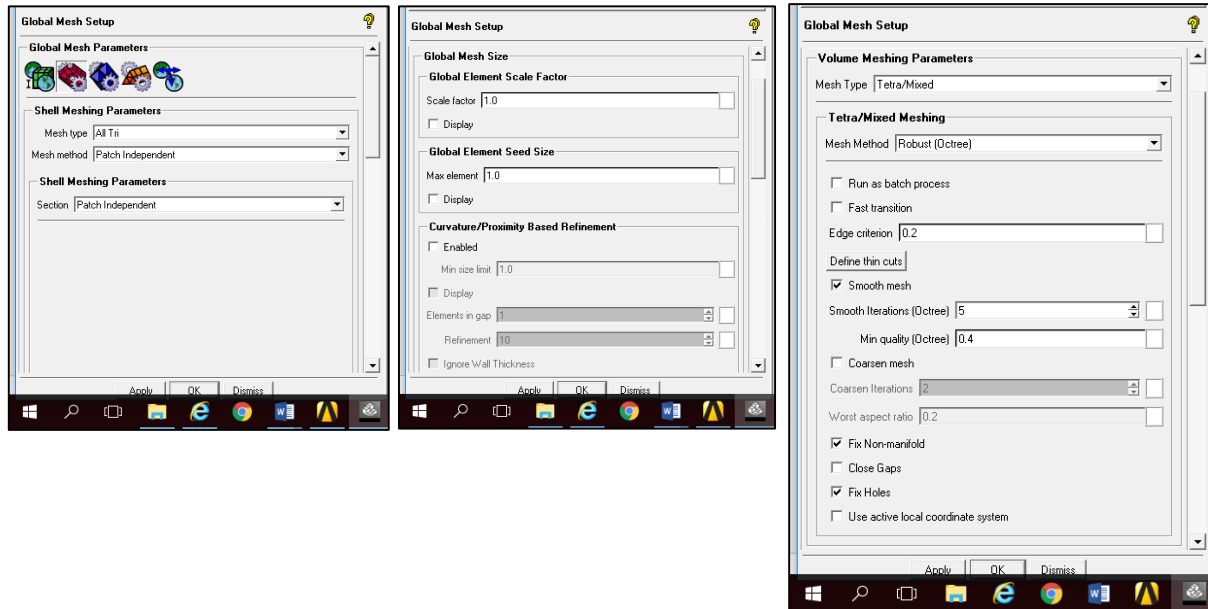


Figure 15 - The various settings used through the meshing process

Once the settings have been defined, the areas used for prism layers inside the model need to be identified and selected, imputing the number of layers needed as well as the dimensions for the maximum size of the prism layers. The diagram below shows the function used to bring up the settings required, and the numbers used for the settings:

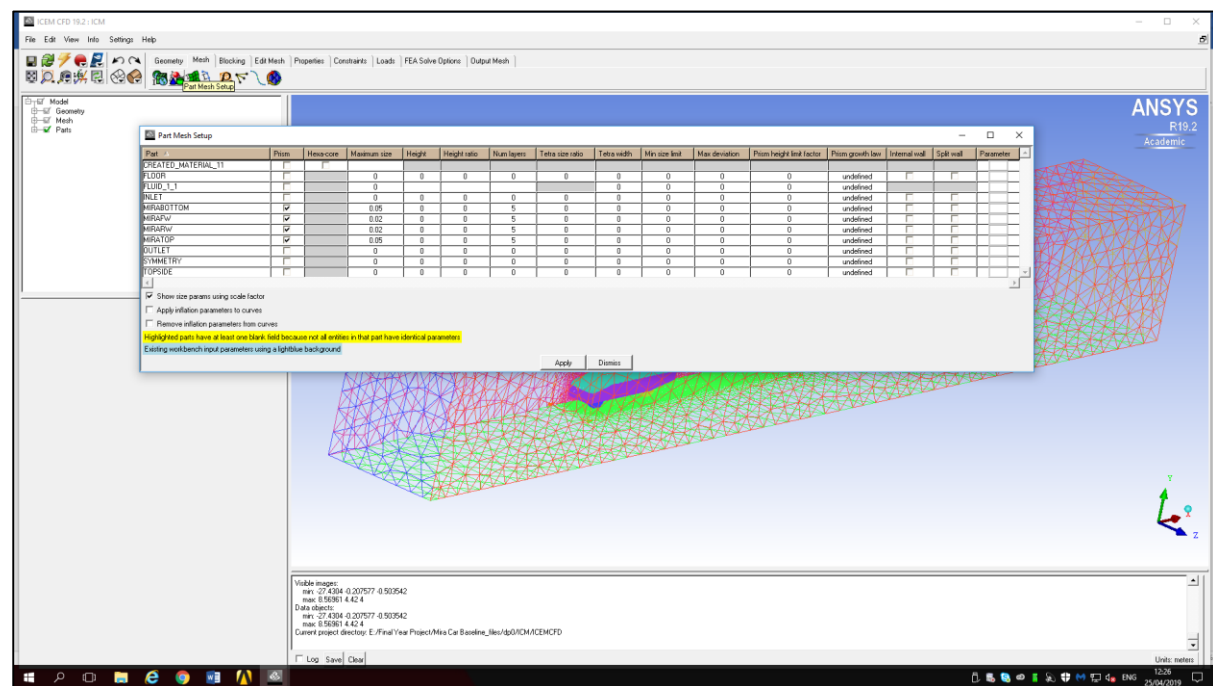


Figure 16 - The settings used to generate prism layers around intended areas

The next step is to apply a density box around the model of the car inside the geometry. The size of the density box used was 0.1 for the base setup. Once the density box has been made, the size, and the position needs to be altered.

Once the density box is in the correct position, the volume mesh can be applied to the geometry, shown in the diagram below:

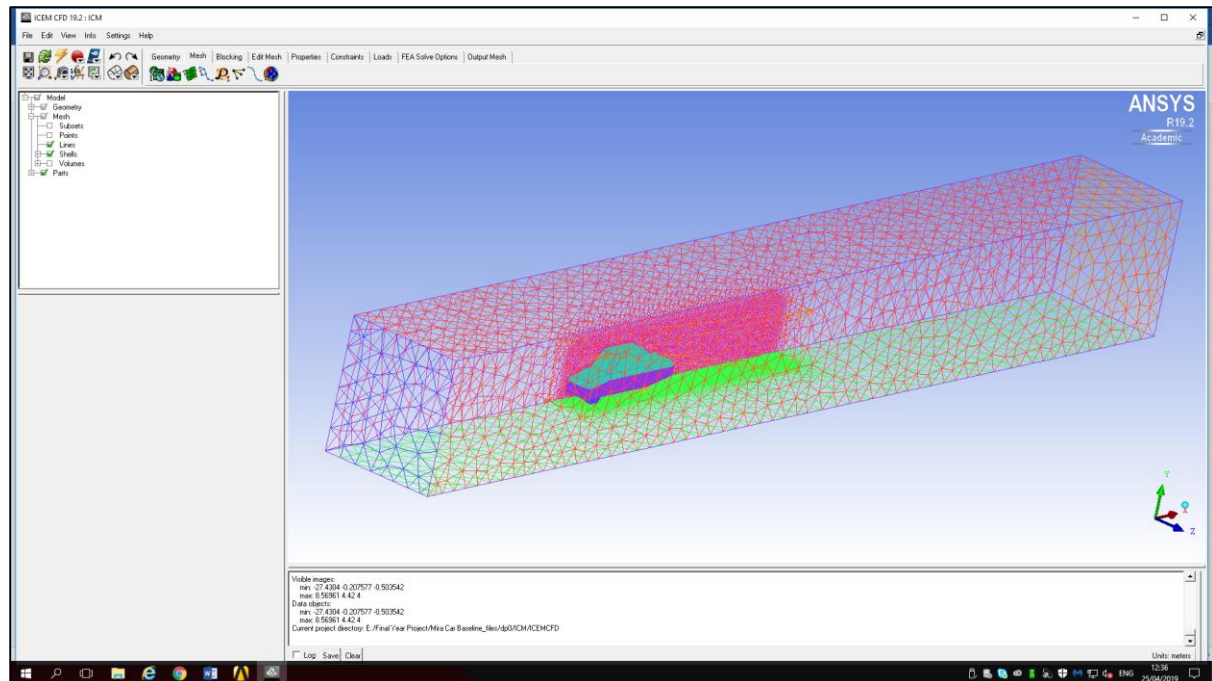


Figure 17 - The volume mesh generated throughout the geometry

Once the mesh has been carried out, the document needs to be saved. Once this has been done, the prism layers mesh can be started, which will take longer compared to the volume mesh. The process used to apply the prism layer mesh is the same as the volume mesh.

After this step has been completed, ICEM can be closed, and fluent can be opened. Once fluent has loaded with the meshed geometry, all the necessary settings need to be inputted. The diagram below shows all the potential settings that need to be changed:

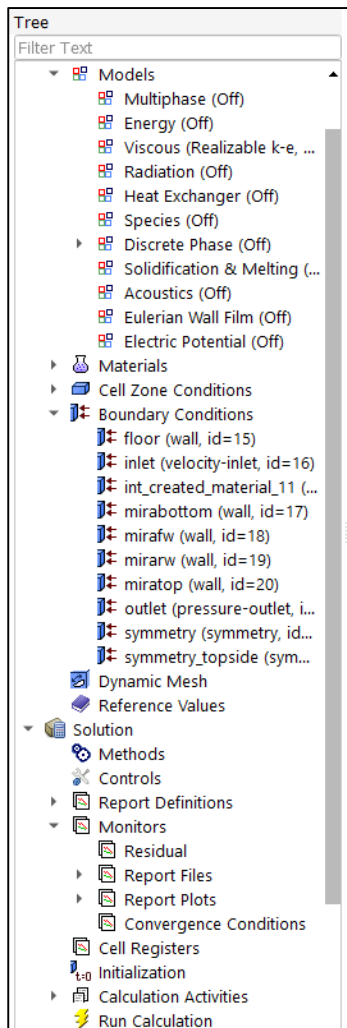


Figure 18 - The various settings that need to be set up before starting the fluent process

The turbulence model, the viscous model shown in figure 18, needs to be set to the realizable K-epsilon model, used with non-equilibrium wall functions.

Under boundary conditions, the inlet needs to be set as a velocity inlet, and under parameters, the velocity specification method needs to be set using components, with the reference frame set to absolute. The speed of the air needs to be set at twenty mph, moving in the x direction.

The reference values can be seen in the appendix. The values need to be computed from inlet face, as otherwise the values for coefficient of lift and drag produced from the calculation will be inaccurate.

The settings used for the methods options, as well as control settings can be seen in the appendix. The limits seen in the appendix are changed to allow the largest number possible to be achieved, otherwise the results of the calculation will not be as accurate as desired.

Under report definitions, two graphs need to be created for coefficient of drag and lift. Force reports are the option needed. When creating the graphs, only the faces of the car need to be selected.

Under the monitors tab, select the residual setting. Once loaded, the convergence criterion option needs to be changed from absolute to none, otherwise the software will finish the calculation early, as it will believe that data being collected has levelled out.

Once all these settings have been changed, the calculation can be initialized. The initialization tab is selected, and the hybrid version used. Now completed, the calculation can be run, and solutions gathered. The run calculation tab is selected. One thousand iterations are used, with the reporting interval set to one, as well as the update interval.

After the calculation has been completed, the values obtained for coefficient of drag and coefficient of lift are 0.325 and -0.131. Looking at the data gathered by Angelina I. Heft, 2012, the value found for the coefficient of drag is similar to numbers discovered through testing, as seen in the table below:

Table 2. Comparison of the drag coefficients of the different configurations of the DrivAer model without ground simulation with the drag values of the mock-up Audi A4 and the BMW 3 series geometries.

	Estate Back	Notchback	Fastback
Audi	0.288	0.260	0.259 ²
BMW	0.257	0.248	- ³
DrivAer	0.319	0.277	0.275

Figure 19 - Data gathered by Angelina I. Heft

For the coefficient of lift, the value produced from testing is similar to numbers produced by V. Naveen Kumar, 2015, which can be seen below:

Table 6.1: Car without Rear Spoiler

S. No	Velocity (Km/h)	Drag Force (N)	Lift Force (N)	Cd	Cl
1	70	153.86	51.109	0.329	0.1183
2	90	255.0	85.711	0.330	0.117
3	110	383.397	129.588	0.331	0.1153
4	130	536.83	182.00	0.333	0.108
5	150	718.946	248.474	0.334	0.096

Figure 20 - Data gathered by V. Naveen Kumar

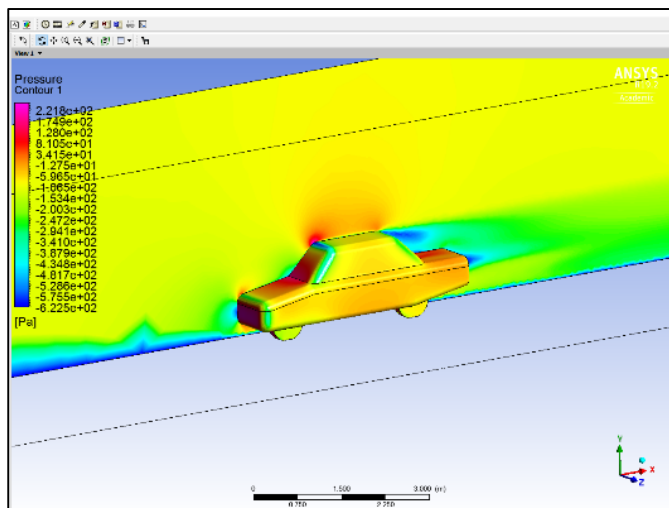


Figure 21 - A visual representation of the air flow around the model

Looking at the diagram on the left, the model shows that high pressure is acting on the front of the vehicle, and along the bonnet of the car and up the windscreen. At the rear of the vehicle, as the air flow moves away from the car, there are two areas of low pressure, which is what is expected.

3.2 Investigations

After completing the base setup case study, investigations into the defining factors of the model need to be made. The three main factors that need to be investigated include domain size, mesh size and the turbulence model used. The values for coefficient of drag and lift being aimed for are 0.296 and -0.08.

3.2.1 Domain Size

The size of the domain is the first investigation to be carried out. To alter the size of the domain, the workbench file used needs to be reopened, and as seen above in figure (), the dimensions for the measurements can be changed. The table below shows the dimensions used for the different domain size tests:

Test Run	Dimensions (m)			
	H16	H20	V18	V19
1	12	27	0.02	3.3
2	14	31.5	0.02	3.85
3	14	36	0.02	4.4
4	14	40	0.02	4.4
5	16	35	0.02	4.4
6	16	36	0.02	4.4
7	16	37	0.02	4.4
8	16	38	0.02	4.4
9	16	39	0.02	4.4
10	16	40	0.02	4.4
11	16	41	0.02	4.4
12	20	45	0.02	5.5

Table 2 - The tests carried out on the domain size of the geometry

Once all the domain sizes have been changed, the results of the calculations can be seen in the graph below:

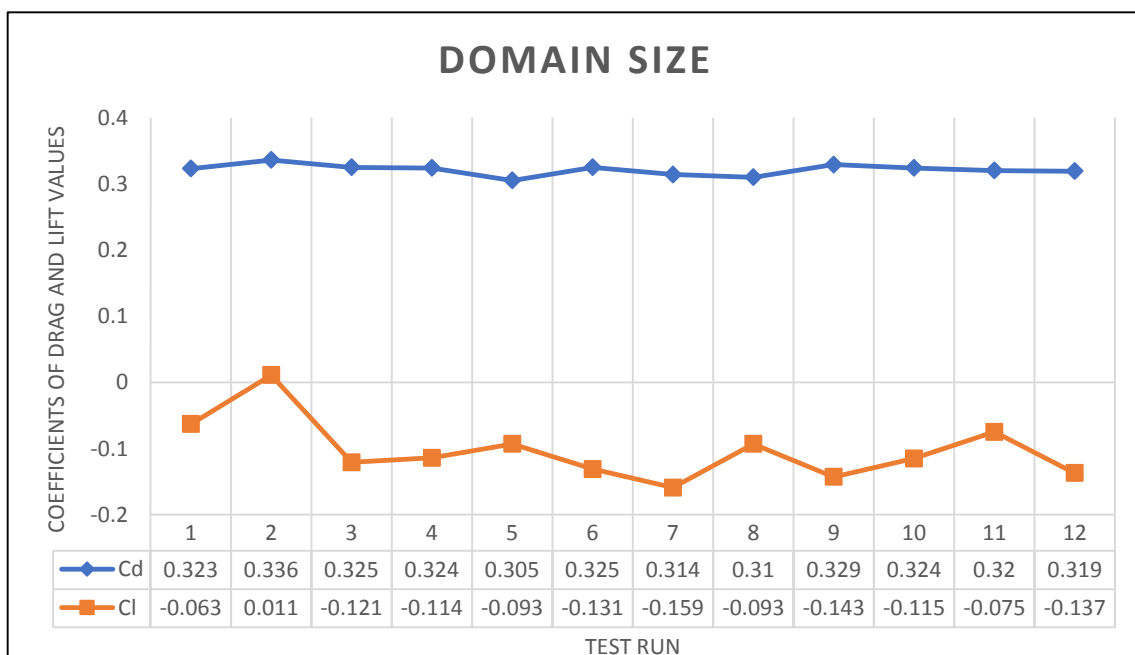


Figure 22 - The coefficient of drag and lift results for the tests changing the domain size

Looking at the results of the domain size case studies, test run five and eight produced values for coefficient of lift and drag closest to the target for these investigations, with the test run five being chosen going forward. Through looking at the results, even a small change to one of the dimensions of the domain has a direct influence of the outcome of the calculations. The two diagrams below illustrate how changing the size of the domain can affect the aerodynamic drag and lift, but not influence turbulent air around the vehicle:

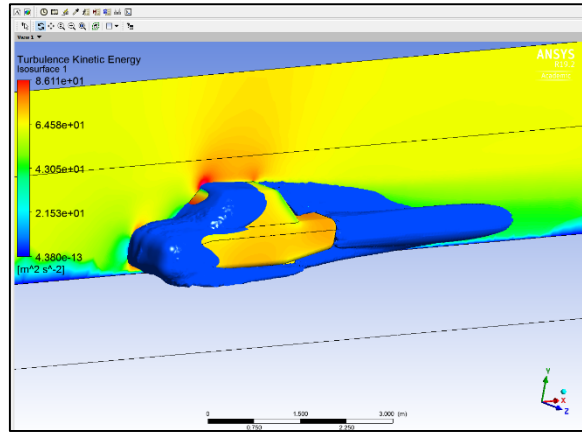


Figure 23 - The turbulent air flow visible in domain size test 1

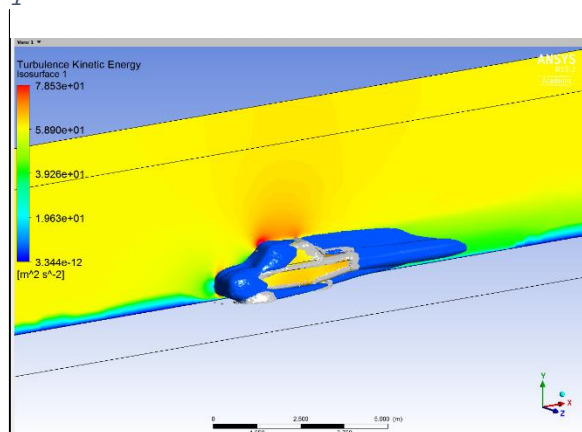


Figure 24 - The turbulent air flow visible in domain size test 12

3.2.2 Mesh Size

The second investigation to be made involves the size of the mesh used. To change the size of the mesh used, the settings need to be opened using the same method as shown in figure 15. Out of the three different settings tabs visible in the figure, global mesh size is the variable being changed. The table below shows the sizes of the mesh tested:

Test Run	Mesh Size
1	3
2	2
3	1.2
4	1.1
5	1
6	0.9
7	0.8

Table 3 - The tests carried out for the mesh size of the geometry

When all the mesh sizes have been changed, calculations can be made, and the following results achieved:

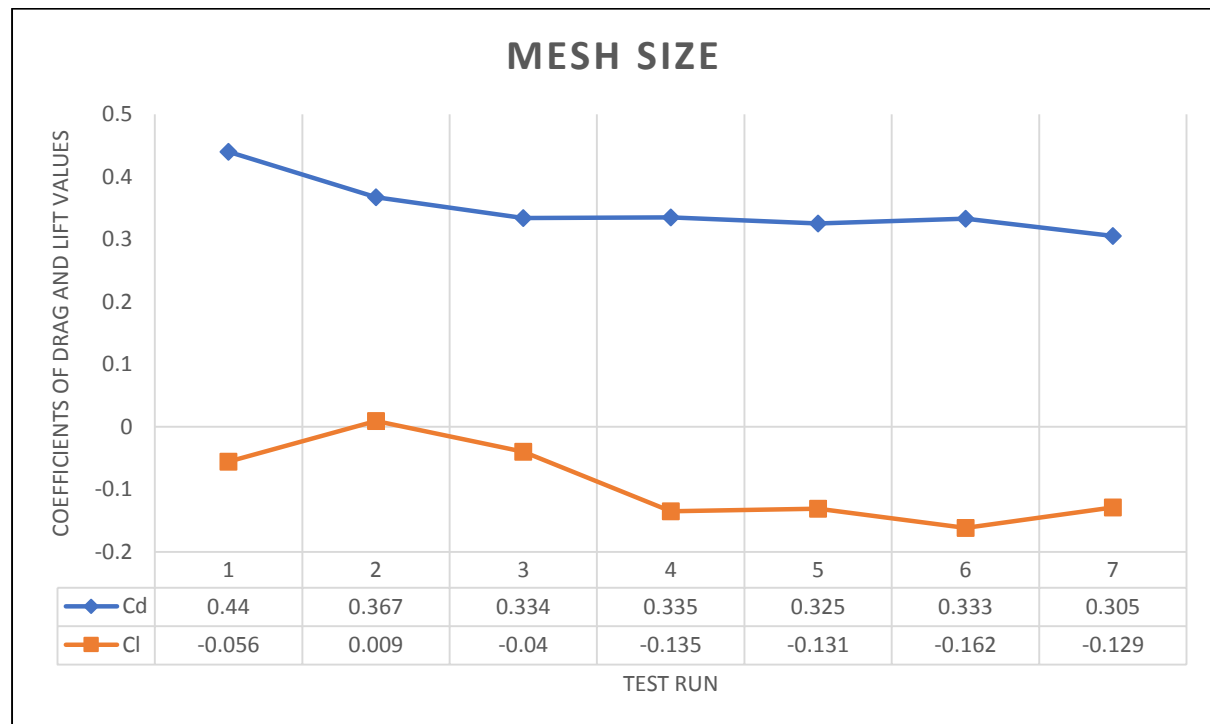


Figure 25 - The coefficient of drag and lift results for the different mesh sizes

Looking at the results of the different mesh sizes, test run seven produced the best outcome for the coefficient of drag and lift. The two diagrams below illustrate the difference between using a fine mesh and a coarse mesh. Using the smaller mesh will improve the validity and accuracy of any tests carried out, as stated and shown in the work by Ahmad NE, 2010:

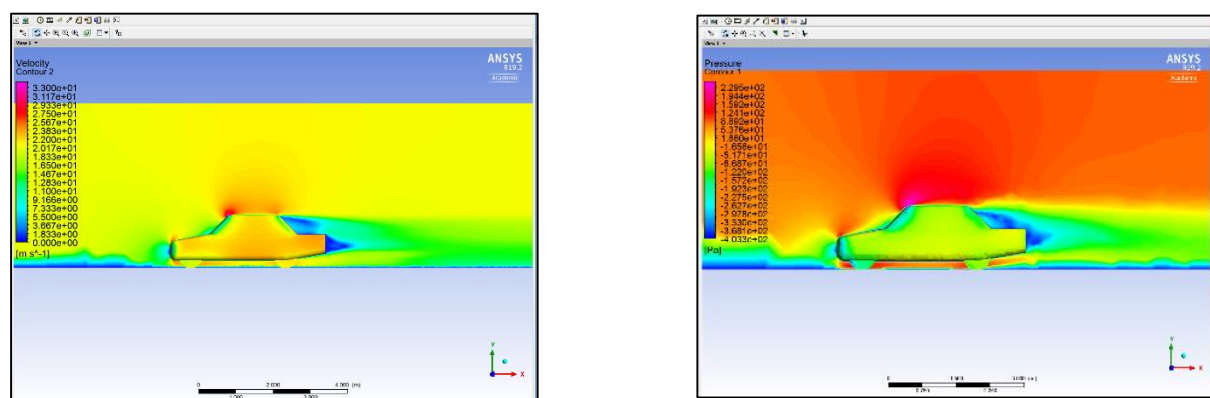


Figure 26 - A visual representation of changing the mesh sizes

3.2.3 Turbulence Model

The third investigation to be carried out is linked to turbulence model. When loading fluent, and choosing a turbulence model, there are variety of models that could be picked. The table below lists the turbulence models tested:

Test Run	Turbulence Model
1	K-Epsilon
2	K-Omega
3	Transition K-KL-Omega
4	Transition SST

Table 4 - The tests carried out for the turbulence model

The following graphs illustrates the results of the calculations carried out using the different turbulence models:

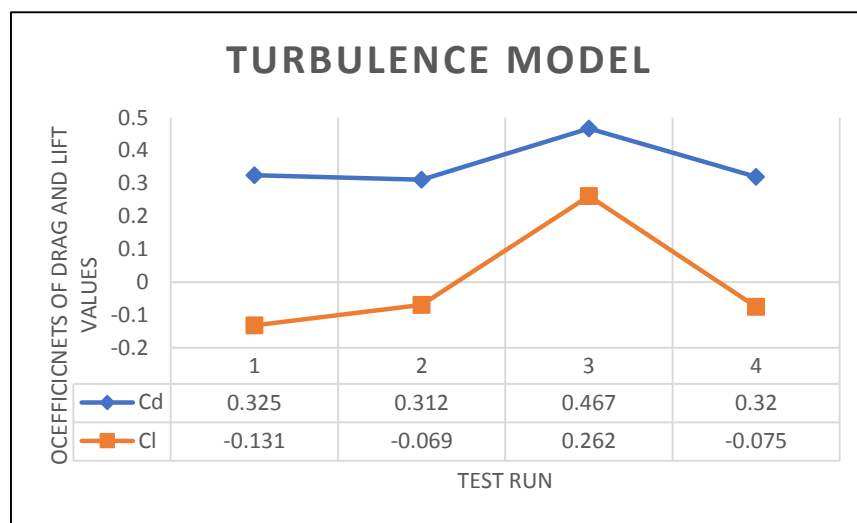


Figure 27 - The coefficient of drag and lift results for changing the turbulence model

Looking at the results of the investigations, Transition K-KL-Omega could not be used as the turbulence model moving forward, due to the high value for coefficient of lift and drag, even though, illustrated by the diagram below, the turbulence model had an advantage in highlighting the change in velocity around the vehicle:

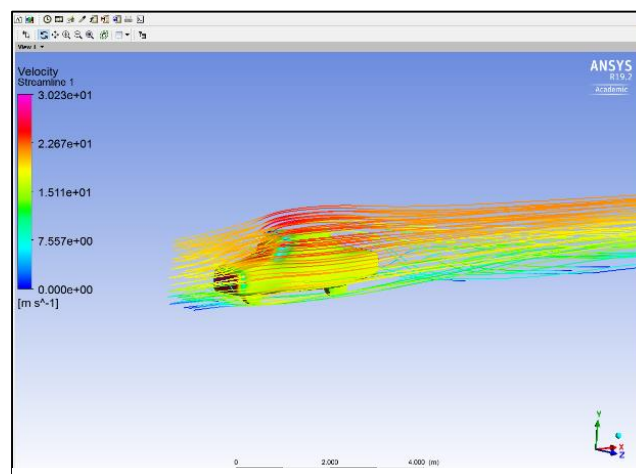


Figure 28 - A visual representation of the change in velocity of the air flow around the model

Out of the three remaining turbulence models tested, K-Omega is preferred over K-Epsilon and Transition SST. This is due to the turbulence model being able to illustrate the changes in turbulent air flow better when compared to the other two, as seen in the diagrams below. Work carried out by Renan Francisco Soares, 2015 also produced similar results:

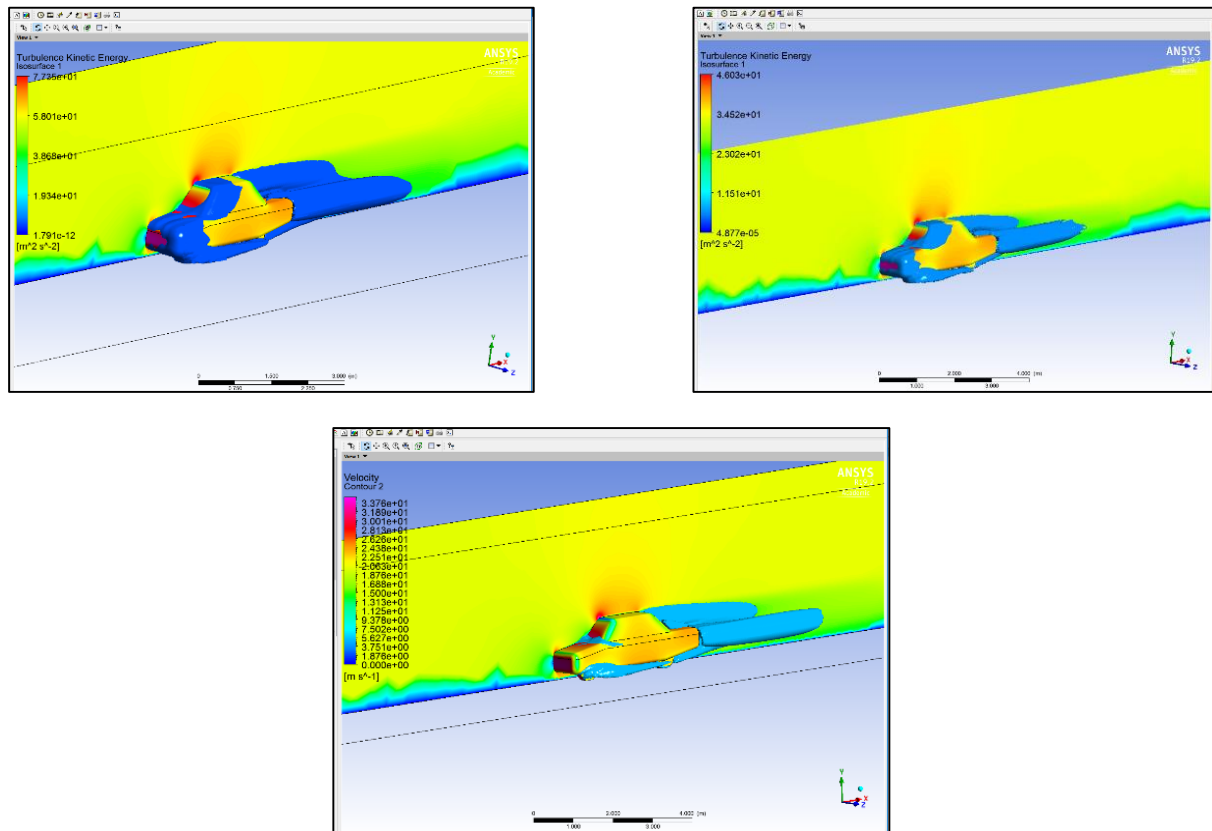


Figure 29 - A visual representation of the change in turbulent air flow around the model; top left illustrates the K-Epsilon turbulence model; top right illustrates the K-Omega turbulence model; with the bottom ones representing the Transition SST turbulence model

3.3 Fully Defined Model

Using the best configuration of the results from the three separate investigations, a fully defined model can be created, and calculations can be run. Test run five from domain size testing, test run seven from mesh size testing, and the K-Omega turbulence model have been used to generate results for the fully defined model of the vehicle. The values for coefficient of drag and lift for the fully defined model are 0.311 and -0.077. Looking at the differences between the original base setup and the fully defined model, the mesh size appears to be the definitive factor, as it improves the accuracy of the results, whereas changing turbulence model or domain size has a smaller effect in comparison.

4.0 Vehicle Geometry Investigations

4.1 Vehicle Shape

There are four generic vehicle shapes; notchback; fastback; pickup; and estate. These four different shapes need to be investigated as the effect of aerodynamic drag and lift will be different for each shape. All the tests carried out have been using the notchback shape. To be able to convert the model from the notchback shape to the alternative shapes, Computer Aided Design (CAD) needs to be used. Once the changes have been made to the designs, the four shapes below are the result:

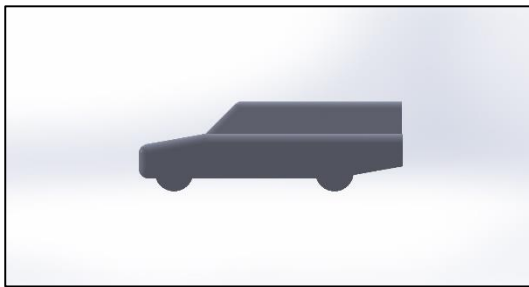


Figure 30 - Cad model of the estate model variant

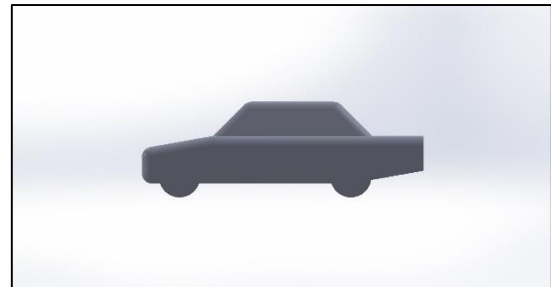


Figure 31 - Cad model of the notchback model variant

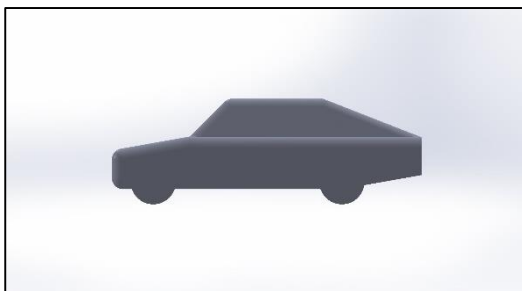


Figure 32 - Cad model of the fastback model variant

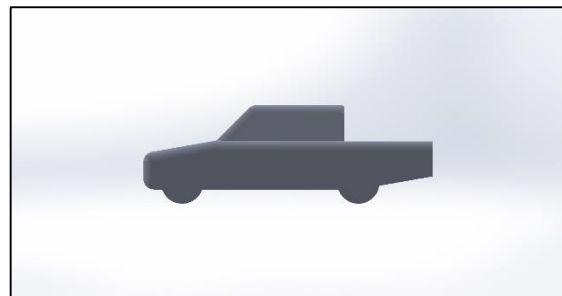


Figure 33 - Cad model of the pickup model variant

As soon as the three alternative vehicle shapes have been created, separate CFD files need to be produced, so that values for coefficient of drag and lift can be calculated. The graph below illustrates the values obtained for the two coefficients:

Test Run	Fully Defined Models
1	Notchback
2	Fastback
3	Pickup
4	Estate

Table 5 - Test run layout for different car shapes

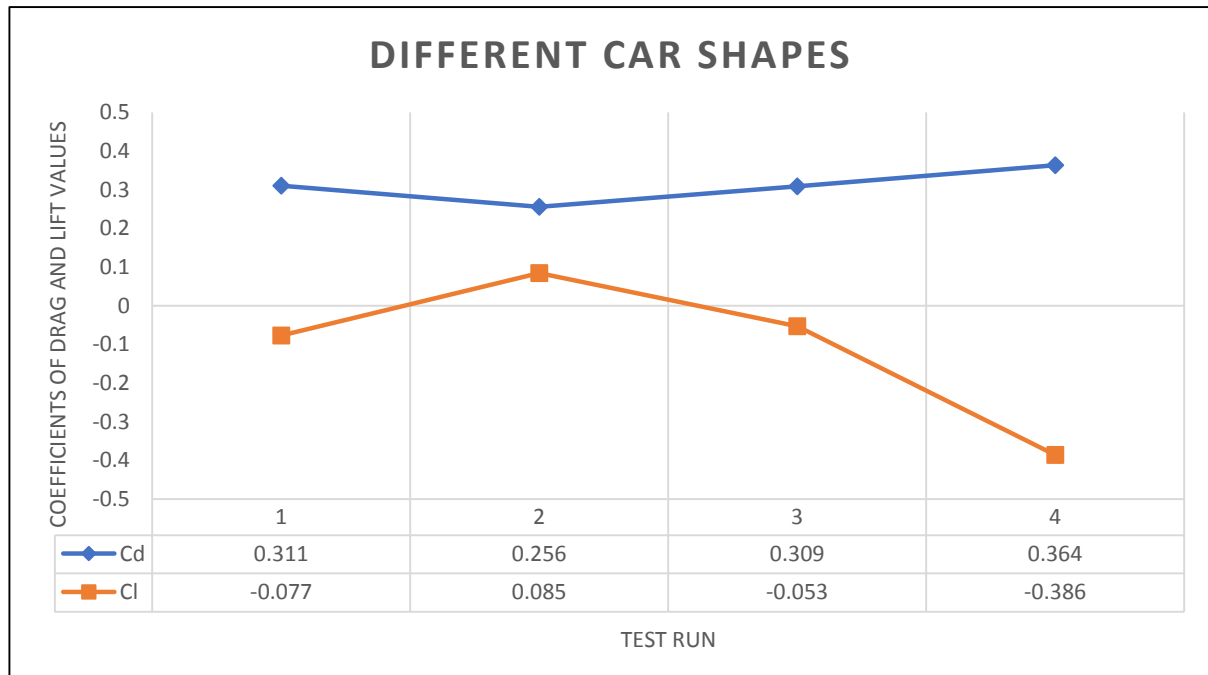
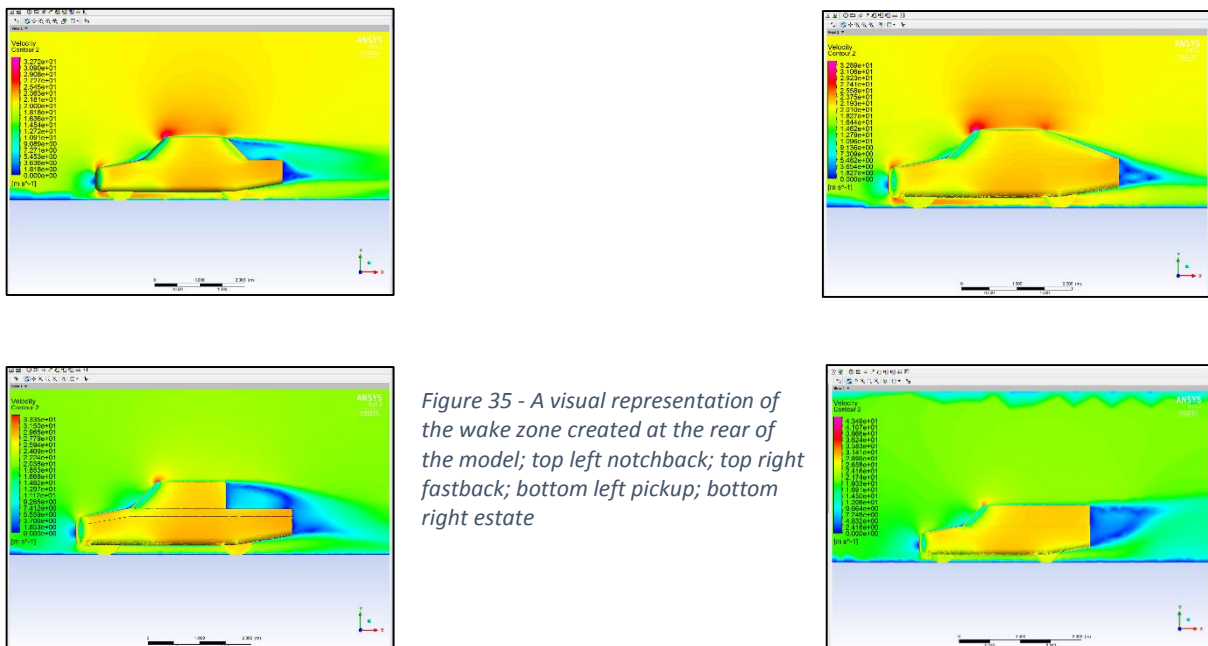


Figure 34 - The coefficient of drag and lift results for the different vehicle shapes

Testing carried out by; Renan Francisco Soares, 2015; Pruthviraj Mohanrao Palaskar, 2016; and George B. 2014 validate the data gathered through testing. However, the value for coefficient of lift produced from the estate variant is lower than expected.

For each of the variants, the size of the wake zone at the rear of the vehicle varies, as seen in the diagrams below:



The fastback variant produces the smallest wake out of the four, as the air flow stays connected to surface of the vehicle for longer and not separating. The notchback version has one small wake zone forming off the rear of the vehicle, and another smaller zone forming as air flow separates from the surface of the car as it travels past the rear windscreen. The pickup variant produces a medium sized wake zone at the rear of the vehicle, but due to the shape of the car, air becomes unattached easily and falls into the unoccupied space. Furthermore, the estate variant produces the largest wake zone, as the air stays attached to the surface of the vehicle, similar to the fastback alternative. However, the area at the rear of the estate version is substantially greater than the fastback, meaning that the wake zone would be larger in size.

The way in which these different models are affected by turbulent air also changes. Looking at the following diagrams, the estate variant is the model most affected, due to having the massive area at the rear of the vehicle filled by low pressure:

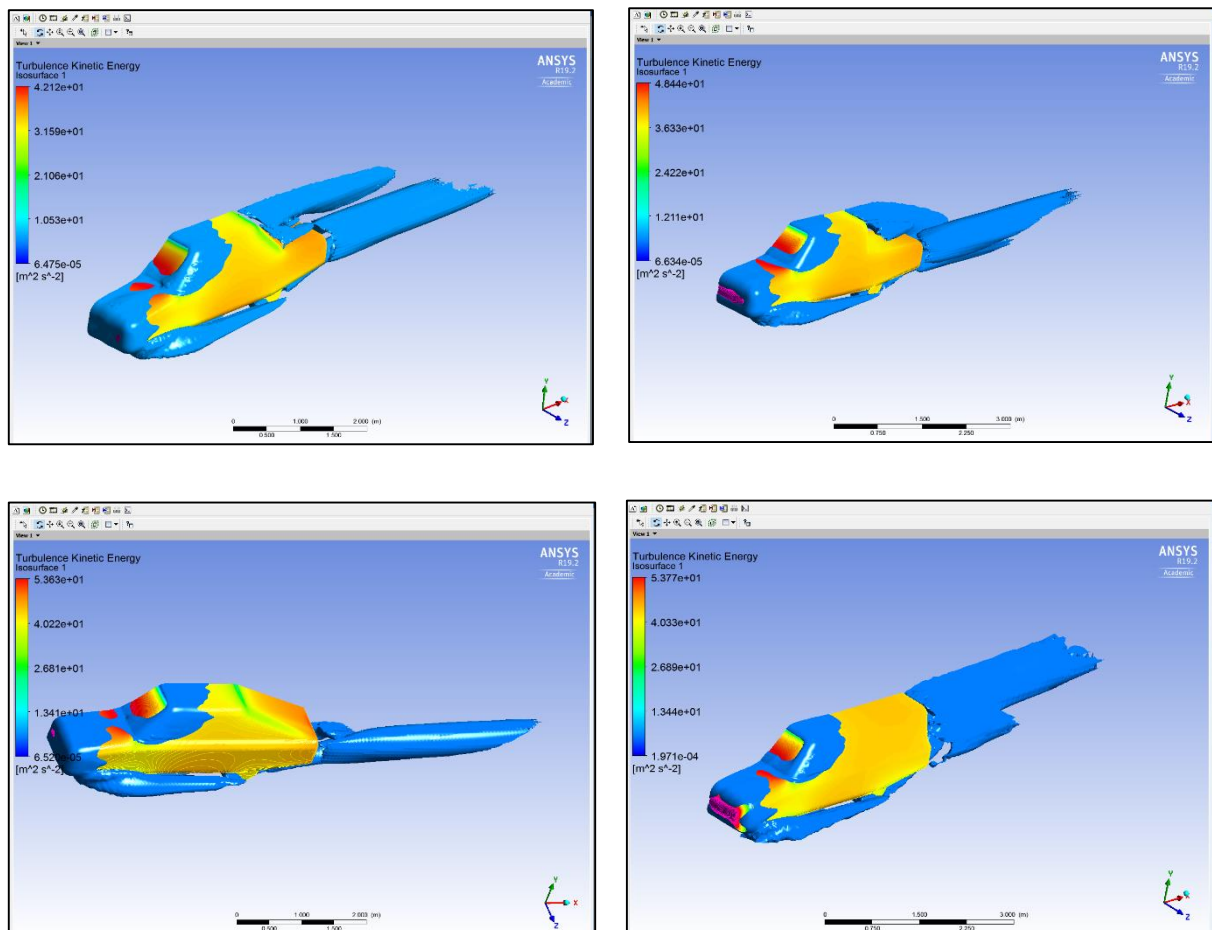


Figure 36 - A visual representation of the turbulent air flow around the vehicle; top left notchback; top right pickup; bottom left fastback; bottom right estate

4.2 Air Dam

After completing the calculations for the different vehicle models, aerodynamic devices were added to test if by adding them, the aerodynamic characteristics of the car improved or became worse. An Air Dam was the first device to be tested. To add an air dam to the models, the CAD models used before need to be opened again and changed. An example of the notchback model with an air dam fitted can be seen below:

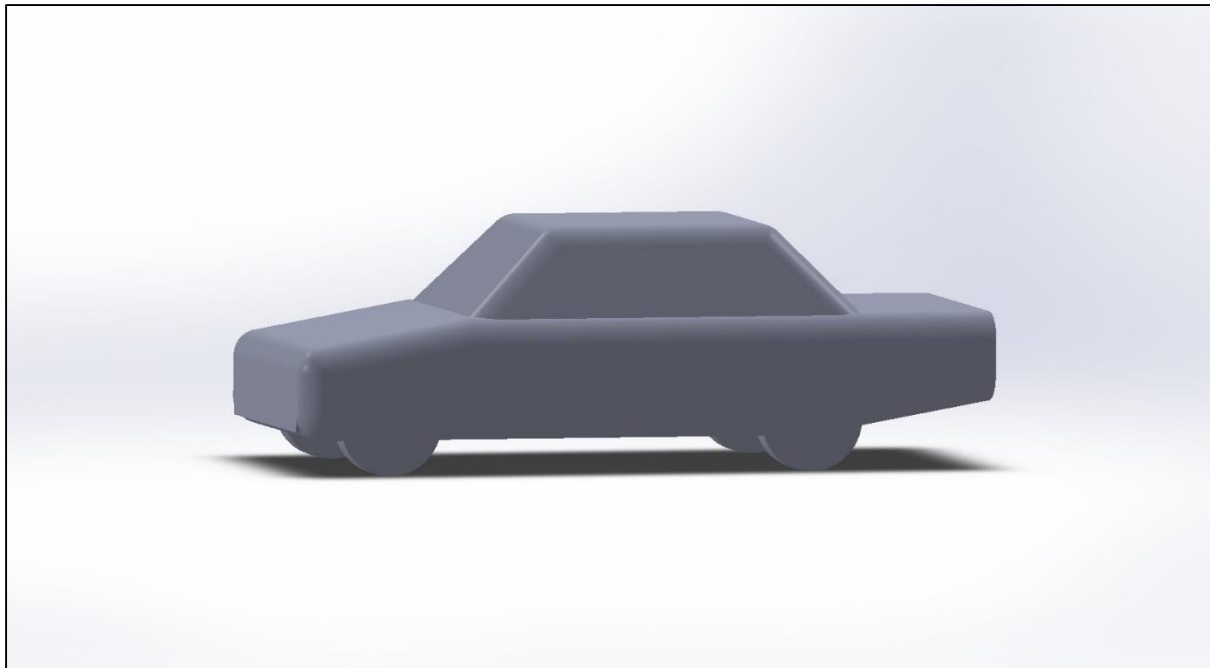


Figure 37 - An example of an Air Dam fitted to the notchback model

Three different lengths of air dam will be tested on each model. The results can be seen below. When the testing was carried out, some of the models could not converge to produce a solution. All solutions for coefficient of drag and lift highlighted green were achieved using the parameters obtained through sections 3.2.1, 3.2.2, and 3.2.3. The boxes highlighted blue were achieved through changing the type of mesh used in fluent, converting from tetrahedral to polyhedral. The test highlighted red illustrates where changing the type of mesh did not work, meaning a change in turbulence model was needed to be able to gather results:

	Estate		Fastback		Notchback		Pickup	
	Cd	Cl	Cd	Cl	Cd	Cl	Cd	Cl
0mm	0.364	-0.386	0.256	0.085	0.311	-0.077	0.309	-0.053
100mm	0.34	-0.337	0.268	0.084	0.325	-0.035	0.343	0.038

200mm	0.36	-0.24	0.361	0.26	0.386	0.238	0.413	0.235
300mm	0.422	-0.227	0.316	0.09	0.352	-0.176	0.354	-0.027

Table 6 - The coefficient of drag and lift results for the three different length Air Dams when attached to each of the different vehicle models

4.2.1 Lift

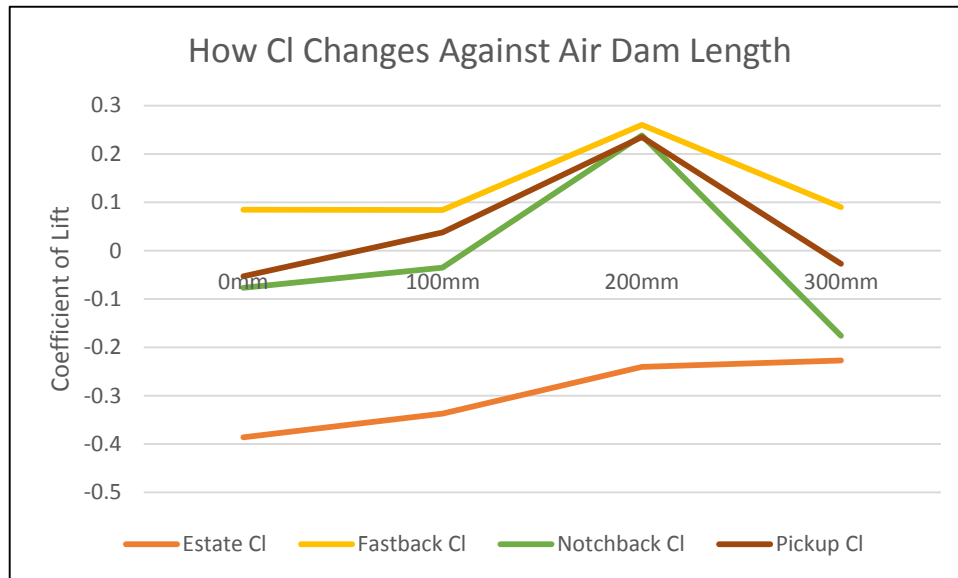


Figure 38 - A graph illustrating the change in coefficient of lift against the four model variants with changing length of air dam

Looking at figure 38, the fastback, notchback and pickup variants follow a similar pattern, with increasing the length of the air dam, the coefficient of lift produced gets worse. However, when moving from 200mm to 300mm, the lift produced by the vehicles improves. Looking at the diagrams below, there is a dramatic increase in wake zone size when increasing the length of the air dam. This is due to the air dam sealing the front of the car from the ground, meaning the air flow is forced to go round the vehicle by either moving along the top of the car, or by going round the side. This is the reasoning for increased amount of slow air around the front wheel of the car:

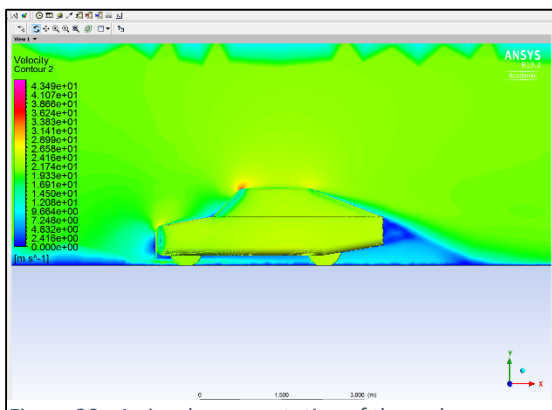


Figure 39 - A visual representation of the wake zone created by attaching a 200mm air dam to the front of the vehicle. Fastback model shown

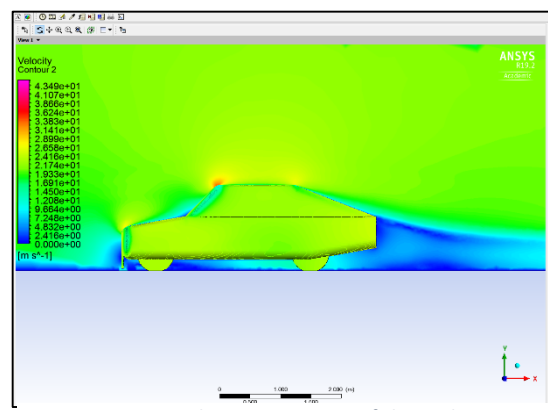


Figure 40 - A visual representation of the wake zone created by attaching a 300mm air dam to the front of the vehicle. Fastback model shown

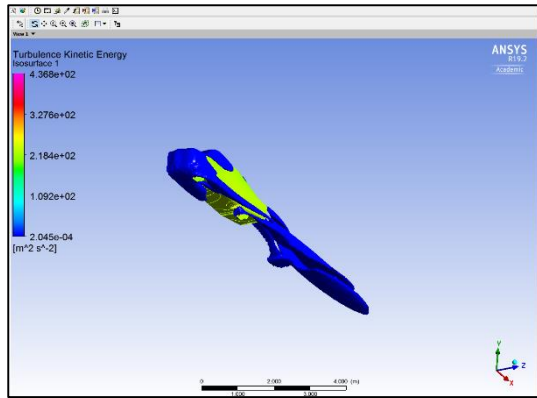


Figure 41 - A visual representation of the turbulent air flow created by attaching a 200mm air dam to the front of the vehicle. Fastback model shown

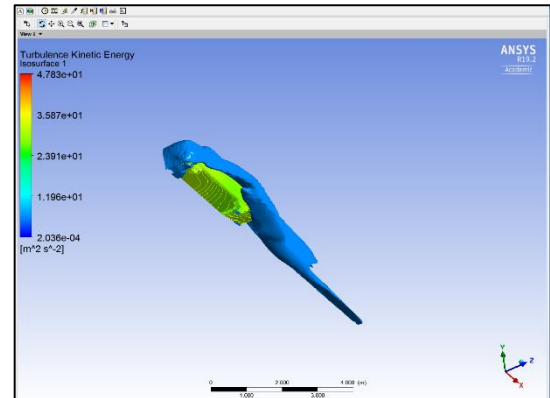


Figure 42 - A visual representation of the turbulent air flow created by attaching a 300mm air dam to the front of the vehicle. Fastback model shown

The increase in coefficient of lift is produced by more high pressure being present under the rear of the vehicle, as seen in the diagrams below. The change in pressure is caused by the change in velocity under the vehicle, as an area of low velocity results in high pressure, as stated by Bernoulli's equation, discussed in section 2.1.2:

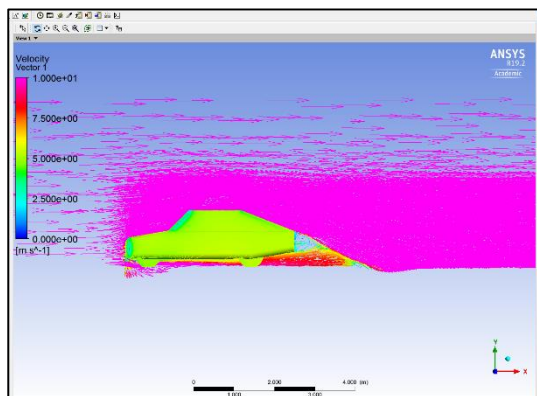


Figure 43 - A visual representation of the velocity of the air flow around the vehicle when a 200mm air dam is attached. Fastback model shown

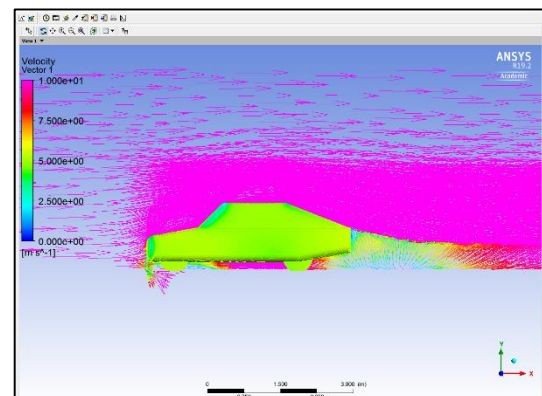


Figure 44 - A visual representation of the velocity of the air flow around the vehicle when a 300mm air dam is attached. Fastback model shown

4.2.2 Drag

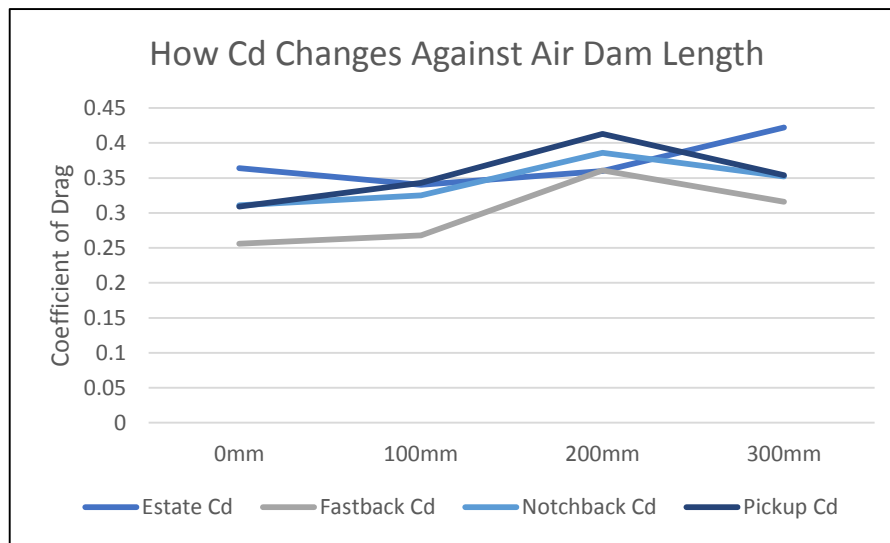


Figure 45 - A graph showing the change in coefficient of drag for the different vehicle shapes with changing air dam lengths

Looking at the graph above, similar to the results seen in figure 38, the notchback, fastback and pickup vehicle shapes follow the same pattern, where the coefficient of drag gets worse as the length of the air dam increases. However, as the length of the air dam reaches 300mm, the coefficient of drag improves. This improvement in coefficient of drag is achieved through the air flow, seen in figures 40 and 41, moving around the vehicle instead to the side instead of being pushed around the top of the car, or fed through the small gap between the bottom of the air dam and the road.

When looking at the estate vehicle, when the 100mm air dam was attached, the coefficient of drag improves. But as the length of the air dam is increased, the Cd value worsens. From the diagrams below, the estate vehicle works differently to the other vehicle models, as it starts to push the air round the sides when the smallest air dam is attached. Furthermore, as the length increases, the air flow hitting the front of the vehicle is not moving to the sides as before, staying connected to the front of the car instead:

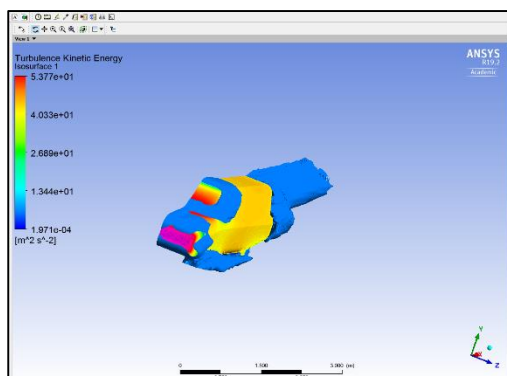


Figure 46 - A visual representation of the turbulent air flow around the estate vehicle with 0mm air dam

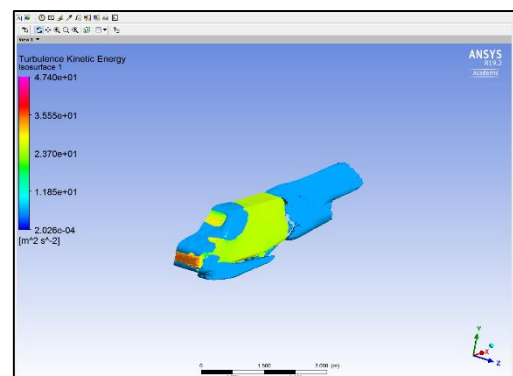


Figure 47 - A visual representation of the turbulent air flow around the estate vehicle with 100mm air dam

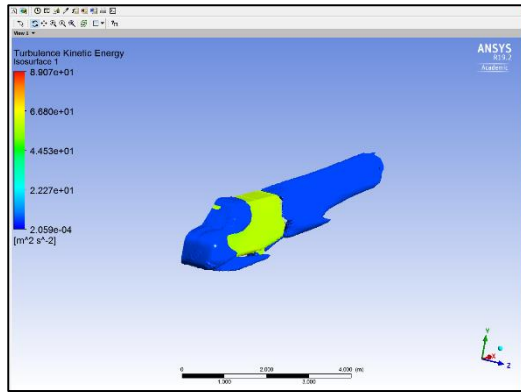


Figure 48 - A visual representation of the turbulent air flow around the estate vehicle with 200mm air dam

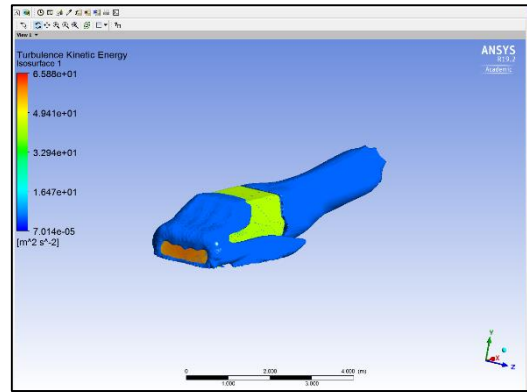


Figure 49 - A visual representation of the turbulent air flow around the estate vehicle with 300mm air dam

4.2.3 Centre of Pressure

The centre of pressure for each of the vehicle variants with different length air dams can also be calculated. When fluent has finished the calculation for the chosen setup, a moment around the model can be taken to find the position of the centre of pressure taken from the rear of the car. Also, the CAD drawings for the models need to be opened, so the distance from the rear of the vehicle to the rear axle can be measured, as seen in the diagram below:

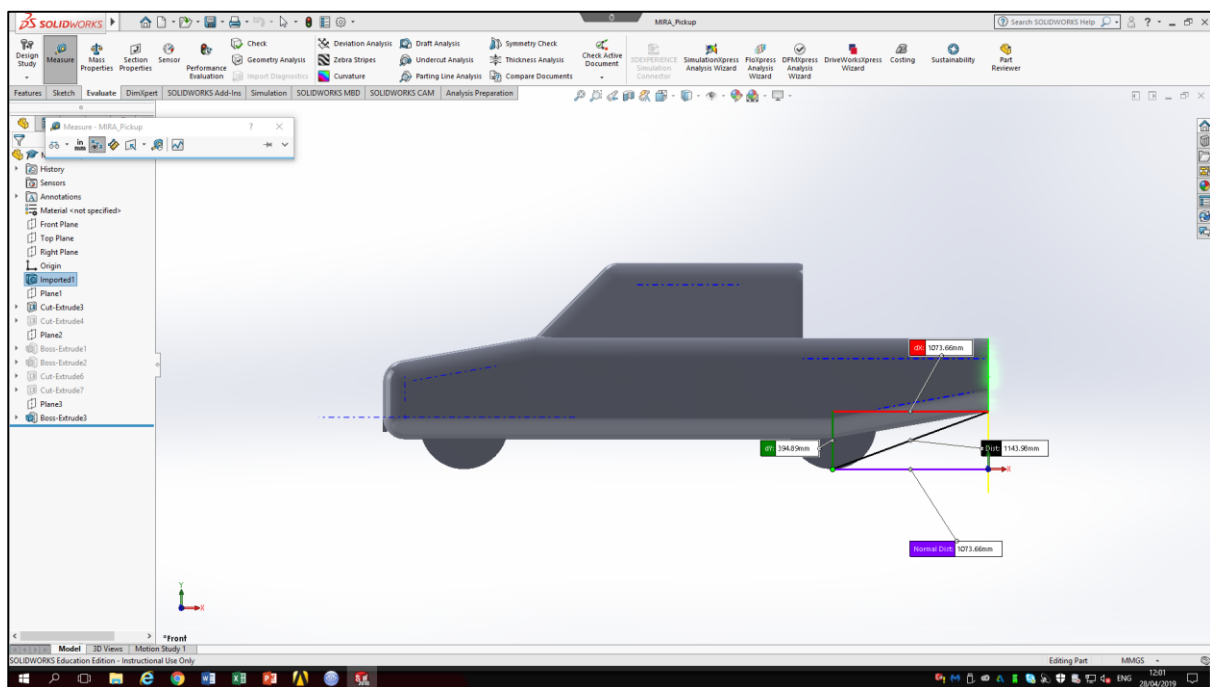


Figure 50 - Measuring the distance between the rear of the vehicle to the rear axle. Pickup model shown

Using the distance measured above, and the measurements taken from fluent, the centre of pressures for the varying vehicle shapes and air dam lengths can be seen below, with measurements and hand calculations shown in the appendix:

	Centre of Pressure (%)			
	Notchback	Fastback	Pickup	Estate
0mm		51.05	62.67	52.63
100mm	54.87	52.82	60.5	52.55
200mm	57.94	53.81	61.17	53.61
300mm	49.95	42.7	48.93	44.75

Table 7 - The centre of pressure results for the different vehicle shapes and air dam lengths

Looking at the results, with all four vehicle shapes, as the air dam length increases, the centre of pressure moves towards the front of the vehicle, at the same time as the coefficients of lift and drag becoming worse. However, as the length of the air dam moves from 200mm to 300mm, the CoP moves quite a distance back towards the rear of the vehicle.

4.3 Spoiler

An addition of a spoiler at the rear of the vehicle will be tested. A spoiler is added to the vehicle using the same procedure as before when adding an air dam. An example of the notchback model with a spoiler fitted can be seen below:

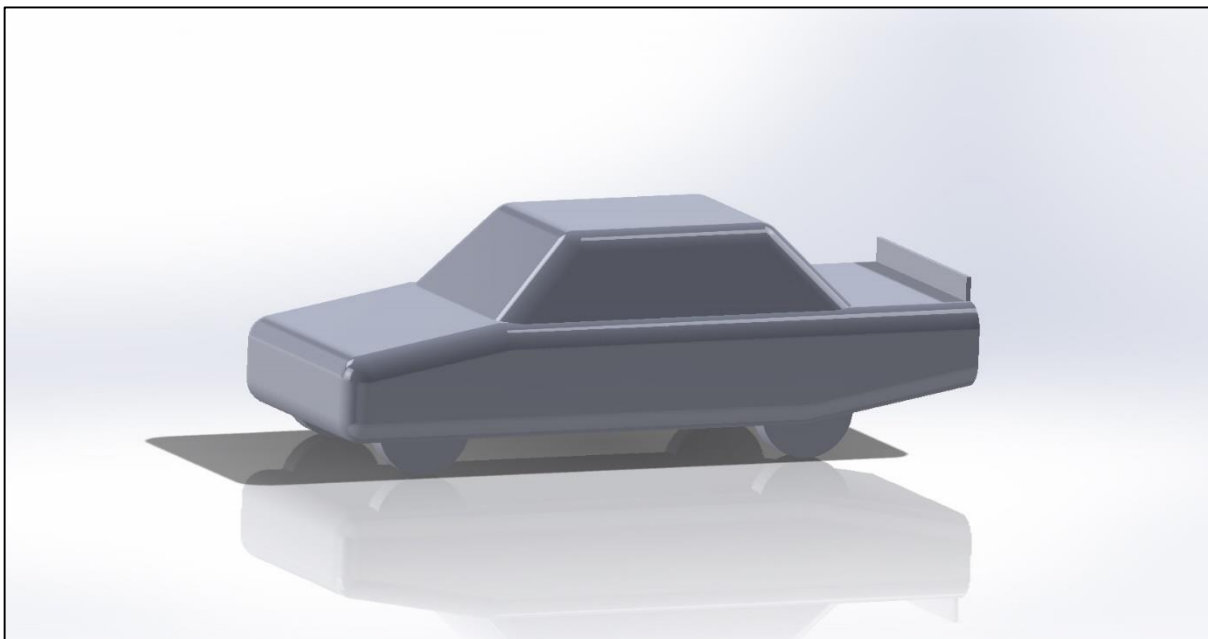


Figure 51 - An example of the spoiler used fitted to the notchback model

Three different heights of spoiler will be tested on each model, with the results being shown below, with the same key is being used as before with the changes to air dam length:

	Estate		Fastback		Notchback		Pickup	
	Cd	Cl	Cd	Cl	Cd	Cl	Cd	Cl
0mm	0.364	-0.386	0.256	0.085	0.311	-0.077	0.309	-0.053
50mm	0.46	-0.568	0.305	0.117	0.349	-0.209	0.401	-0.131
100mm	0.53	-0.683	0.33	-0.107	0.399	-0.32	0.431	-0.134
150mm	0.651	-0.84	0.454	-0.375	0.441	-0.395	0.453	-0.297

Table 8 - The coefficient of lift and drag results for the different vehicle shapes, with varying height of spoiler

4.3.1 Lift

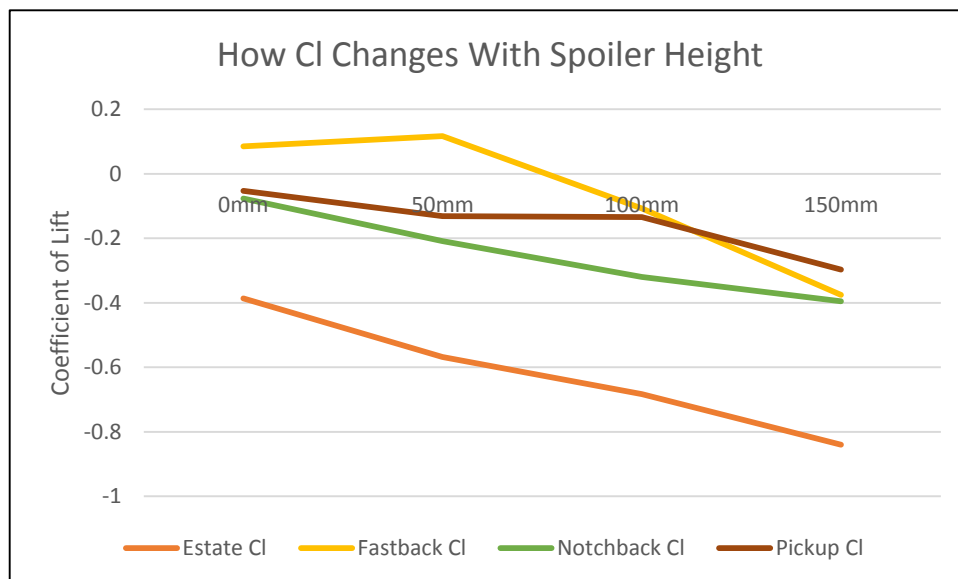


Figure 52 - A graph showing the change in coefficient of lift for changing height of spoiler for the different vehicle shapes

Looking at figure 52, the effect of the spoiler on the coefficient of lift seen is what is expected, as an area of high pressure is created in front of the spoiler, producing more downforce, as seen in the diagram below. Investigations carried out by Renan Francisco Soares, 2018, and Christian Bach Rasmussen, 2018, further improve the validity of the results:

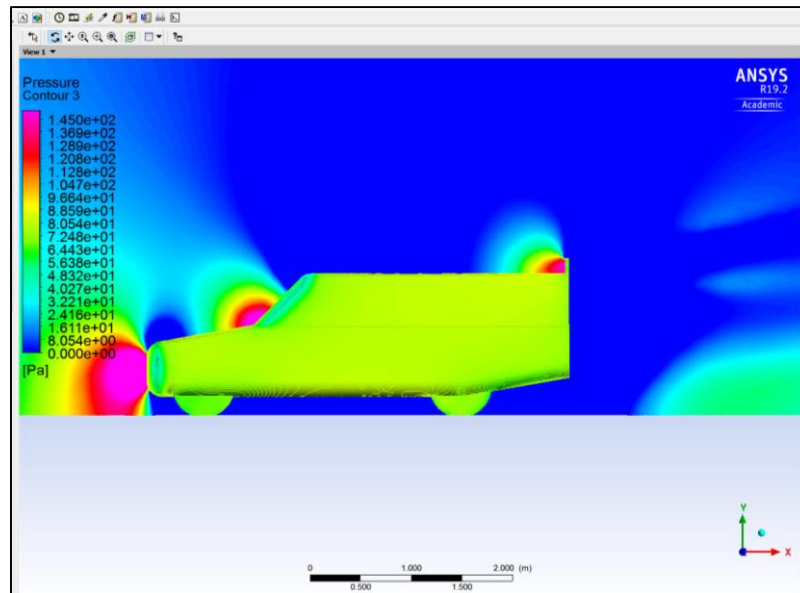


Figure 53 - A visual representation of high pressure forming in front of the spoiler. Estate model shown

However, when the 50mm spoiler was attached to the fastback model, the coefficient of lift produced was worse than with no spoiler fitted. When looking at the areas of high pressure produced from the spoiler, there is an area of low pressure next to the spoiler, connected to the high pressure produced by the spoiler. When comparing to the pressures produced from the 100mm spoiler, shown in the diagrams below, the area of high pressure in front of the spoiler is not connected to any other area of low or high pressure:

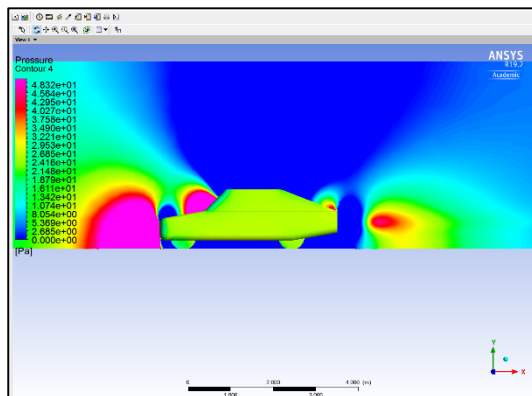


Figure 54 - A visual representation of the pressure distribution around the fastback model with 50mm spoiler

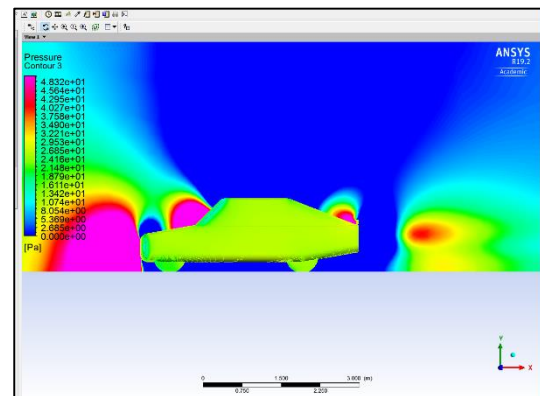


Figure 55 - A visual representation of the pressure distribution around the fastback model with 100mm spoiler

4.3.2 Drag

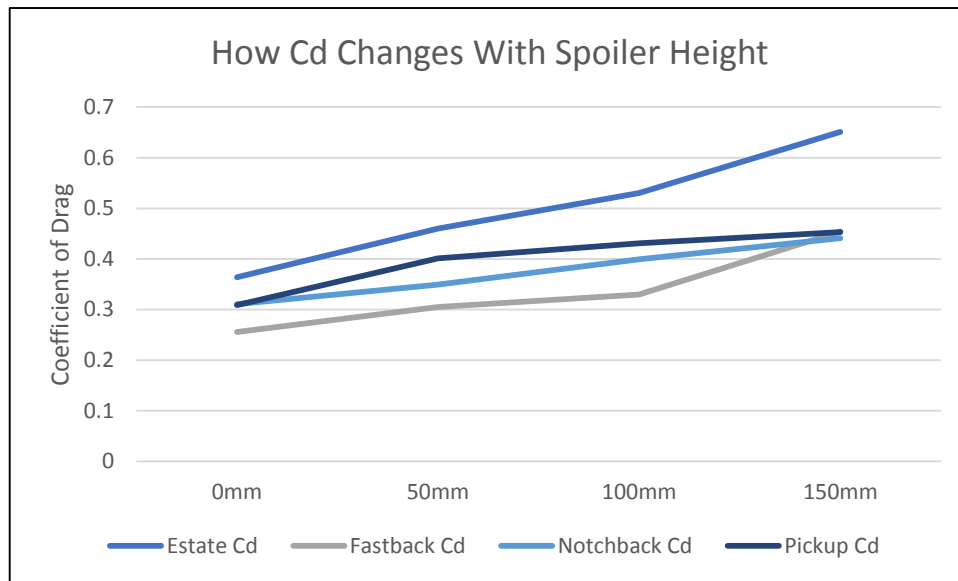


Figure 56 - A graph showing the change in coefficient of drag for changing heights of spoiler for different vehicle shapes

Looking at figure 56, the results follow the expected pattern, as with the increasing height of spoiler provides more obstruction in the direction the air travels. Out of the four vehicle shapes tested, the estate and fastback models suffered the most from an increase in the coefficient of drag, as the direction of the air flows straight into the path of the spoiler. Whereas, for the notchback and pickup models, the path the air follows does not flow directly through the spoiler, meaning the increase in coefficient of drag is not as drastic, when compared to the other vehicle shapes, an example shown below:

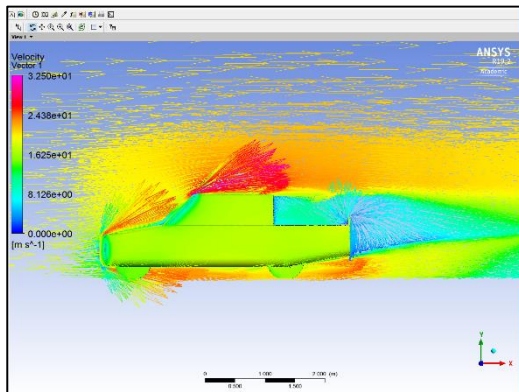


Figure 57 - A visual representation of the direction of the air flow around the pickup model, fitted with 150mm spoiler

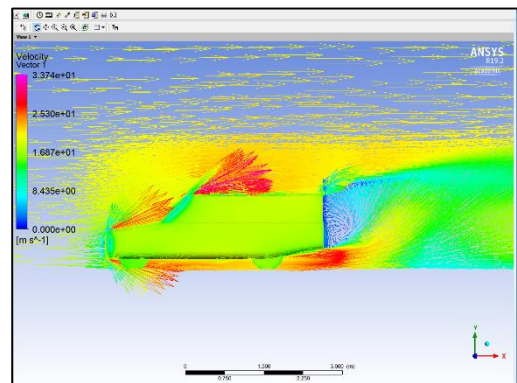


Figure 58 - A visual representation of the direction of the air flow around the estate model, fitted with 150mm spoiler

4.3.3 Centre of Pressure

	Centre of Pressure (%)			
	Notchback	Fastback	Pickup	Estate
0mm		51.05	62.67	52.63
50mm	56.17	54.99	61.69	53.81
100mm	57.24	55.58	61.88	54.2
150mm	59.16	57.94	62.47	54.01

Table 9 - The calculated centre of pressures for the four vehicle shapes fitted with spoilers of varying heights

Looking at the results in table 9, for three vehicle shapes; notchback; fastback; and estate, the centre of pressure moves towards the front of the vehicle, as the height of the spoiler increases. The fastback model receives the greatest change in centre of pressure, moving forward by almost 8%. On the other hand, the centre of pressure for the pickup model moves fractionally towards the rear of the vehicle.

5.0 Conclusion

Through all the investigations carried out, the shape of vehicle has a direct effect on the coefficient of lift and drag produced, with the fastback shape having a smaller value for coefficient of drag compared to the others. However, the reduced coefficient of drag means the coefficient of lift produced by the fastback model is worse compared to the other variants.

With respect to the aerodynamic devices investigated, the addition of a spoiler to the rear of the vehicle provided an enhancement of the coefficient of lift, but consequently meaning an increase in the coefficient of lift. The addition of an air dam to the different models increased the coefficient of drag and lift, but as the length of the air dam was increased, the coefficients for lift and drag did improve. Neither of the aerodynamic devices aided in the reduction of coefficient of drag.

6.0 Evaluation

To further enhance the validity of results gathered from the investigations carried out, wind tunnel testing could have been utilised, by using 3D printed models of the four different vehicle models. As well as this, more CFD tests could have been carried out, to find the ideal length for the air dam in which the coefficient of lift and drag would both be enhanced. Additionally, CFD tests could be carried out to investigate whether changing the angle at which the spoiler was positioned could improve the coefficient of lift and drag values. Furthermore, additional CFD tests could be made to investigate the use more aerodynamic devices, such as a splitter.

7.0 Bibliography

Ahmad NE, A.-S. E. & G. A., 2010. Mesh optimization for ground vehicle. *CFD Letters*, Volume 2, pp. 54-65.

Angelina I. Heft, T. I. a. N. A. A., 2012. *Introduction of a New Realistic Generic Car Model for Aerodynamic Investigations*, s.l.: SAE International.

Christian Bach Rasmussen, L. M. L., 2018. *Race Car Aerodynamic Design and Optimization via CFD and the Discrete Adjoint Method*. s.l.:Aalborg University.

Clancy, L. J., 1975. *Aerodynamics*. s.l.:Pitman Publishing Limited.

D Wood, M. A. P. A. K. P., 2014. *Experimental Data for the Validation of Numerical Methods - SAE Reference Notchback Model*, s.l.: SAE International.

Electropaedia, 2005. *Aerodynamic lift and drag and the theory of flight*. [Online]
Available at: https://www.mpoweruk.com/flight_theory.htm
[Accessed 11 11 2018].

Engineering Toolbox, 2003. *Reynolds Number*. [Online]
Available at: https://www.engineeringtoolbox.com/reynolds-number-d_237.html
[Accessed 25 10 2018].

Francesco Mariani, F. R. N. B. F. C. a. L. S., 2016. *Spoilers Optimization to Reduce the Induced Stresses on a Racing Helmet*, s.l.: SAE International.

Garry K.P, W. S. C. K. A. a. W. D., 1994. *The Effect on Longitudinal Position of a Road Vehicle Model in a Wind Tunnel Test Section*, s.l.: SAE International.

Geoffrey M. Le Good, K. P. G., 2004. *On the Use of Reference Models in Automotive Aerodynamics*, s.l.: SAE International.

Hall, N., 2015. *Boundary Layer*. [Online]
Available at: <https://www.grc.nasa.gov/www/k-12/airplane/boundlay.html>
[Accessed 11 11 2018].

Hall, N., 2015. *What is Lift?*. [Online]
Available at: <https://www.grc.nasa.gov/www/k-12/airplane/lift1.html>
[Accessed 11 11 2018].

Howell, J., 1994. *The Influence of Ground Simulation on the Aerodynamics of Simple Car Shapes with an Underfloor Diffuser*. s.l., Royal Aeronautical Society.

Jeffrey Hoffman, B. M. S. A. J. W. a. S. W., 2001. *Effect of Test Section Configuration on Aerodynamic Drag Measurements*, s.l.: SAE International.

Katz, J., 2016. *Automotive Aerodynamics*. illustrated ed. s.l.:John Wiley & Sons.

Keith Douglass, J. C. D. K. S. M. E. Y., N.D. *Carbon Emissions Reduction via Increased Fuel Economy*. [Online]
Available at:
<file:///E:/Final%20Year%20Project/Research/Carbon%20Emissions%20Reduction%20via%20Increase>

[d%20Fuel%20Economy.pdf](#)

[Accessed 11 11 2018].

Kentaro Machida, M. K. a. A. O., 2015. *Aerodynamic Development of the New Honda FIT/JAZZ*, s.l.: SAE International.

Kevin R. Cooper, T. B. G. D. J. S. G. S., 1998. *The Aerodynamic Performance of Automotive Underbody Diffusers*, s.l.: SAE International.

Kronbichler, M., 2010. *Computational fluid dynamics*. [Online]

Available at: file:///E:/Final%20Year%20Project/Research/CFD_notes.pdf

[Accessed 11 11 2018].

Mclean, D., 2012. *Understanding Aerodynamics: Arguing from the Real Physics*. s.l.:John Wiley & Sons.

Nuclear Power, N.D. *Reynolds Number*. [Online]

Available at: <https://www.nuclear-power.net/nuclear-engineering/fluid-dynamics/reynolds-number/>

[Accessed 25 10 2018].

Princeton, N.D. *Bernoulli's Equation*. [Online]

Available at: https://www.princeton.edu/~asmits/Bicycle_web/Bernoulli.html

[Accessed 25 10 2018].

Pruthviraj Mohanrao Palaskar, V. K. a. R. V., 2016. *Methodology Development to Accurately Predict Aerodynamic Drag and Lift for Passenger Vehicles Using CFD*, s.l.: SAE International.

RAC, 2018. *Euro 1 to Euro 6 video guide – find out your vehicle's emissions standard*. [Online]

Available at: <https://www.rac.co.uk/drive/advice/emissions/euro-emissions-standards/>

[Accessed 11 11 2018].

Renan Francisco Soares, A. K. S. G. O. K. G. a. J. H., 2018. *On the Aerodynamics of an Enclosed-Wheel Racing Car: An Assessment and Proposal of Add-On Devices for a Fourth, High-Performance Configuration of the DrivAer Model*, s.l.: SAE International.

Renan Francisco Soares, F. J. d. S., 2015. *Influence of CFD Setup and Brief Analysis of Flow Over a 3D Realistic Car Model*, s.l.: SAE International.

Stephane Cyr, K.-D. I. a. S.-H. P., 2011. *Accurate Reproduction of Wind-Tunnel Results with CFD*, s.l.: SAE International.

Strachan R.K, K. K. a. L. N., 2007. *The vortex structure behind an Ahmed reference model in the presence of a moving ground plane*. s.l.:Springer.

Technical F1 Dictionary, N.D. *Boundary layer*. [Online]

Available at: http://www.formula1-dictionary.net/boundary_layer.html

[Accessed 11 11 2018].

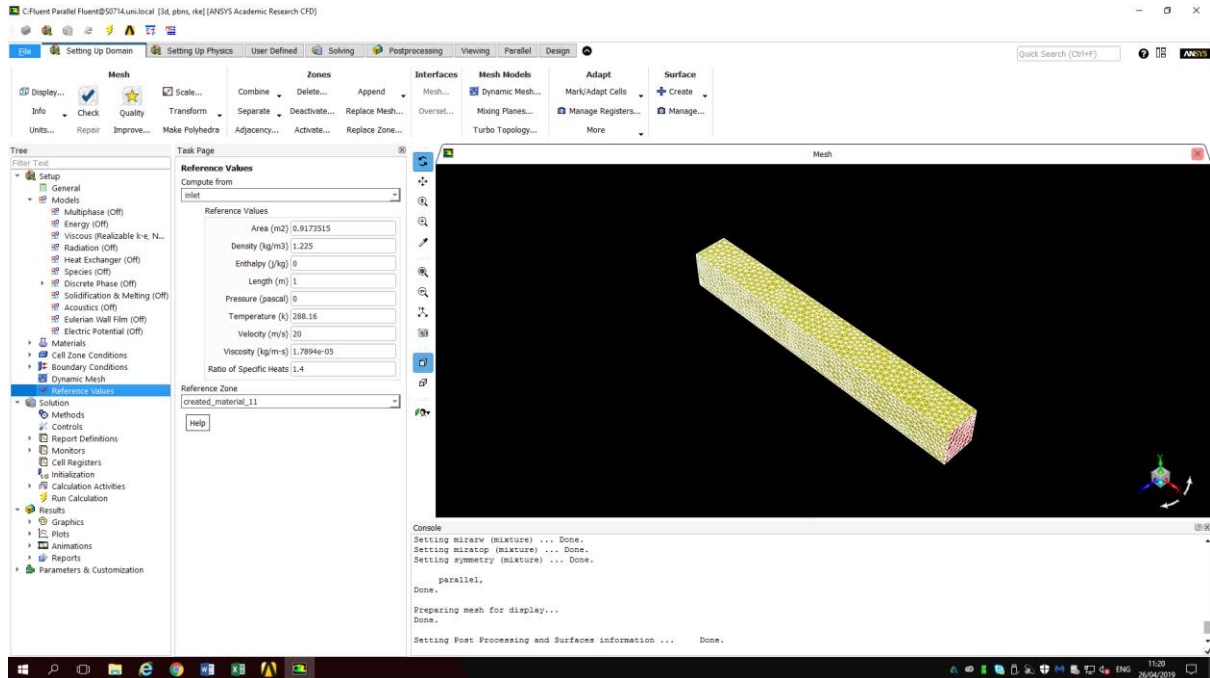
V. Naveen Kumar, K. L. N. L. N. V. N. R. a. Y. S. R., 2015. Investigation of Drag and Lift Forces over the Profile of Car with Rears spoiler using CFD. *International Journal of Advances in Scientific Research*.

White, F. M., 2011. *Fluid Mechanics*. 7th ed. s.l.:McGraw-Hill.

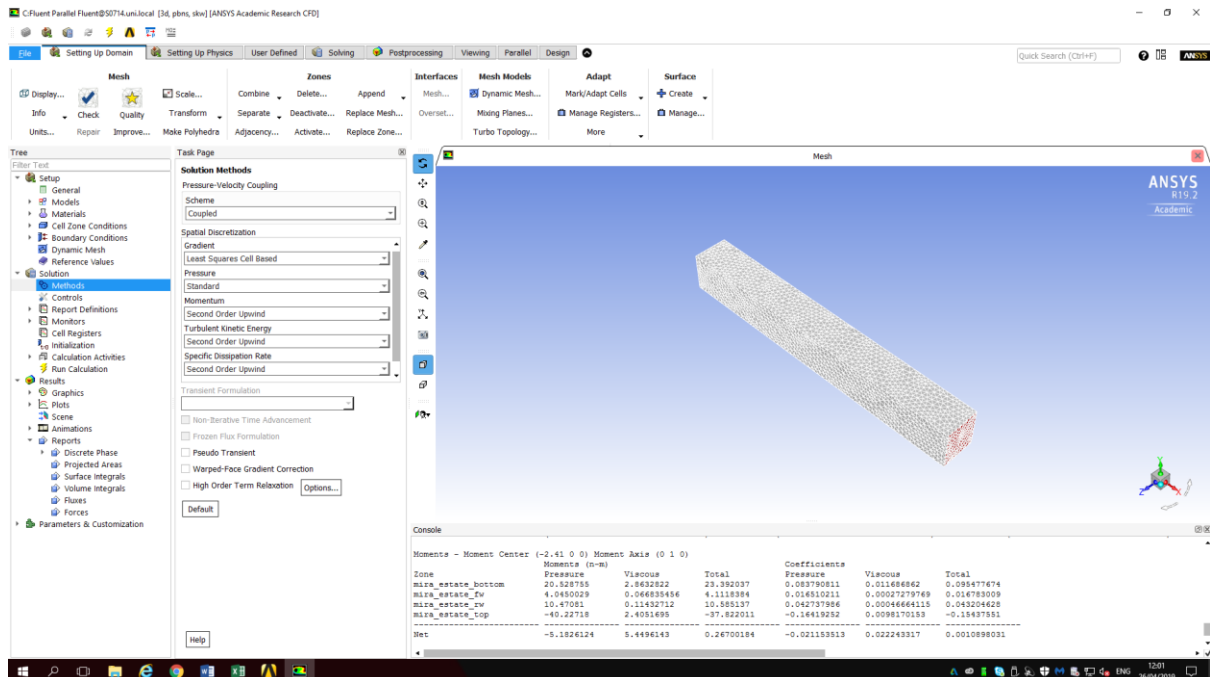
Wilcox, D. C., 2000. *Basic Fluid Mechanics*. 2, illustrated ed. s.l.:DCW Industries.

Yingchao Zhang, W. D. a. Y. Z., 2014. *Aerodynamic Shape Optimization Based on the MIRA Reference Car Model*, s.l.: SAE International.

8.0 Appendix



Appendix 1 - Reference values used during CFD process



Appendix 2 - Solution methods used for CFD process

Centre of Pressure

Wetted Area = 1079.10mm

Manometer readings

Actual wetted area = 1079.10mm

Wetted Area

$W = 2.4 \rightarrow 1073.66$
 $2.4 \times 1073.66 = 2.594784 / 2.53925$
 $\times 100\% = 52.63\%$

Fastback

$W = 2.37 \rightarrow 1073.66$
 $2.37 \times 1073.66 = 2.546374 / 2.33915$
 $\times 100\% = 51.05\%$

Pump

$W = 2.665 \rightarrow 1073.66$
 $2.665 \times 1073.66 = 2.861344 / 2.53925$
 $\times 100\% = 62.67\%$

Esplanade

100mm curdam = -2.469 -1.32434 / 2.53925 = 52.55%

100mm curdam = -2.435 -1.36134 / 2.53925 = 53.61%

300mm curdam = -2.21 -1.12634 / 2.53925 = 44.75%

100mm spoiler = -2.44 -1.36634 / " = 53.81%

100mm spoiler = -2.43 -1.31634 / " = 54.20%

150mm spoiler = -2.445 -1.31134 / " = 54.01%

Fastback

100mm curdam = -2.445 -1.34134 / " = 52.82%

200mm curdam = -2.44 -1.36134 / " = 53.81%

300mm curdam = -2.158 -1.08434 / " = 42.70%

50mm spoiler = -2.47 -1.39634 / " = 54.99%

100mm spoiler = -2.485 -1.41134 / " = 55.58%

150mm spoiler = -2.545 -1.47134 / " = 57.94%

41 | Page

Pachylodonta			
ICOM - Curdam =	-2.463	-1.37328	/ 3.53525 = 59.83%
ICOM - Curdam =	-2.545	-1.47124	/ 3.53525 = 53.54%
ICOM - Curdam =	-2.545	-1.47124	/ 3.53525 = 53.54%
ICOM - Curdam =	-2.545	-1.47124	/ 3.53525 = 53.54%
ICOM - Spelae =	-2.527	-1.45378	/ 3.53525 = 53.33%
ICOM - Spelae =	-2.527	-1.45378	/ 3.53525 = 53.33%
ICOM - Spelae =	-2.527	-1.45378	/ 3.53525 = 53.33%
Pickup			
ICOM - Curdam =	-2.641	-1.54624	/ 3.53525 = 60.52%
ICOM - Curdam =	-2.641	-1.54624	/ 3.53525 = 60.52%
ICOM - Curdam =	-2.641	-1.54624	/ 3.53525 = 60.52%
ICOM - Spelae =	-2.641	-1.54624	/ 3.53525 = 60.52%
ICOM - Spelae =	-2.641	-1.54624	/ 3.53525 = 60.52%
ICOM - Spelae =	-2.641	-1.54624	/ 3.53525 = 60.52%

Appendix 5 - Hand calculations used to generate centre of pressure percentages