

Aerodynamic Vehicle Testing and Analysis

by Amy Cosford

FILE	6618_AMY_COSFORD_AERODYNAMIC_VEHICLE_TESTING_AND_ANALYSIS_IS_128450_597185645.DOCX (286)	WORD COUNT	7948
TIME SUBMITTED	27-JAN-2019 10:05PM (UTC+0000)	CHARACTER COUNT	56967
SUBMISSION ID	99883088		



PRIFYSGOL CYMRU

Y Drindod Dewi Sant

UNIVERSITY OF WALES

Trinity Saint David

Aerodynamic Vehicle Testing and Analysis

Computational Fluid Dynamics

Amy Cosford

P150093

Module Code: M6X00408

Tim Tudor

25/01/19

1. INTRODUCTION	5
1.1 AIMS	5
2. BACKGROUND RESEARCH	7
2.1 LAMINAR VS TURBULENT FLOW	7
2.2 DRAG	7
2.2.1 <i>Bernoulli's Equation</i>	7
2.3 LIFT	10
2.4 OTHER VALIDATION STUDIES	11
3. CFD METHODS	12
3.1 OVERVIEW	12
3.2 DOMAIN STUDY	13
3.3 INFLATION LAYER STUDY	18
3.4 MESHING METHODS	22
3.4.1 <i>Quick Delaunay vs Robust Octree Volume Mesh</i>	22
3.4.2 <i>Mesh Controls</i>	23
3.4.3 <i>Mesh Quality</i>	25
3.4.4 <i>Mesh Study</i>	27
3.5 TURBULENCE MODELS	30
3.5.1 <i>Model Types</i>	30
3.6 WALL TREATMENTS	31
3.6.1 <i>Standard Wall Functions</i>	32
3.6.2 <i>Non-Equilibrium Wall Treatments</i>	32
3.6.3 <i>Enhanced Wall Treatments</i>	33
3.7 TURBULENCE STUDY RESULTS	33
3.8 FINAL BASELINE SETTINGS	37
4. AERODYNAMIC MODIFICATION	38
5. CONCLUSION	41
6. FURTHER WORK	41
7. REFERENCES	42
8. APPENDIX	ERROR! BOOKMARK NOT DEFINED.

P150093

FIGURE 1: MIRA NOTCHBACK REFERENCE CAR.....	6
FIGURE 2: STAGNATION POINT (AVIATION, 2014).....	8
FIGURE 3: PRESSURE TAP LOCATIONS FOR THE MIRA REFERENCE CAR (GAYLARD AND BAXENDALE 1998)	8
FIGURE 4: LEFT: PERFECT THEORETICAL FLOW PATTERN, RIGHT; REALISTIC FLOW PATTERN (BERNARD 2001)	9
FIGURE 5: AIRFOIL	10
FIGURE 6: AHMED BODY GEOMETRY (USER: CHB 2005).....	11
FIGURE 7: MIRA NOTCHBACK GEOMETRY (PALASKAR, KUMAR AND VAIDYA 2016).....	12
FIGURE 8: DOMAIN STUDY RESULTS.....	14
FIGURE 9: CASE COMPARISON (BELOW) OF PRESSURE DISTRIBUTION OF DOMAIN 4 (TOP LEFT) AND MIRA TUNNEL DIMENSIONS (TOP RIGHT).....	16
FIGURE 10: CASE COMPARISON VELOCITY CONTOURS SHOWING MESH DETAIL OF DOMAIN 4 (TOP LEFT) AND MIRA TUNNEL DOMAIN (TOP RIGHT).....	17
FIGURE 11: ILLUSTRATION OF BOUNDARY LAYER SEPARATION (ELGER, ET AL. 2013).....	18
FIGURE 12: PRISMS LAYER STUDY RESULTS	19
FIGURE 13: PRISM LAYER GROWTH (LANFRIT 2005)	19
FIGURE 14: CASE COMPARISON OF 1 PRISM LAYER (LEFT) AND 6 PRISM LAYERS (RIGHT).....	20
FIGURE 15: VELOCITY CASE COMPARISON BETWEEN 1 PRISM LAYER AND 6 PRISM LAYERS	21
FIGURE 16: PRESSURE CONTOUR CASE COMPARISON OF 1 PRISM (LEFT) AND 6 PRISM LAYERS (RIGHT).....	21
FIGURE 17: DELAUNAY TRIANGULATION METHOD.....	22
FIGURE 18: UNSTRUCTURED DELAUNAY MESH (LEFT) AND STRUCTURE ROBUST OCTREE MESH (RIGHT).....	23
FIGURE 19: MESH REFINEMENT USING LOCAL MESH CONTROLS (LANFRIT 2005)	23
FIGURE 20: SCALE OF SKEWNESS (ANSYS 2009).....	25
FIGURE 21: SKEWNESS BETWEEN PRISM LAYERS AND TETRA MESH	25
FIGURE 22: BAD ORTHOGONAL QUALITY AROUND WHEELS	26
FIGURE 23: ORTHOGONAL QUALITY VECTOR REPRESENTATION (SHARCNET N.D.)	26
FIGURE 24: MESH STUDY 1 RESULTS	27
FIGURE 25: MESH STUDY 2 RESULTS	28
FIGURE 26: PART MESH 0.05 (LEFT) VS 0.02 (RIGHT)	29
FIGURE 27: BOUNDARY LAYER (ANSYS 2012).....	31
FIGURE 28: GRAPH OF Y+ AND U+ (ANSYS 2012).....	32
FIGURE 29: TURBULENCE STUDY RESULTS.....	33
FIGURE 30: STANDARD K-EPSILON REAR VELOCITY FIELD (LEFT) VALIDATED CASE VELOCITY FIELD (RIGHT)	35
FIGURE 31: STANDARD K-EPSILON VELOCITY VECTORS.....	35
FIGURE 32: RNG K-EPSILON TURBULENCE PROPERTIES.....	36
FIGURE 33: STANDARD K-EPSILON TURBULENCE PROPERTIES	36
FIGURE 34: CASE COMPARISON OF K-OMEGA BSL (LEFT) AND K-EPSILON STANDARD (RIGHT) VELOCITY PROFILES	37
FIGURE 35: MIRA NOTCHBACK REAR WING MODIFICATION	38
FIGURE 36: REAR WING PROFILE.....	38
FIGURE 37: MODIFIED MIRA NOTCHBACK PRESSURE CONTOURS	39
FIGURE 38: MODIFIED MIRA NOTCHBACK VELOCITY CONTOURS	39
FIGURE 39: MODIFIED MIRA NOTCHBACK TURBULENCE PROFILE	40

P150093

GRAPH 1: MIRA PRESSURE TAP DATA	6
GRAPH 2: DOMAIN STUDY DRAG RESULTS	14
GRAPH 3: DOMAIN STUDY LIFT RESULTS	15
GRAPH 4: PERCENTAGE ERROR FOR DRAG AND LIFT RESULTS ACHIEVED IN COMPARISON TO MIRA RESULTS.....	15
GRAPH 5: BEST DOMAINS PRESSURE TAP DATA COMPARISON	16
GRAPH 6: LIFT VALUES WITH INCREASING PRISM LAYERS.....	19
GRAPH 7: DRAG VALUES WITH INCREASING PRISM LAYERS	20
GRAPH 8: RESULTANT DRAG COEFFICIENTS WITH DECREASING PART MESH	27
GRAPH 9: RESULTANT LIFT COEFFICIENT WITH DECREASING PART MESH	27
GRAPH 10: MESH STUDY 2 LIFT COEFFICIENT RESULTS	28
GRAPH 11: MESH STUDY 2 DRAG COEFFICIENT RESULTS.....	29
GRAPH 12: STANDARD K-EPSILON LOWER PRESSURE TAP DATA	34
GRAPH 13: STANDARD K-EPSILON UPPER PRESSURE TAP DATA	34

1. Introduction

Vehicle aerodynamics continues to be one of the most crucial areas for development in racing and automotive disciplines. Before computational methods were readily available, aerodynamic improvements would have been limited to rigorous wind tunnel testing on a trial and error basis and any mathematical analysis could only be done with hand calculations. Nowadays Computational Fluid Dynamics (CFD) software is utilised to conduct comprehensive optimisation of aerodynamic designs. ANSYS is one of many integrated software's allowing the user to create simulated domains imitating what would be a real world wind tunnel. By inputting accurate vehicle geometry the user is then able to create a "mesh" of both the surrounding domain and vehicle geometry, methods of which will be discussed in the relevant subsections of this report. The turbulence model chosen by the user largely affects how the software calculates relative values such as pressure distribution, drag, and lift. Time and processor power are just some of the limitations of CFD software, such that large domains, fine meshes, and multi-equation turbulence models take much longer to solve. CFD software is generally governed by strict mathematical methods whereas real world aerodynamics is extremely complex and cannot easily be boxed into the specific methods used in CFD modelling.

1.1 Aims

CFD studies firstly require the validation of results by following what is known as the MIRA Reference Car Study. This is an investigation into the effects of certain parameters on MIRA's reference car geometry and domain with the objective of producing drag and lift values within close proximity to those achieved in MIRA's wind tunnel. The MIRA Notchback car was chosen for this study due to its importance in the calibration of other wind tunnels and uses in many validation studies.

It is the aim of this study to closely replicate drag and lift results produced by MIRA by conducting studies on domain size, boundary layer conditions, mesh optimisation and turbulence model. The resultant drag and lift values of each run will be plotted on a table to observe the effect of the changed variable in that particular study. During the processing phase of each study, figures will be produced of pressure contours, turbulence contours and velocity vectors in order to analyse the model's flow characteristics. Pressure tap data from each study will be compared to pressure tap data obtained from the MIRA reference car and results collated for analysis.

Once optimal settings have been found, an aerodynamic modification will be made to the MIRA reference car with the aim to either reduce drag force and/or increase downforce. The updated geometry will then be run with ideal settings to discover whether the modification had a positive or negative affect on the objective outcome.

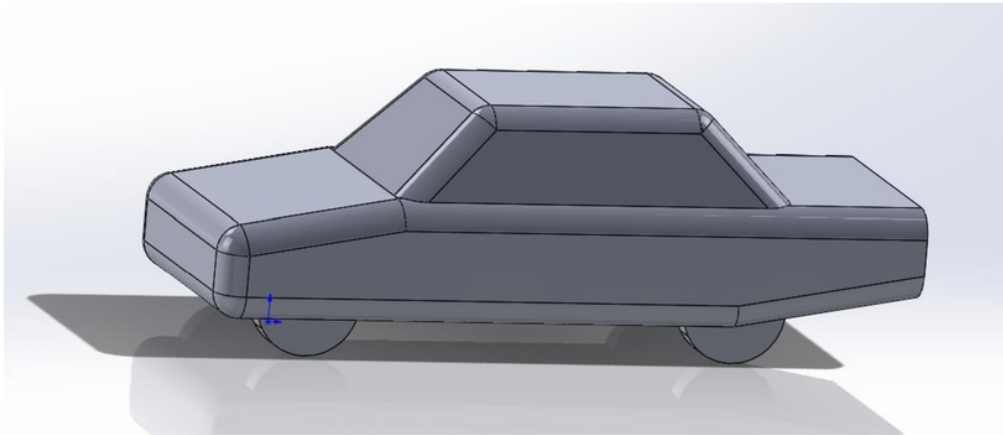
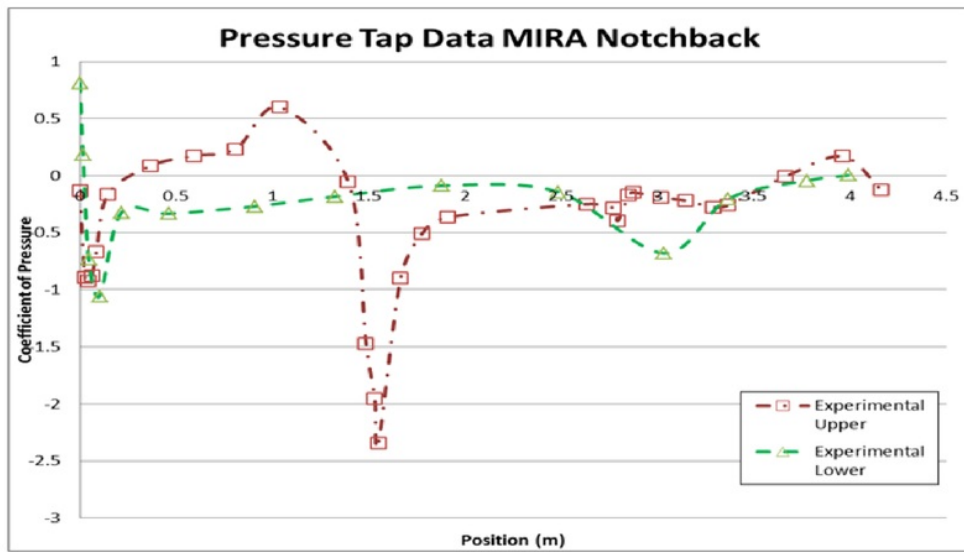


Figure 1: MIRA Notchback reference car



Graph 1: MIRA Pressure tap data

2. Background research

2.1 Laminar vs Turbulent Flow

Flow characteristics can be divided into 2 main bodies of flow: laminar and turbulent, and their classification depends on the magnitude of Reynolds number. Reynolds number describes the ratio of inertial forces and viscous forces, such that:

$$Re = \frac{\text{inertial forces}}{\text{viscous forces}} = \frac{\rho v L}{\mu} \quad [1]$$

Where ρ is the fluid density, v is the fluid velocity, L is the distance travelled by the fluid, and μ is the dynamic viscosity (Anderson 1984).

Laminar flow occurs at low Reynolds numbers of below 2000 (Elger, et al. 2013) and describes an organised state of flow whereby neighbouring layers of fluid move smoothly and parallel to each other. Fluids in laminar flow do not create any eddies or swirls and flow is very uniform.

Turbulent flow however is a chaotic state of flow characterised by intense cross-stream mixing, eddies of various size and non-uniform areas of varying pressure and velocity. Turbulent flows occur at Reynolds numbers above 3000.

An unpredictable state of flow between laminar and turbulent occurs at Reynolds numbers between 2000 and 3000, and is known as the transitional stage. Suggested in the name it describes the flow phases where the fluid is transitioning from laminar to turbulent.

Using equation 1 it can be calculated that for an air velocity of approximately 20 m/s at 20°C over a distance of 4 metres: the approximate length of a car, the Reynolds number can be calculated as:

$$Re = \frac{\rho v L}{\mu} = \frac{1.204 \times 20 \times 4}{1.825 \cdot 10^{-5}} = 5.28 \cdot 10^6$$

Thus the flow modelled in this study is turbulent. Properties of air at 1 atmospheric pressure taken from (Engineers Edge 2000).

2.2 Drag

2.2.1 Bernoulli's Equation

Bernoulli's equation can be used to describe many aspects of aerodynamic theory and defines the relationship between air velocity and pressure.

$$P_1 + \frac{\rho V_1^2}{2} = P_2 + \frac{\rho V_2^2}{2} \quad [2]$$

$$\Delta P = \frac{1}{2} \rho (V_2^2 - V_1^2) \quad [3]$$

From equation 3 it can be said that the change in pressure is inversely proportional to the square of the air velocity. When applied to vehicle aerodynamics, understanding the effect of pressure distribution over a vehicle is crucial to comprehend the effects of drag and lift.

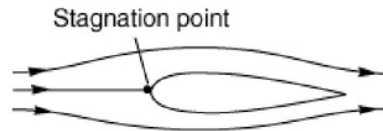


Figure 2: Stagnation Point (Aviation, 2014)

The stagnation point of a geometrical shape is the point at which the velocity is zero, and the coefficient of pressure at this point is $C_p = 1$, thus at every other velocity above 0 the coefficient of pressure will be less than 1. Regarding the MIRA reference car, the stagnation point is at the very front of the car, and pressure distribution can be measured using pressure taps along the upper and lower surfaces of the car, as shown in Figure 3.

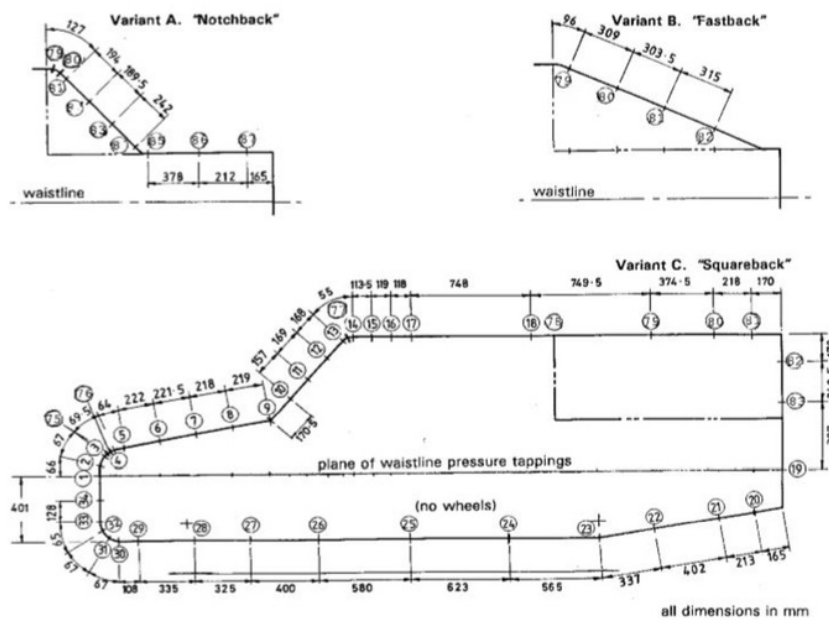


Figure 3: Pressure Tap Locations for the MIRA reference car (Gaylard and Baxendale 1998)

Drag force is the resistant force acting upon a vehicle parallel to the direction of the free stream. One component of drag force is form drag or pressure drag, whose main contribution to total drag force is in regard to the geometric properties of the vehicle (Katz 2006). Air flow characteristics along the car's upper and lower surfaces cause an area of low pressure to build up behind the car due to the effect of flow separation in the space left behind by the car, known as the wake region. If the vehicle were

P150093

theoretically perfectly streamlined, with no sharp geometrical changes and negating viscosity influences, the corresponding pressure distribution would produce equal and opposite forces on the front and rear of the car, thus producing no drag (Bernard 2001).

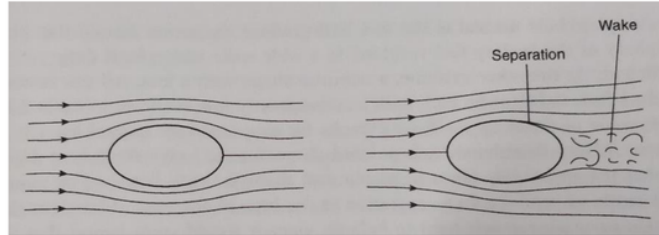


Figure 4: Left: Perfect theoretical flow pattern, right; realistic flow pattern (Bernard 2001)

However since viscosity effects cannot be ignored and vehicles inherently have complex geometry, the realistic flow patterns are more like that displayed on the right of Figure 4.

Friction or surface drag is also a contributor to total drag force, though usually to a lesser magnitude than pressure drag. Surface drag is known as the frictional shear force created by the cars surface. If the surface were to be exceptionally smooth, the friction drag would be reduced. Road cars tend to have uneven surfaces on the underside of the car making their surface drag effects considerably higher.

The coefficient of drag can be calculated by:

$$C_D = \frac{F_D}{0.5 \times \rho v^2 A} \quad [4]$$

(Katz 2006)

Where F_D is the drag force, ρ is the density of air, A is the vehicle frontal area and v is the velocity.

Reducing the drag force is necessary in order for cars to increase top speed whilst keeping the engine characteristics the same. The majority of drag force is due to pressure distribution influenced by the geometrical properties of the car, therefore drag can be reduced in a number of ways. If the car were to be designed more streamlined this would reduce the low pressure region behind the car and thus reduce drag force. The addition of an angled rear wing directs the air in a more gentle slope from the roof to wing, helping to reduce the flow separation and thus drag (Cakir 2012).

Formula 1 uses a Drag Reduction System (DRS) to encourage overtaking manoeuvres in approved DRS activation zones. The system alters the angle of the adjustable rear wing at the driver's input, reducing drag and enabling the driver to overtake the car in-front, provided that they are within a 1 second gap (Formula1 n.d.).

2.3 Lift

Lift is one of the most important considerations in aerodynamic design and is the component of force acting normal to that of the free stream. Bernoulli's principle can be applied again to explain the production of positive or negative lift, the latter more commonly known as downforce.

In the example of an airfoil shaped wing, the air flowing over the top surface has a further distance to travel from the leading edge to the trailing edge of the wing, and thus accelerates to a higher velocity than the air flowing underneath the wing. Consequently, an area of high pressure builds up underneath the wing and creates lift.

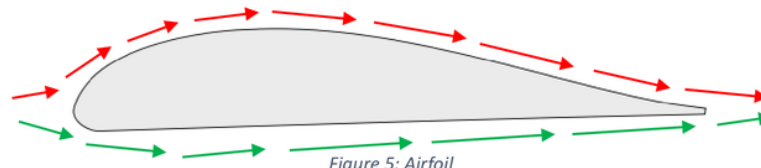


Figure 5: Airfoil

Racing disciplines make use of spoilers with airfoil designs inverse to that shown in Figure 5 in order to generate downforce rather than lift. Ensuring the underside of the vehicle is at a lower pressure is also desirable and can be achieved by designing in an airdam style front spoiler (McBeath 1999). Front spoilers block off the passage of air underneath the car and subsequently induce an area of low pressure, thus increasing downforce. Developments on the front spoiler include what is known as a *splitter*; a protruding lower lip extending outwards in front of the car. The area of high stagnation pressure at the front of the car pushes down on the splitter and aids in creating downforce without inducing drag.

Where the reduction of drag aids in increasing straight line speed, an increase in downforce is crucial to help generate grip and enable the car to accelerate round corners at a higher velocity. Thus it is the aim of racing aerodynamicists to design chassis and bodywork components with this in mind. The coefficient of lift can be calculated:

$$C_L = \frac{F_L}{0.5 \times \rho v^2 A}$$

2.4 Other Validation Studies

One such other study is that concerning the Ahmed body. The original study was regarding the investigation of drag force and explored the effects of a slanted rear-end against a vertical rear-end with slant angles increasing from 0-40° incline (Dobrev and Massouh 2014). Similar to the MIRA reference car study, the AHMED body study is used in the calibration of wind tunnels for a wider range of automotive and aviation uses such as for planes, helicopters and trucks.

From an automotive sense however, the Ahmed body is not representative of real-world vehicle geometry, hence the necessity for a more typical geometry to be created and validated against. The MIRA geometry is beneficial in that it is more realistic of modern day vehicle silhouettes yet sufficiently simple enough to recreate and explore in CFD packages.

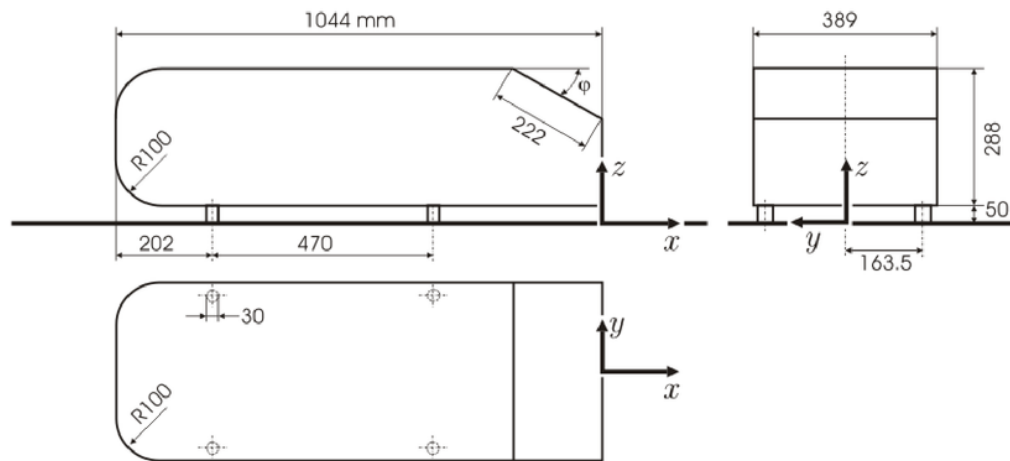


Figure 6: Ahmed Body Geometry (User:Chb 2005)

3. CFD Methods

3.1 Overview



Generally, CFD methods begin with the creation of accurate geometry and a domain in which it will be tested. In this study the MIRA car has previously been designed in Solidworks based on the dimensions shown in Figure 7, and imported into the geometry section of ANSYS. The surrounding domain representing the wind tunnel is then the subject of the first study, and is an investigation into what effect the domain volume has on the resultant coefficients of drag and lift.

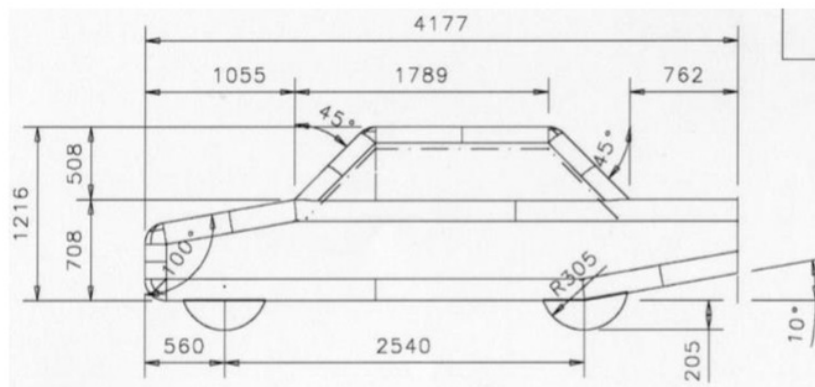
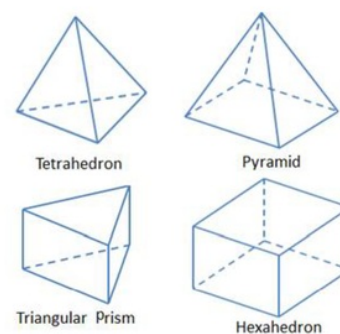


Figure 7: MIRA Notchback geometry (Palaskar, Kumar and Vaidya 2016)

The second and third studies are regarding meshing methods, with the former investigating the effects of prism layers at the boundary. Mesh generation can be described as the division of the domain into smaller elements in which the CFD software calculates for the fluid velocity and other variables in each individual cell, thus the solution combines the results to give a complete picture of fluid flow across the whole domain. Mesh generation is a constant compromise between computational costs and accuracy. The finer the mesh, the more accurate the results will be but at a cost to computational time.

Regarding the vehicle itself, local mesh controls can be used here to generate a separate mesh from the domain. Since geometric features are extremely important to the overall flow characteristics it's necessary that the mesh generation should be extremely fine and can



P150093

take long periods of time to calculate. The domain mesh doesn't necessarily have to be overly defined except in the regions close to the car. Density boxes can be created to refine the mesh in certain areas of importance. Choice of cell type in the mesh is also of importance and will be explored in its respective subsection.

Proceeding the ICEM-CFD meshing stage, the solving is then done by ANSYS' Fluent whereby user inputs influence how the solving software simulates fluid flow and the subsequent characteristics of lift and drag. One of the main influences here is the chosen turbulence model. Turbulence models determine how the solver simulates flow at the boundary walls, the number of different variables to solve for, and what these variables represent. The majority of the turbulence models under investigation are variations of the Reynolds-Averaged Navier-Stokes equations, which are based on Euler's equations regarding pressure variation, which represent the conservation of mass, momentum, and energy for incompressible inviscid flow.

If the flow was laminar, assuming that the velocity of the inlet flow does not change with respect to time, the Navier-Stokes equations can be used to predict the pressure distribution and flow velocity. However as the flow becomes turbulent the fluctuations in velocity mean it can no longer be treated as if the velocity over time. Such a scenario must then be solved using a series of time-averaged Navier-Stokes equations called Reynolds-Averaged-Navier-Stokes (RANS) formulations which observe that the flow experiences local fluctuations in velocity and can be treated in a time-average manner (Frei 2017). Different turbulence models add different variables such as for turbulent kinetic energy and specific dissipation rates, therefore the turbulence model study in this report investigates the effects of such.

3.2 Domain Study

Lanfrit notes that the recommended computational domain length should be at least 3 car lengths in front of the car, and 5 car lengths behind (Lanfrit 2005) and the car frontal area must not exceed 1.5% of the domain cross-sectional area. However there are many other studies claiming various domain sizes work best. MIRA's own full scale aerodynamic wind tunnel is 15m in length, 7.9m in width, and 4.4m in height (HORIBA MIRA n.d.), therefore it's logical to suggest that the requirement for a domain study shouldn't be necessary since the domain size tested by MIRA is already known. Although CFD has been well developed over the last decade, the complexities of aerodynamics means the computational methods still produce inaccuracies due to the many inherent variables in the simulation. Therefore conducting a domain study will give an idea of the effect of domain size and car placement on the overall lift and drag coefficients.

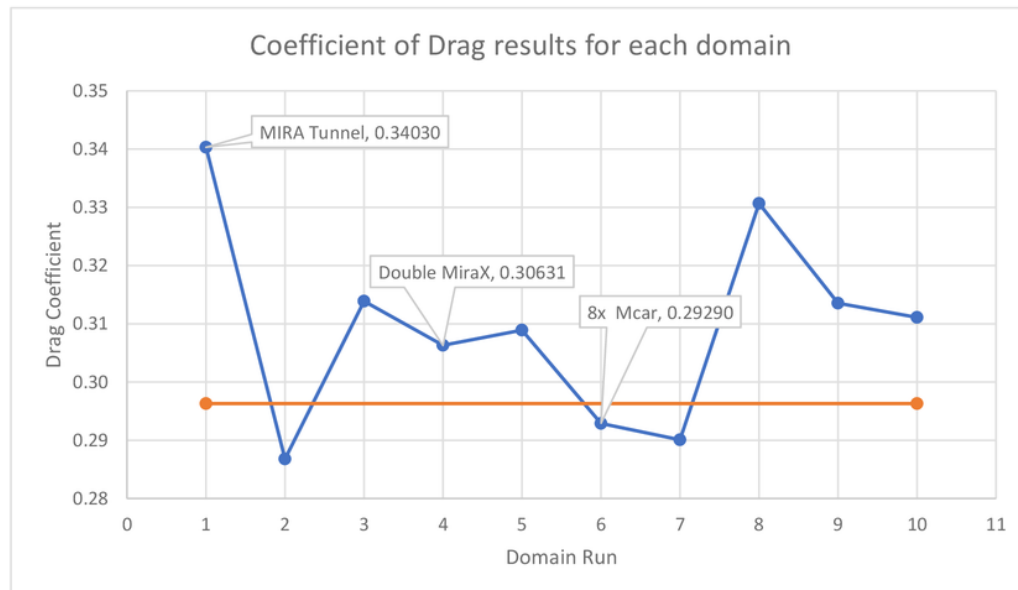
P150093

The domains tested in this study are based on the Lanfrit recommended domain size and the MIRA domain size, with adjustments made along the course of the study. Due to the symmetrical geometry of the MIRA Notchback only half of the car and domain needs to be modelled, keeping processing time and power down. For the sake of validity, density box dimensions and mesh controls are kept constant throughout the study, such that the only changes are to the domain volume and results can be analysed as such.

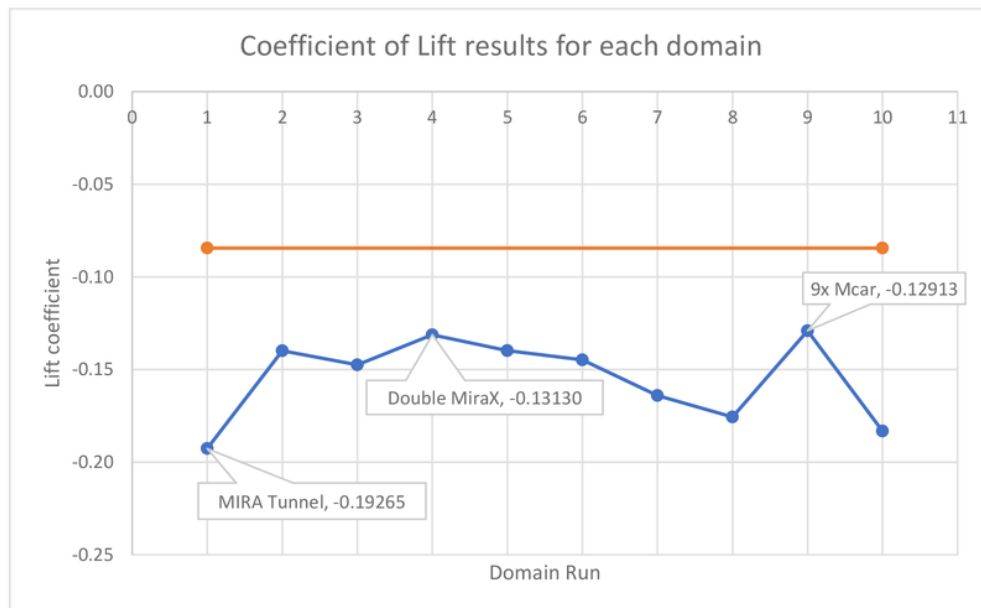
Run	Comments	Dimensions in Car Lengths (m)				MIRA Volume (m ³)	Cd 0.2963	Cl -0.0845	Cd Error %	Cl Error %	Total % Error
		In Front	Behind	Side	Above						
1	MIRA Tunnel	1	2	1	1	260.7	0.34030	-0.19265	14.85	127.99	142.838
2	5x Mcar	1	3	1	1	360.36	0.28681	-0.13990	3.20	65.56	68.765
3	6x Mcar	2.5	2.5	1	1.5	429	0.31388	-0.14747	5.93	74.52	80.454
4	Double MIRA2	2	4	1	1	514.8	0.30631	-0.13130	3.38	55.38	58.763
5	Double MIRA2	2	4	2	2.5	675	0.30892	-0.13982	4.26	65.47	69.727
6	8x Mcar	3	4	1	1.5	549.12	0.29290	-0.14481	1.15	71.37	72.520
7	Lanfrit	3	5	2	4	564.06	0.29010	-0.16408	2.09	94.18	96.270
8	Lanfrit2	3	5	3	3	1713.6	0.33065	-0.17561	11.59	107.82	119.415
9	9x Mcar	3.5	4.5	1	1	643.5	0.31355	-0.12913	5.82	52.82	58.638
10	11x Mcar	4	6	2	2	1053	0.31110	-0.18320	4.99	116.81	121.802

Figure 8: Domain Study Results

A comprehensive study was conducted into predominantly the effect of domain length on drag and lift values, though slight changes were made throughout to width (Z) and height (Y). The values achieved by MIRA in their tunnel was a drag coefficient of 0.2963, and lift -0.0845. The worst result in terms of drag was that of the MIRA tunnel dimensions whereby the drag coefficient was inaccurate to 14.85%. Shown in graph 2, a domain size of 8 car lengths proved to achieve the most accurate drag results, however figure 8 shows the value of lift was a lot more inaccurate than others.

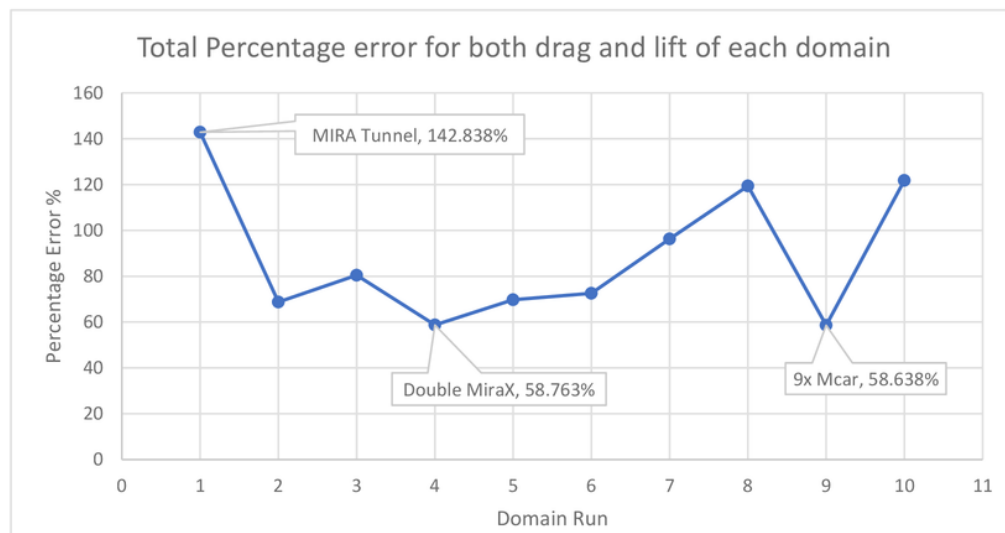


Graph 2: Domain Study Drag Results



Graph 3: Domain Study Lift results

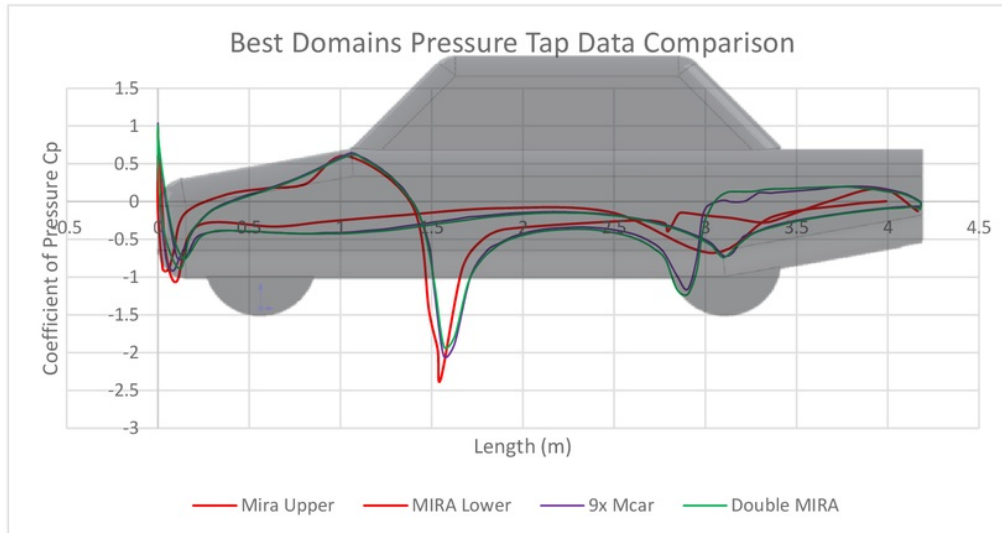
Domains 4 and 9 produced the best results of lift, whereas again the MIRA tunnel dimensions proved to be the most inaccurate. Domain 4 was double the length of the MIRA tunnel, and kept all other dimensions equal to MIRA's, whereas domain 9 was equal to 9 car lengths and with the same values of width and height. As total percentages of drag and lift error, domains 4 and 9 were accurate to within 0.1% with respect to each-other, however approximately 58% inaccurate with respect to MIRA measure values.



Graph 4: Percentage Error for drag and lift results achieved in comparison to MIRA results

P150093

Domain 4, named "Double MiraX", has a lower volume than domain 9, therefore solving time and processing power for future runs would be decreased and beneficial when it comes to refinement of the mesh and turbulence modelling. Analysis on the ANSYS Results post-processor also shows minimal differences in the pressure distribution shown in Graph 5, therefore the decision was made to use the smaller of the two domain volumes.



Graph 5: Best Domains Pressure Tap Data Comparison

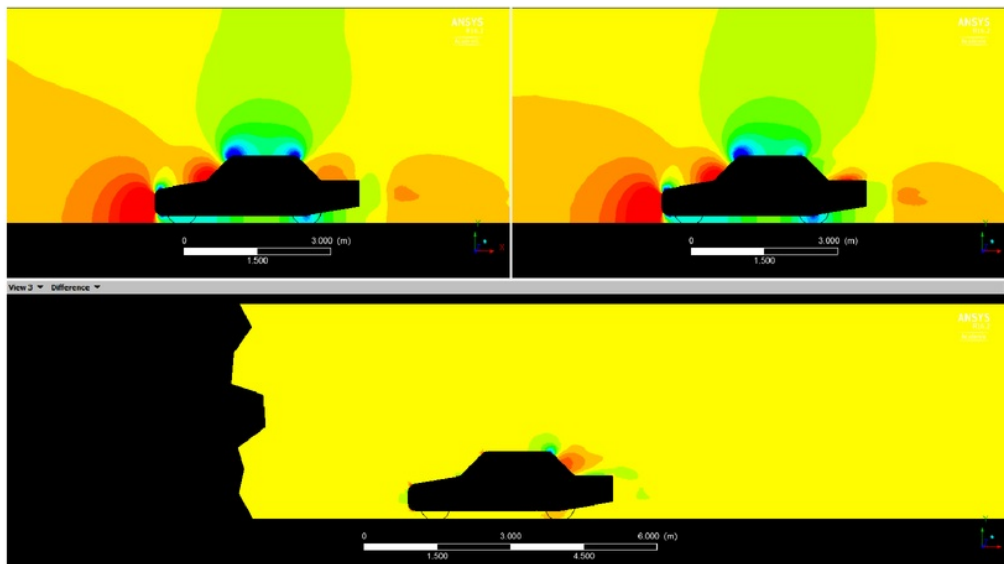


Figure 9: Case Comparison (below) of Pressure distribution of Domain 4 (top left) and MIRA tunnel dimensions (top right)

P150093

The case comparison shown in figure 9 compares the chosen domain 4 pressure distribution contours to that of the MIRA tunnel dimensions. Here, the car placement in the MIRA tunnel is clearly too far forward and does not completely capture the pressure field in front of the car.

Figure 10 shows the velocity contour differences between the 30m length domain on the top left and the MIRA tunnel dimensions top right. The velocity characteristics in the chosen domain are slightly more streamlined than the MIRA domain, due to the fact that the air has a longer distance to travel before hitting the car and is travelling faster further above in the free stream, shown in the domain pressure contours of figure 9.

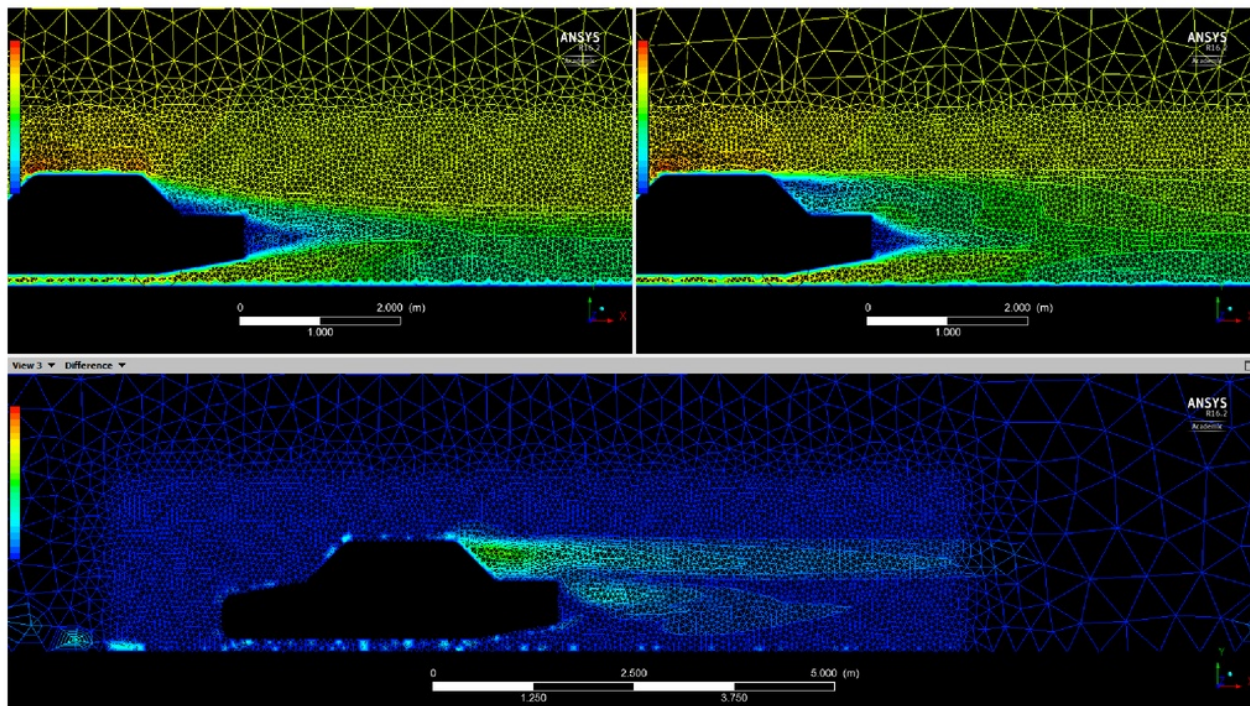


Figure 10: Case comparison velocity contours showing mesh detail of domain 4 (top left) and MIRA tunnel domain (top right)

It is important to note that the domain study was conducted with a course part mesh of 0.07 and without inflation layers at the boundary. The boundary layer between fluid flow and solid body is extremely important in the computational simulation of aerodynamics. ANSYS's ICEM CFD meshing software defaults to 3 prism layers, whereas other sources suggest 5 layers achieve the best results. Due to conflicting recommendations, the decision was made instead to conduct a separate study on the effects of inflation layers, hence the reasoning behind conducting the domain study without a representative boundary layer mesh.

3.3 Inflation Layer Study

Inflation layers act to give detail to the fluid flow at the boundary. The no-slip condition in fluid mechanics states that fluid velocity at a surface is equal to zero, thus the closer the fluid is to a surface the slower its speed (Elger, et al. 2013). Boundary layer separation is extremely difficult to simulate due to the inherent mathematical complexities. Separation occurs under adverse-pressure gradient conditions. Pressure gradients are favourable where the accelerating fluid has a higher velocity and lower pressure, overcoming the viscosity effects of the fluid. Adverse-pressure gradients are opposite to this, where the mainstream pressure increases and acts to slow down the flow in the boundary layer. As a result of this combined with the friction forces closer to the surface, the fluid velocity is decelerated sufficiently to cause the reversal of the direction of flow in the boundary layer, resulting in an established circular flow pattern. It is this flow separation that causes the eddies of flow and turbulence in the wake region (Elger, et al. 2013).

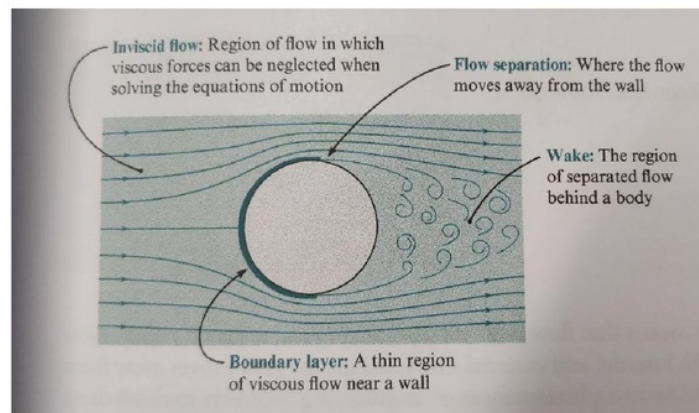


Figure 11: Illustration of boundary layer separation (Elger, et al. 2013)

In order to capture the flow details at the boundary layer the tetrahedral mesh would have to be extremely fine, since the boundary layer can be extremely thin. However this would compromise solving time due to the increased number of cells. Instead, building layers of prism cells help to capture the flow details at the boundary layer whilst still giving accurate results.

The aim of the inflation layer study is to use the ideal domain from the first study and build up prism layers until the coefficients of drag and lift get closer to those measured by MIRA. Hence, each computational run will have an added layer of prism cells.

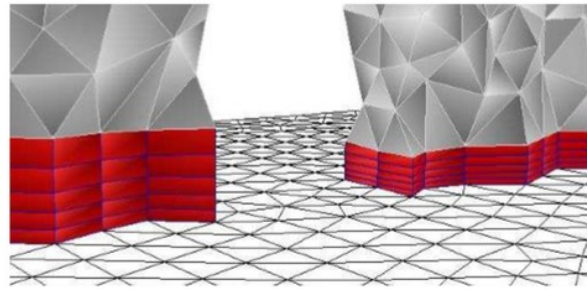
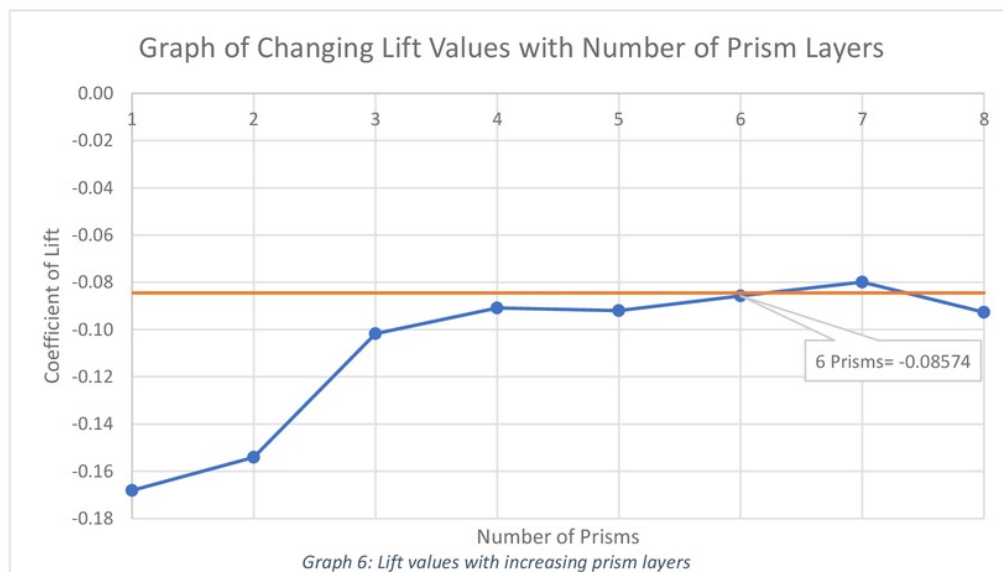


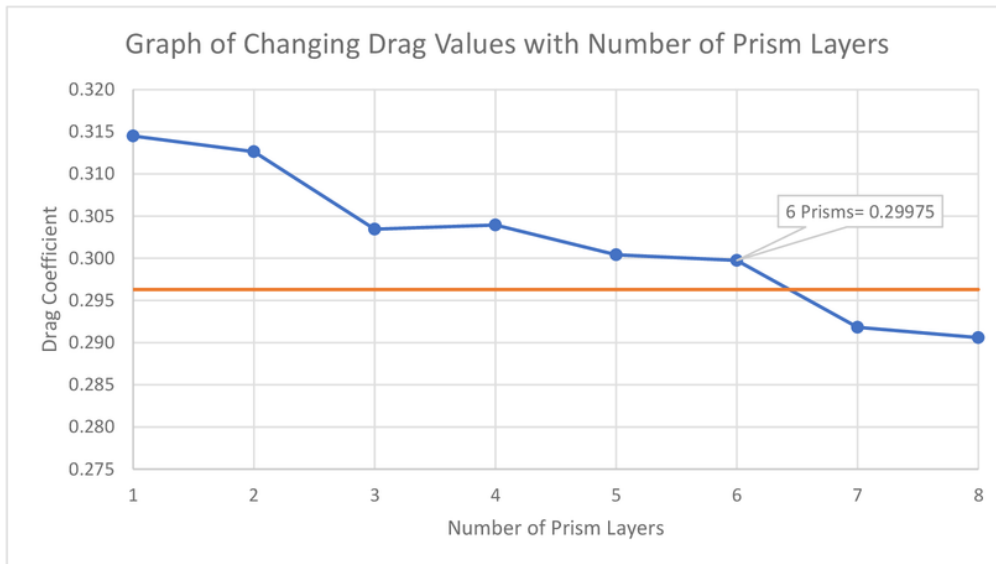
Figure 13: Prism Layer Growth (Lanfrit 2005)

Initially, the study was started with a relatively coarse part mesh of 0.06m, however as the number of prism layers increased the quality of the mesh decreased such that the file could not be loaded into ANSYS' Fluent. Analysis of the mesh in ICEM-CFD saw there was a vast number of skewed cells in the space underneath the car at the merging between prism layers and tetrahedral mesh. Therefore the part mesh underwent slight refinement throughout as a result of these errors.

Prism Layers	Part Mesh	Cd	Cl	Cd Error %	Cl Error %	Total Error %
1	0.06	0.31449	-0.16813	6.1390	98.9704	105.1095
2	0.06	0.31263	-0.15408	5.5113	82.3432	87.8545
3	0.055	0.30345	-0.10177	2.4131	20.4379	22.8510
4	0.053	0.30395	-0.09085	2.5818	7.5136	10.0955
5	0.053	0.30042	-0.09202	1.3905	8.8959	10.2863
6	0.053	0.29975	-0.08574	1.1644	1.4627	2.6271
7	0.053	0.29182	-0.07992	1.5120	5.4213	6.9333
8	0.053	0.29059	-0.09267	1.9271	9.6675	11.5946

Figure 12: Prisms Layer Study Results





Graph 7: Drag values with increasing prism layers

It was found that the mesh producing best results was that of 6 prism layers, albeit with slightly more refined local mesh controls. It produced values extremely close to that of the MIRA wind tunnel results. Coefficients of drag and lift got increasingly more accurate up until approximately 4 prism layers where the plotted results start to trail off and even out. Past 6 prism layers the results don't tend to get any more accurate but computational costs increase.

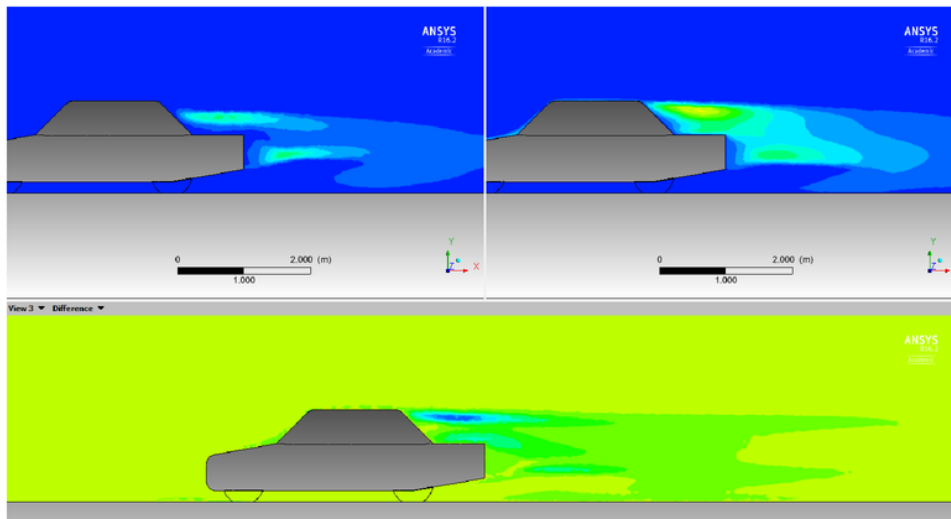


Figure 14: Case Comparison of 1 Prism layer (left) and 6 Prism Layers (right)

P150093

Figure 14 shows a case comparison between the turbulence contour results when running the mesh with 1 prism layer or 6 prism layers. With 6 prism layers the solver was able to much more accurately capture the flow separation and resultant turbulence in the wake region of the car.

In terms of velocity, as can be seen in Figure 15, the region underneath the car is much smoother and more defined with 6 prism layers and thus yielding much more accurate results. The difference in velocity is shown beneath the two comparison cases and it can be noted that the velocity underneath the car is slightly higher with 6 prisms than that of 1 prism layer. It would be assumed that since the region beneath the car seems to have higher velocities, the localised pressure in that area would be lower and thus increase the downforce. However as can be seen on the pressure comparison in Figure 16, the area around the front wheels is at a slightly lower pressure and thus could be part of the reason why the values of negative lift are more with only 1 prism than with 6.

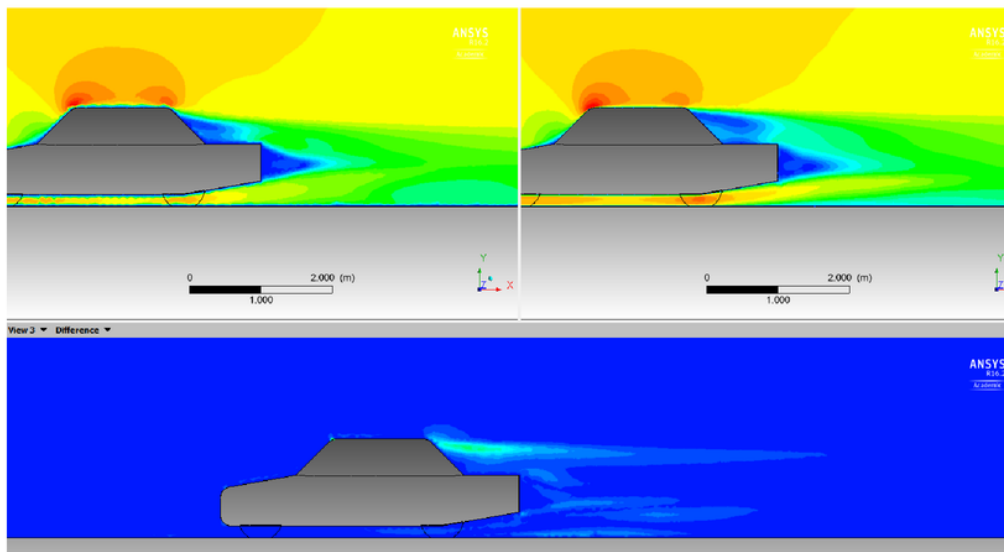


Figure 15: Velocity Case Comparison between 1 prism layer and 6 prism layers

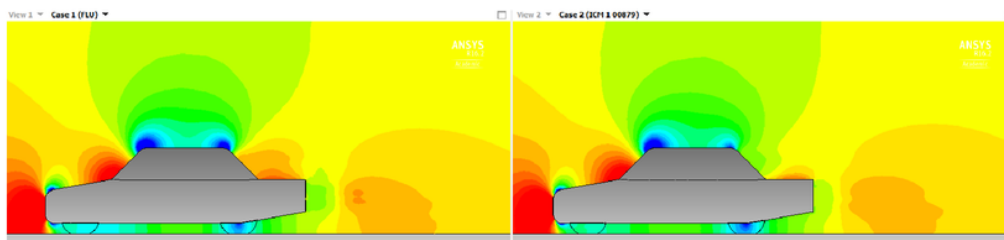


Figure 16: Pressure Contour case comparison of 1 prism (left) and 6 prism layers (right)

3.4 Meshing Methods

3.4.1 Quick Delaunay vs Robust Octree Volume Mesh

Common types of meshing methods in ICEM-CFD are a “quick Delaunay” method and “robust octree”. The Delaunay method is based on the Delaunay triangulation process, whereby tetrahedral cells are produced as a result of 3 points plotted on the circumference of a circle, such that the joining of these points create a triangle (Tucker 2016). The mesh then grows progressively from this on the basis that no nearby point is inside the circumcircle of any other triangle. A 2D representation of this is shown in figure 17, and the 3D translation simply replaces the circles with spheres and is generated in a similar fashion.

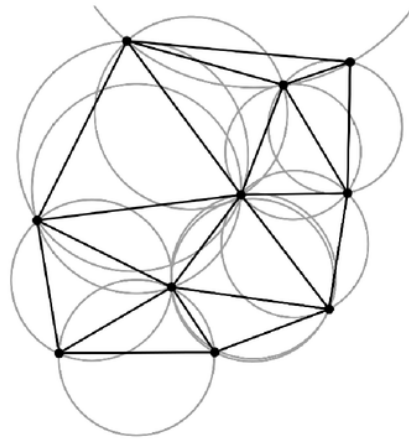


Figure 17: Delaunay Triangulation Method

The Quick Delaunay method uses a bottom-up approach to build the mesh; it is built from the bottom of the domain and grows out in an unstructured advancing front manner. Advantageously, this method is fast to compute and saves time and power constraints, however in order to produce accurate results the CAD geometry must be of very good quality and designs with complicated surfaces sometimes do not mesh properly, therefore the Robust Octree method is usually preferred.

The Robust Octree meshing process uses a top-down approach on mesh generation. By that, it produces a tetrahedral mesh with a structured approach. A grid is first produced, and tetrahedral cells are generated within the confines of these grid cells. The octree mesh algorithm then subdivides the elements in areas of necessary increased refinement, for example where the geometry becomes more complex, but maintains larger elements in the free stream flow area to keep computational costs down.

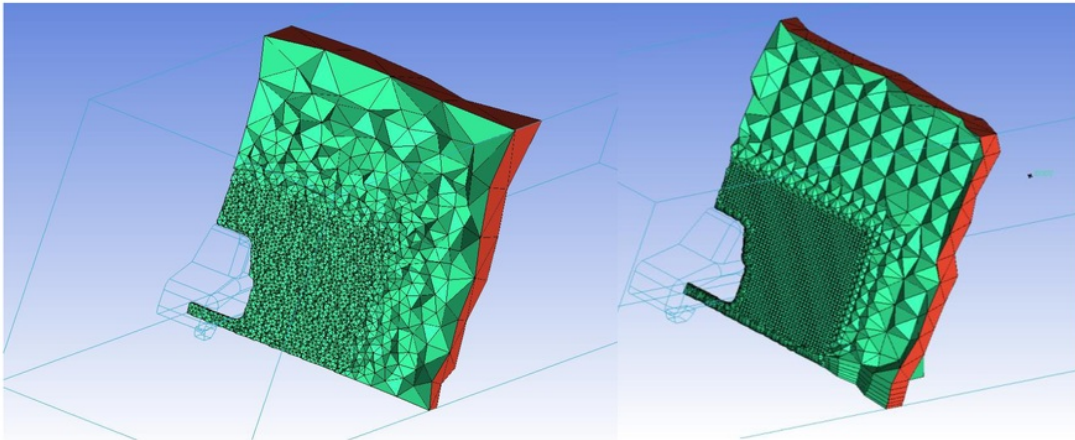


Figure 18: Unstructured Delaunay mesh (left) and structure Robust Octree mesh (right)

3.4.2 Mesh Controls

Mesh tools enable much more user control over how the mesh is structured. The two main types of mesh controls in ICEM-CFD are that of global controls and local controls. Global mesh controls enable the user to specify a maximum element size in the free-stream domain, whereas local mesh controls enable inputs such as refined element sizes for specific faces of the geometry, as well as number of prism layers.

Generally, the free-stream element size doesn't necessarily have to be exceptionally large due to the fact it doesn't have to capture large changes in flow field variables. Elements in the near vicinity of the vehicle however must be much more refined due to the triggered fluctuations in flow. This can be done by applying a density box in the area for mesh refinement, selected by the user. Using entity bounds around the car, the density box can be scaled in any cartesian direction depending on how much of the flow is desired to be refined.

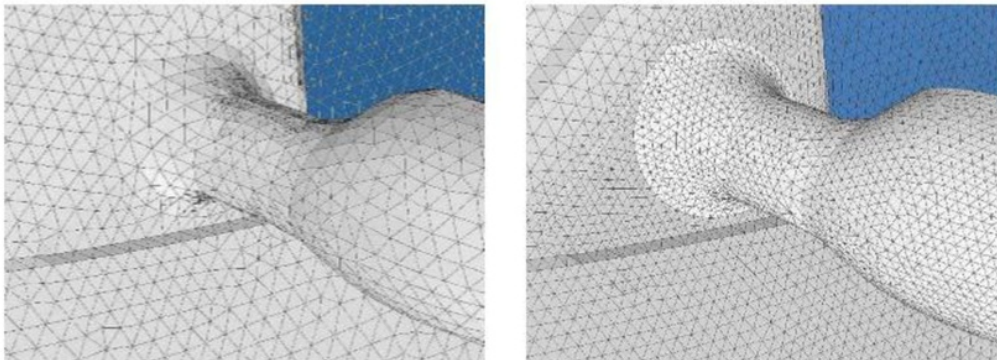


Figure 19: Mesh refinement using local mesh controls (Lanfrit 2005)

P150093

For the purposes of this study, a density box scaled 2.7x the length of the car in the X-direction, 1.5x the height in the Y-direction, and 3x the width in the Z-direction was used for each run to keep results from being affected by a change in size. Density box element size was also kept consistent at 0.065m.

Local mesh controls apply specified element sizes to faces of the domain or geometry. Since prism layers are extruded from the local mesh elements on selected faces, changing this element size also affects the number of prisms in one layer. Refined local meshes create much more accurate representations of the vehicle geometry, demonstrated in figure 19, thus much more accurate end results.

3.4.3 Mesh Quality

Mesh quality is crucial for convergence characteristics and a bad quality mesh can lead to difficulties solving and divergence of the model. Analysis of mesh generation comes down to values of skewness and orthogonal quality. Skewness is described as the magnitude of disfiguration of elements compared with the ideal element, where values close to 1 represent extremely skewed cells.

$$\text{Skewness} = \frac{\text{optimal element volume} - \text{actual element volume}}{\text{optimal element volume}}$$

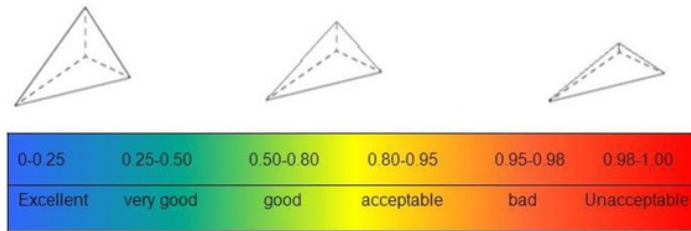


Figure 20: Scale of Skewness (ANSYS 2009)

As has been found in the prism layers study, skewness was of a greater scale where geometry took harsh directional changes, and where prism layers meet tetra mesh. With 8 prism layers on the floor and car there is limited space for the tetrahedral mesh to generate in between, such that as the tetra elements approach the prisms, they get flatter and more skewed.

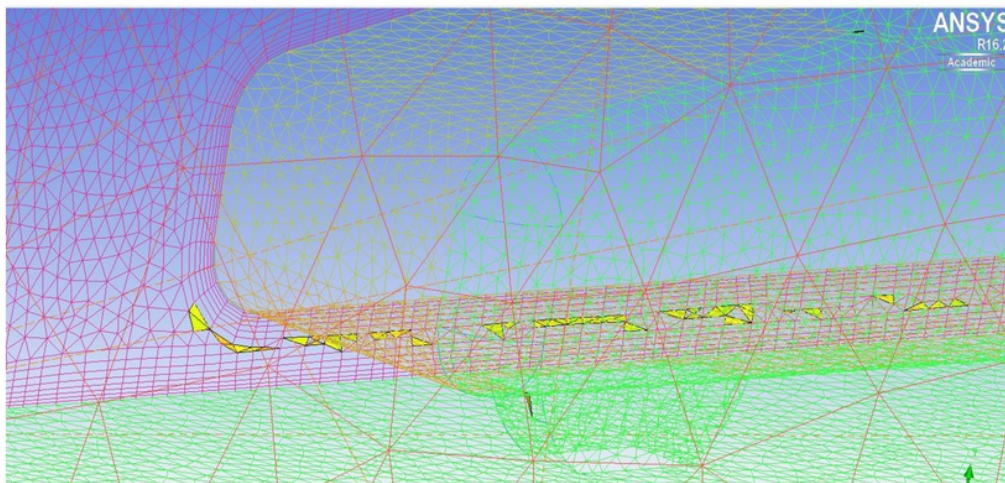


Figure 21: Skewness between Prism layers and tetra mesh

P150093

Another way of analysing the mesh is looking at the orthogonal quality, which describes the quality of the cell with respect to its faces and the cells surrounding it. $\vec{A}$ vectors represent vectors normal to the cell face, $\vec{f}$ vectors are those from the centroid of the cell to the centroid of each face, and $\vec{c}$ vectors are from the centroid of the cell to the centroid of the cells adjacent. Values of orthogonal quality follow the inverse scale of that of skewness, whereby values close to 1 represent good orthogonal quality.

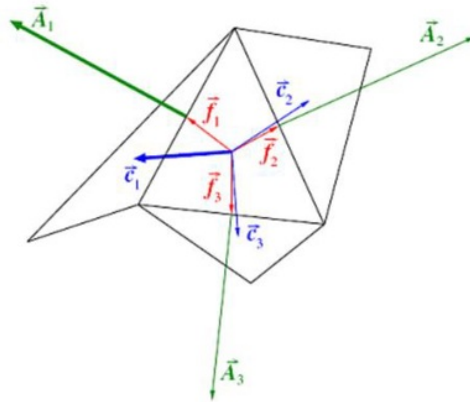


Figure 23: Orthogonal Quality Vector Representation (sharcnet n.d.)

Particularly around the wheels of the car, orthogonal quality tends to worsen due to the decreasing volume between the wheel and ground as the two meet. Solving this would require mesh refinement or the blocking of the wheels such that the mesh around them isn't to such a low quality. However doing this isn't representative of real world scenarios.

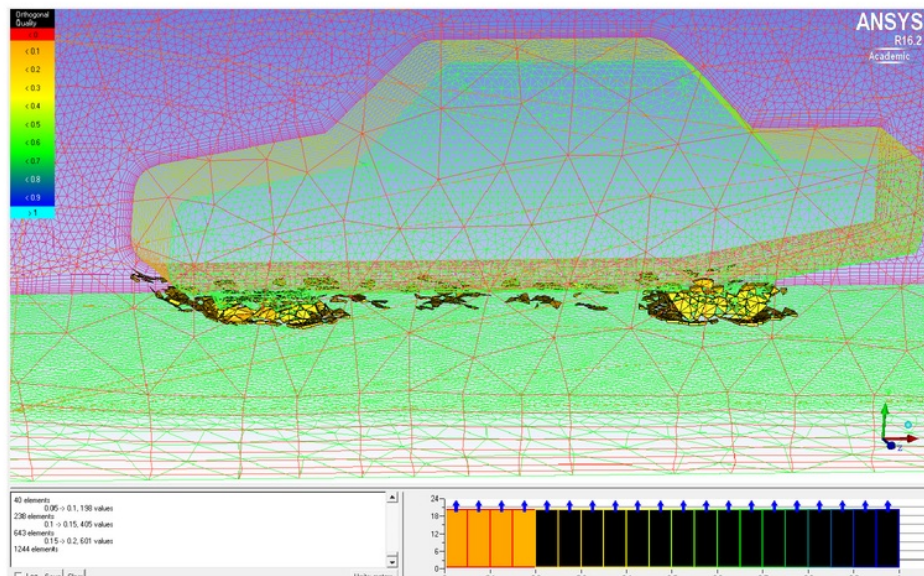


Figure 22: Bad Orthogonal Quality around wheels

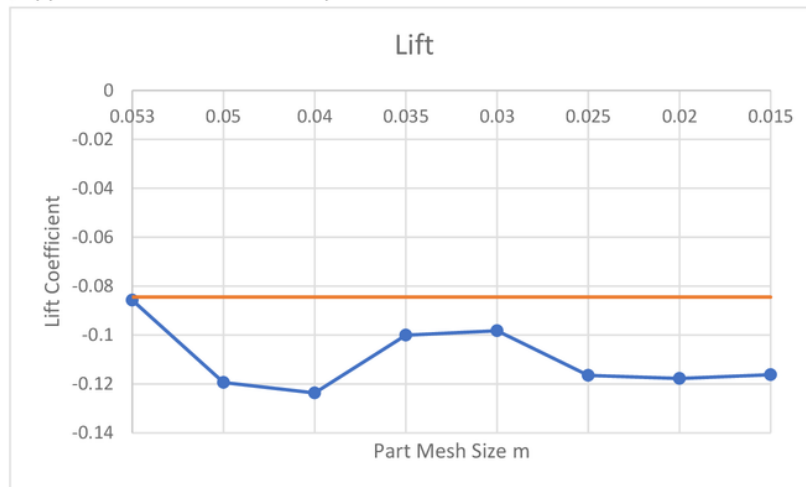
3.4.4 Mesh Study

Initially it was decided that for the mesh study, the global mesh would be kept at a constant 0.5m, and the local mesh should undergo refinement. The results of which are shown in figure 24.

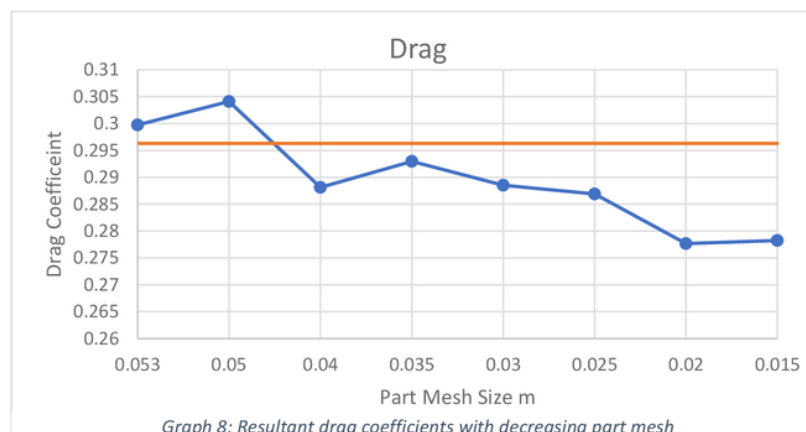
Global Size	Part Mesh	Cd	Cl	Cd % error	Cl % error	Total
0.5	0.053	0.29975	-0.08574	1.1644	1.4675	2.6318
0.5	0.05	0.30411	-0.11938	2.6358	41.2781	43.9139
0.5	0.04	0.28813	-0.12369	2.7573	46.3787	49.1360
0.5	0.035	0.29296	-0.10003	1.1272	18.3787	19.5059
0.5	0.03	0.28852	-0.098222	2.6257	16.2391	18.8648
0.5	0.025	0.28691	-0.11648	3.1691	37.8462	41.0152
0.5	0.02	0.27768	-0.11777	6.2842	39.3728	45.6570
0.5	0.015	0.278245	-0.116245	6.0935	37.5680	43.6615

Figure 24: Mesh Study 1 Results

As can be seen, the refinement in the mesh leads to an overall reduction in accuracy of the drag and lift values, opposite to what should be expected.



Graph 9: Resultant lift coefficient with decreasing part mesh



Graph 8: Resultant drag coefficients with decreasing part mesh

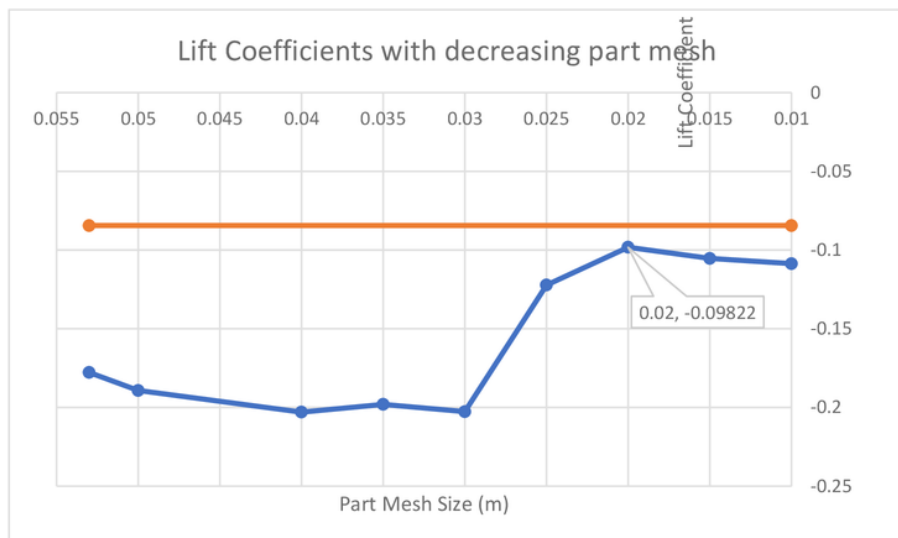
P150093

It is seen that the coefficient of drag gets gradually worse as the local mesh on the car faces is more refined, and the coefficient of lift follows no such correlation regarding part mesh size. In this initial mesh study and studies preceding, the “topside” planes were modelled as a wall boundary condition. Upon reflection the decision was made to make an alteration in the solver which rather modelled this plane as a symmetry plane. This condition then models the “topside” plane as if it were a symmetrical plane and flow was uninterrupted, rather than a wall plane where flow is complex.

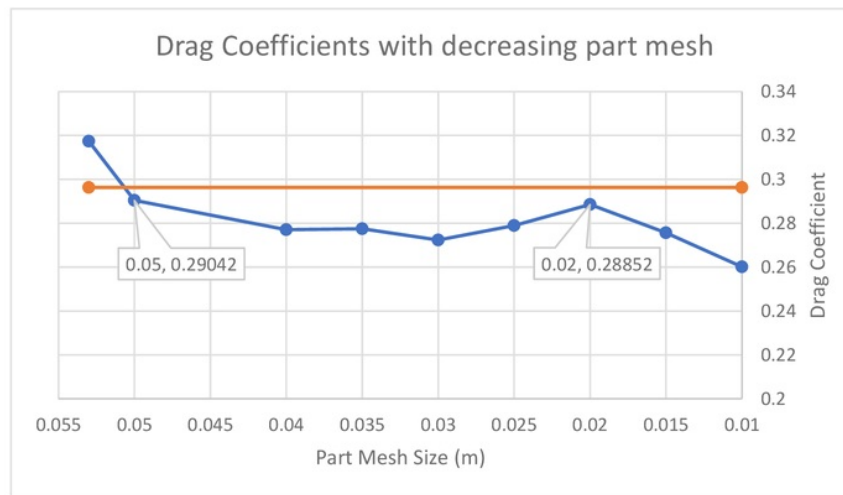
The impact of this is that the values produced from the previous studies won’t necessarily be the same when modelled with the topside as a symmetry plane. However since boundary conditions near to a wall have inherent complexities, it is theorised that this may produce a more valid mesh refinement study.

Global Size	Part Mesh	Cd	Cl	Cd % error	Cl % error	Total
0.5	0.053	0.3174	-0.17778	7.1212	110.3905	117.5117
0.5	0.05	0.29042	-0.18923	1.9845	123.9408	125.9253
0.5	0.04	0.27704	-0.20299	6.5002	140.2249	146.7250
0.5	0.035	0.27748	-0.19806	6.3517	134.3905	140.7422
0.5	0.03	0.27237	-0.20272	8.0763	139.9053	147.9816
0.5	0.025	0.27896	-0.12227	5.8522	44.6982	50.5504
0.5	0.02	0.28852	-0.09822	2.6257	16.2367	18.8624
0.5	0.015	0.27563	-0.10534	6.9760	24.6627	31.6388
0.5	0.01	0.26012	-0.10867	12.2106	28.6036	40.8141

Figure 25: Mesh Study 2 Results



Graph 10: Mesh Study 2 lift coefficient results



Graph 11: Mesh Study 2 drag coefficient results

The second study's results proved more valid than the first whereby values did tend to decrease as the part mesh gradually got more defined. Lift coefficients especially tended to follow this trend with values being lowest with a part mesh less than 0.02. However these meshes took especially long to generate in addition to taking a long time to converge due to the massive number of cells.

Coefficients of drag didn't change as drastically as those of lift. A part mesh of 0.05 had the closest value of drag, however a part mesh of 0.02 had the most accurate values of both coefficients and thus the best mesh.

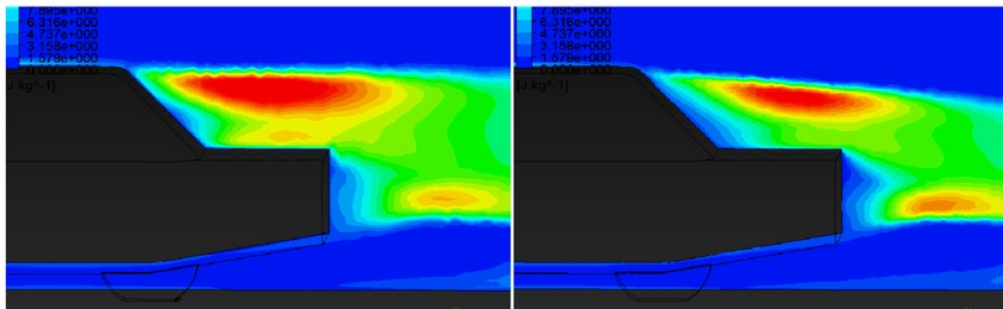
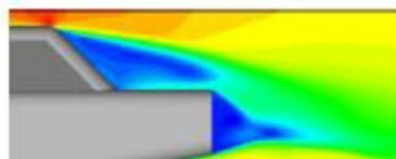


Figure 26: Part mesh 0.05 (left) vs 0.02 (right)

Figure 25 shows a case comparison between a course mesh on the left and fine mesh on the right. Flow separation direction of the fine mesh is much more representative of patterns seen in Palaskar's validation study, seen below.



(b). Notchback, present CFD velocity field

P150093

3.5 Turbulence Models

3.5.1 Model Types

Turbulence models are the most complex part of computational fluid dynamics where each one solves in a completely different way and therefore differing in regard to application. Some of the most popular models tend to use RANS methods to solve the Navier-Stokes equations, with added equations to solve certain variables.

K-Epsilon

The default 2-equation turbulence model tends to be the realizable adaptation of the k-epsilon model, and has proven to have good convergence and is well-documented in many validation cases. K represents the kinetic energy per unit mass of turbulent fluctuations, whereas the added variable to solve for is the kinetic dissipation rate (epsilon) (Wasserman 2016). This model predicts how the energy is going to be lost by accounting for fluid viscosity effects.

The realizable version of the k-epsilon model adds another variable for mean flow distortion on the kinetic dissipation rate and thus creates a modified equation for epsilon. The RNG form adds another extra term to the already modified epsilon equation, considering the effects of smaller scale flow on turbulent diffusion (CFD-Online n.d.). Resultantly the RNG form produces slightly lower turbulence levels and tends to be less accurate than the previous forms and thus is not commonly used in many large-scale applications (Wasserman 2016).

K-Omega

K-omega is another two-equation model whereby the variable omega represents the specific dissipation rate, which defines the rate at which turbulent kinetic energy is converted into internal thermal energy. Aside from the standard k-omega model, two other models exist: baseline (BSL) and shear stress transport (SST). The baseline (BSL) model is a refinement on the standard and is a combination of the standard k-omega and k-epsilon behaviours, whereby blending functions are used to define where the flow should be modelled using either k-omega or k-epsilon equations. K-omega is used to simulate the flow at the inner boundary region due to its capability at replicating near-wall flow structures, and when transitioning into the free-stream region the k-epsilon equations are introduced (sharcnet n.d.). The SST K-omega model simply improves the BSL model with the inclusion of turbulent shear stress transportation when modelling turbulent viscosity effects.

P150093

3.6 Wall Treatments

Some turbulence models have an added option of near-wall modelling conditions, depending on the non-dimensional distance Y^+ and the non-dimensional velocity U^+ .

U^+ is the ratio of local velocity u to frictional velocity u_τ at the near wall, such that:

$$u^+ = \frac{u}{u_\tau} \quad u_\tau = \sqrt{\frac{\tau_{wall}}{\rho}}$$

Y^+ is then the dimensionless distance which can be calculated:

$$y^+ = \frac{u_\tau y}{\mu}$$

Where u_τ is the friction velocity, μ is the kinematic viscosity, and y is the distance from the wall.

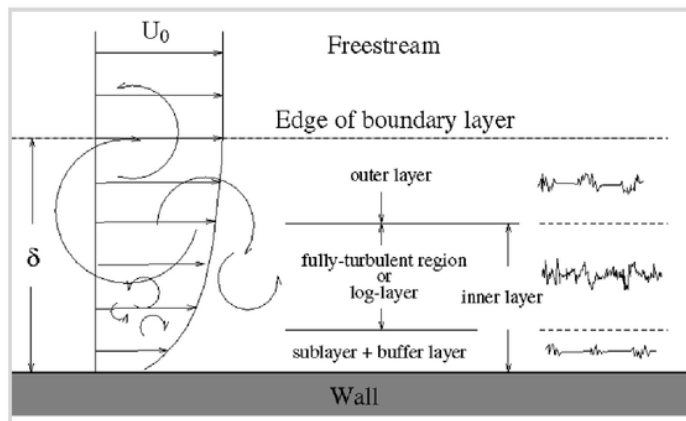


Figure 27: Boundary Layer (ANSYS 2012)

The boundary layer can be split into significant layers: the sublayer which is closest to the wall, the log-layer whereby the flow is fully-turbulent, and the outer layer closest to the freestream. The relationship between Y^+ and U^+ for the sublayer where $y^+ < 11.225$ generally follows a linear pattern whereby:

$$y^+ = u^+$$

However the middle turbulent layer follows the logarithmic relationship and is valid for y^+ values between 30 and 300 (ANSYS 2006):

$$u^+ = \frac{1}{k} \ln(y^+) + B$$

Where k and B are constants 0.41 and 5.1, respectively.

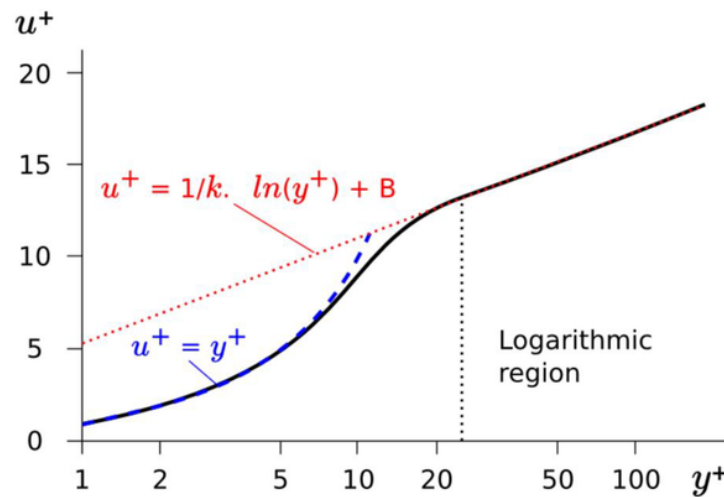


Figure 28: Graph of Y^+ and U^+ (ANSYS 2012)

These equations enable the predictability of flow at the boundary layer, such that wall treatments can be used to model this rather than refining the mesh at the boundary. Solving using wall treatments essentially means the mesh doesn't need to be as dense at the boundary layer and thus solve time is reduced.

3.6.1 Standard Wall Functions

Standard wall functions are the default option in Fluent and is used commonly in industrial application. For the k-epsilon model, this function assumes that values of turbulent kinetic energy and kinetic dissipation rate in wall-adjacent cells are equal in accordance with the local equilibrium hypothesis. Thus K is solved for the whole domain including the boundary layer, whereas epsilon is only solved for the freestream (ANSYS 2006). When flows tend to conform to strong pressure gradients and localised non-equilibrium the reliability of this model producing accurate results is compromised, thus the need for improved wall treatments.

3.6.2 Non-Equilibrium Wall Treatments

The non-equilibrium wall treatment adopts a two-layer concept consisting of wall-adjacent and wall-neighbouring cells, and adjusts for pressure gradient effects and for the computation of a budget turbulent kinetic energy in wall-neighbouring cells. Wall-neighbouring cells are assumed to be split into a viscous sublayer and fully turbulent layer and the proportionality of such is used in computing the budget turbulent kinetic energy. Where standard wall functions works under the assumption that turbulent kinetic energy and its dissipation rate are equal at wall-adjacent cells, the non-equilibrium treatment disputes this and thus accounts for distortion in flow velocity due to pressure gradients. This treatment is the ANSYS-recommended wall function for the K-Epsilon model due to its capability in solving for rapid changes in pressure and turbulence (ANSYS 2006) such as flow separation regions.

P150093

3.6.3 Enhanced Wall Treatments

As opposed to the standard wall treatment, enhanced treatments solves the turbulent dissipation rate and turbulent viscosity all the way down to the viscous sublayer. The entire domain is split into two regions: viscosity-affected and fully turbulent, where the intersection between the two is calculated from the turbulent Reynolds number:

$$Re_y = \frac{\rho y \sqrt{k}}{\mu}$$

Blending functions are used to smooth the intersection between the two laminar and turbulent regions where wall laws are enhanced to account for the blending of the regions.

3.7 Turbulence Study Results

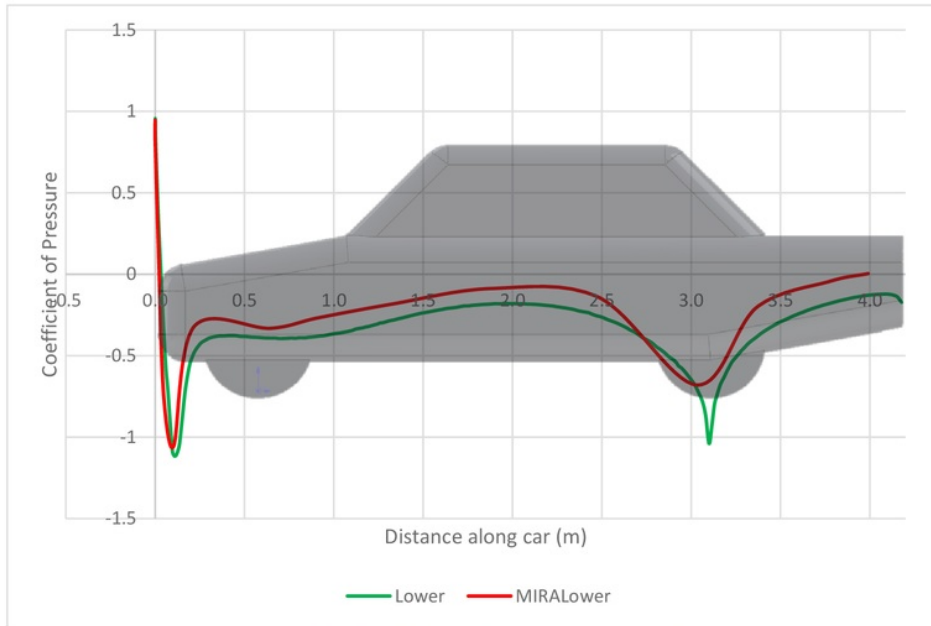
	Turbulence Model	Wall Functions	CD	CL	Cd Error %	Cl Error %	Total Error %
1 eqn	Laminar		0.62388	-0.16599	110.557	96.438	206.995
	Spalart-Allmaras		0.30736	-0.13945	3.733	65.030	68.762
	K-Epsilon						
2 eqn	Standard	Standard	0.29370	-0.08369	0.877	0.963	1.841
		Scalable	0.29377	-0.08563	0.854	1.337	2.191
		Non-Eq	diverged				
		Enhanced					
	RNG	Standard	0.28412	0.012392	4.111	114.665	118.776
		Scalable	0.28412	0.013387	4.111	115.843	119.953
		Non-Eq	0.28611	0.010977	3.439	112.991	116.430
		Enhanced	0.28767	0.042804	2.913	150.656	153.568
	Realizable	Standard	0.28149	-0.12213	4.998	44.533	49.531
		Scalable	0.28149	-0.12213	4.998	44.533	49.531
		Non-Eq	0.28852	-0.09822	2.626	16.237	18.862
		Enhanced	0.28793	-0.09634	2.825	14.012	16.837
	K-Omega						
	Standard		0.28525	-0.03087	3.729	63.473	67.203
	BSL		0.29927	-0.09979	1.002	18.095	19.097
	SST		0.30689	-0.17382	3.574	105.704	109.278
3 eqn	Transition k-kl Omega		0.35801	0.16296	20.827	292.852	313.679
4 eqn	Transition SST		0.30464	0.03556	2.815	142.083	144.898

Figure 29: Turbulence study results

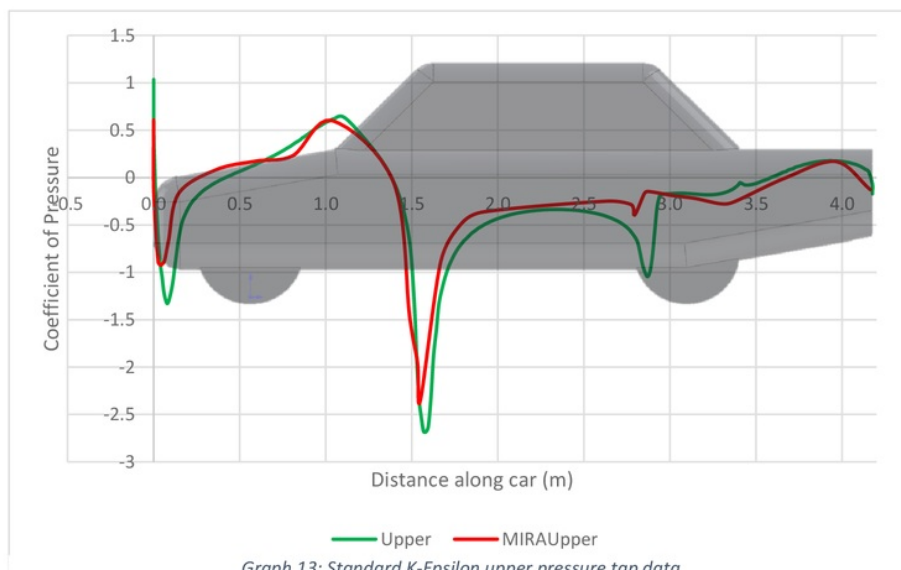
As the turbulence model increases the number of equations it solves for, the consequent computational time and power increases. In the first study with a Laminar turbulence model, the residuals did not converge to an adequate value and rather continued to oscillate throughout the 1000 iterations it was run for. Unsurprisingly this is due to the fact the flow is turbulent therefore a laminar flow turbulent model is not advised.

P150093

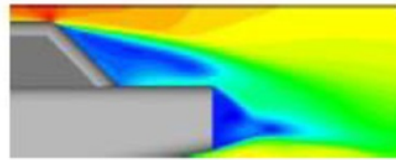
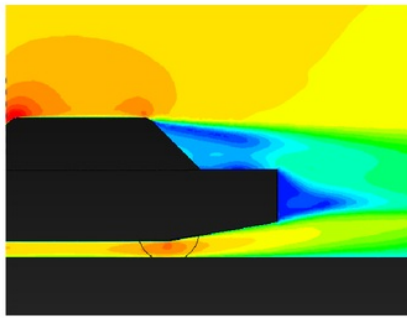
Out of the tested K-Epsilon models, the standard option resulted in the most accurate results within less than 1% of both drag and lift with standard wall functions. Due to the simplified assumptions that the standard model adheres to, it's interesting to find that it produced the most accurate results. However outputting the pressure tap data from this turbulence model does not quite match up accurately with that measured by MIRA as seen on graphs 12 and 13.



Graph 12: Standard K-Epsilon lower pressure tap data



Graph 13: Standard K-Epsilon upper pressure tap data



(b). Notchback, present CFD velocity field

Figure 30: Standard K-Epsilon rear velocity field (left) validated case velocity field (right)

Post-processing of the standard K-Epsilon model shows that the velocity profile at the rear of the car is quite similar to that of the validation study by Palaskar et al (Palaskar , Kumar and Vaidya 2016).

The wake region directly behind the car is very similar in shape to that produced in the 2016 Notchback validation study. Velocity at the rear slant isn't quite as turbulent as expected, and the velocity vectors show that circulation here isn't as pronounced at the boundary of flow separation where freestream flow meets circulating vortices. This could be due to the difference in flow separation from the roof of the car. The direction of flow separation slants downwards in the validated case whereas this study shows it be more horizontal. The validated case was done with a realizable K-epsilon model with non-equilibrium wall functions, which as discussed earlier is able to better model flow separation.

Directly behind the tail however shows defined circulation of flow as seen in figure 31.

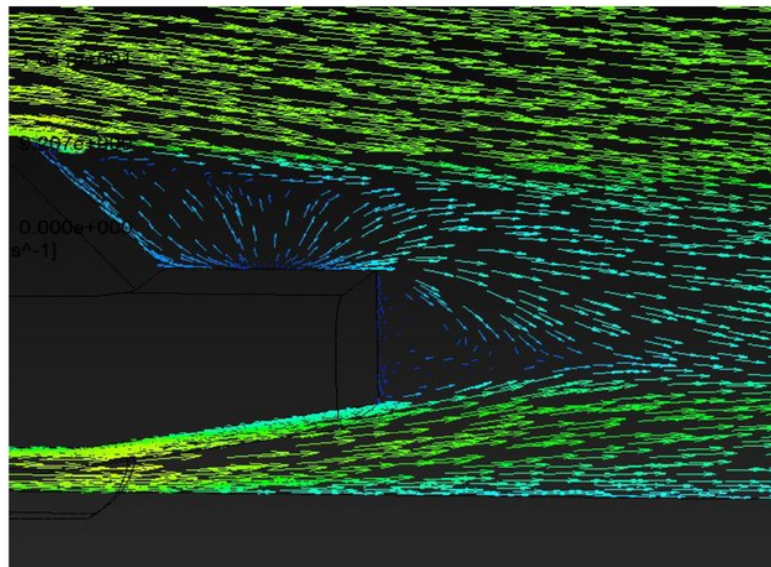


Figure 31: Standard K-Epsilon Velocity Vectors

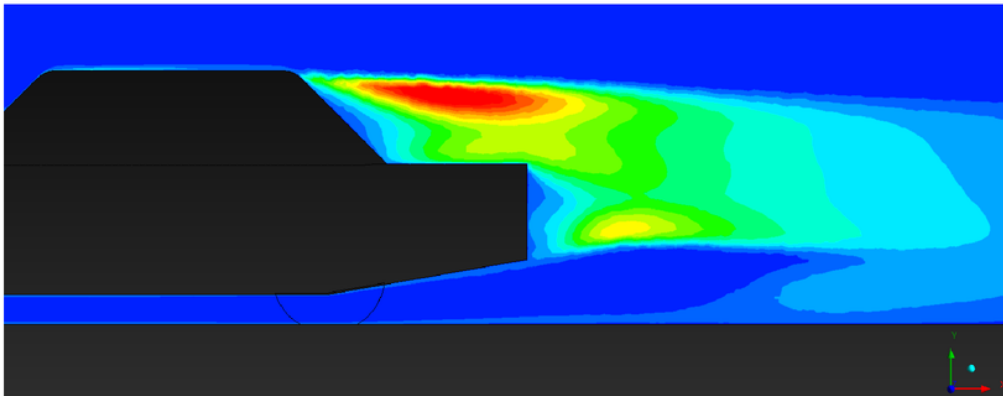


Figure 33: Standard K-Epsilon turbulence properties

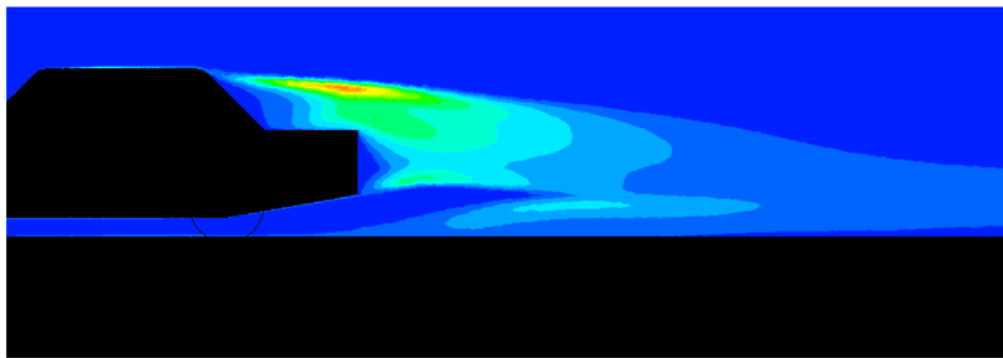


Figure 32: RNG K-Epsilon turbulence properties

The turbulence study results table shows that the RNG K-Epsilon model produced positive lift no matter what the functional wall treatment. Section 3.5.1 discussed the disadvantages of the RNG model in that it tends to underestimate levels of turbulence and therefore produce inaccurate results. Figures 32 and 33 show the relative difference in turbulence between the standard K-epsilon model and RNG model, respectively.

Figure 34 shows a case comparison between the K-Omega baseline study and the standard K-Epsilon. The BSL K-Omega study yielded the most accurate results of drag and lift out of the three K-Omega options, however still a total of 19% inaccuracy. The flow separation profile is slightly more similar to that of Palaskar's study, however the velocity profile extends too much into the wake region of the car.

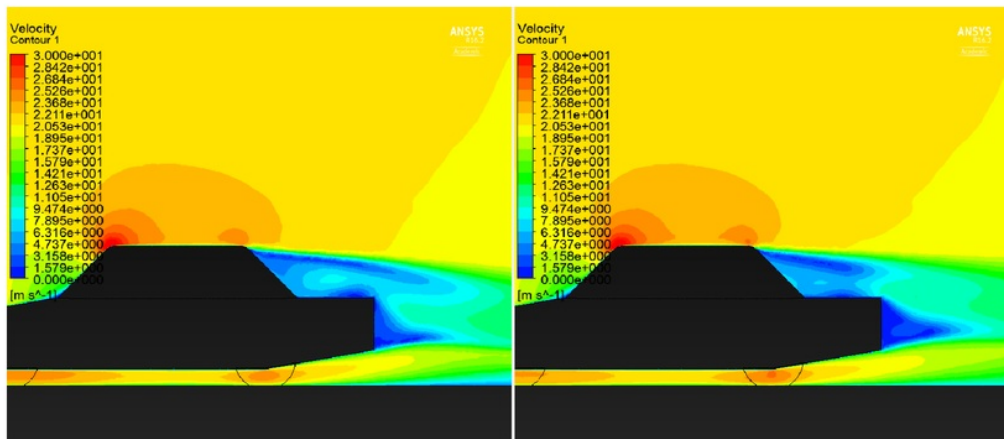


Figure 34: Case comparison of K-Omega BSL (left) and K-Epsilon Standard (right) velocity profiles

3.8 Final Baseline Settings

The best results from each study are compiled to produce a final baseline set-up for future aerodynamic simulations, such that the solutions of which conform with validated methods.

- Domain size: approximately 2 car lengths in front, 4 behind, 1 up, 1 to the side
- Prism layers: 6
- Part mesh density: 0.02m
- Turbulence model: Standard K-Epsilon with standard wall functions

4. Aerodynamic Modification

Since baseline settings are known to produce accurate results, a modification can be made to the MIRA geometry and tested with the same settings to determine the effect of the alteration. Rear wings are commonly used to increase downforce whereby the wing profile directs air flow in a way that creates a pressure differential between the upper and lower surfaces, such that the higher pressure above the wing pushes down to produce downforce.

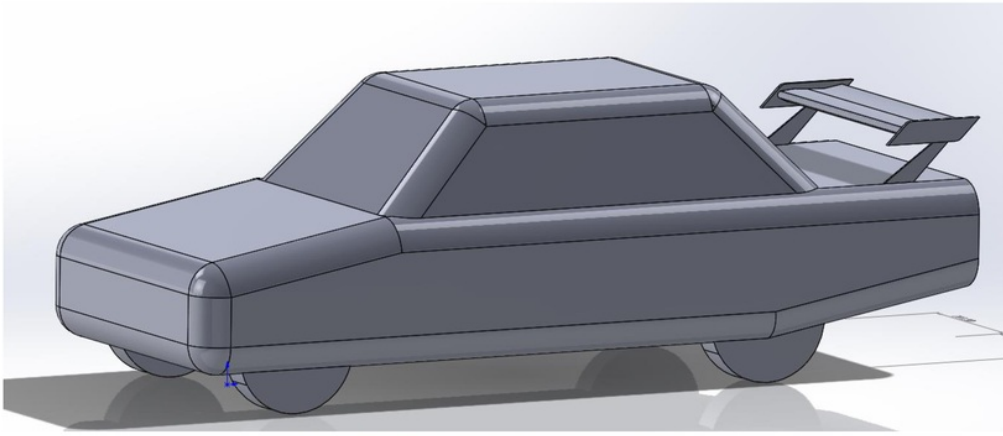


Figure 35: MIRA Notchback rear wing modification

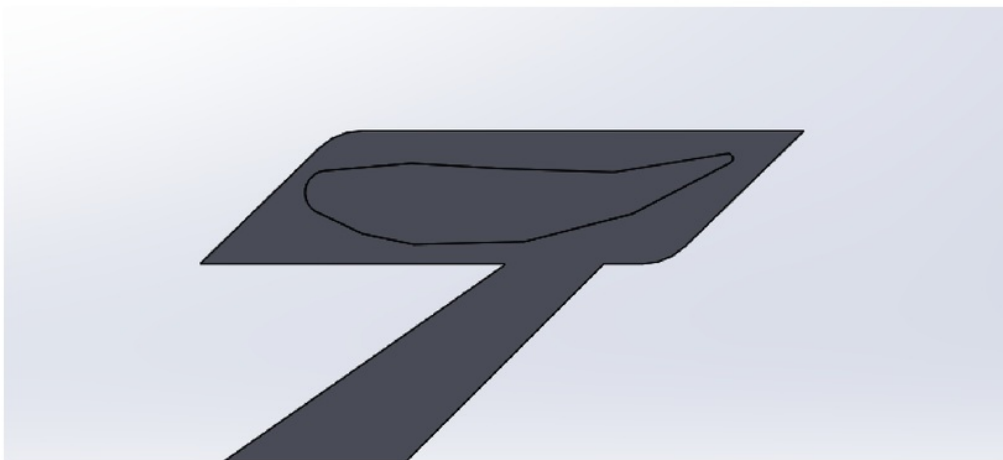


Figure 36: Rear Wing profile

P150093

Due to the increased complexity of the geometry, it wasn't possible to input the geometry into Fluent without first making a slight modification to the mesh. Orthogonal quality was completely unacceptable in the tight angled area underneath the end plates using the same settings determined previously. Thus, the wing had a separate part mesh of 0.01, and an additional density box was added surrounding the wing to properly capture the flow characteristics.

The resultant coefficients of drag and lift were 0.3087 and -0.2186 respectively, a 160% increase in the coefficient of lift and thus producing more downforce.

Figure 37 shows the pressure contours of the MIRA Notchback car with added rear wing. The increased pressure on the upper side of the wing is what aids in the creation of downforce and thus theoretically improving grip if the modification were to be made to a real vehicle.

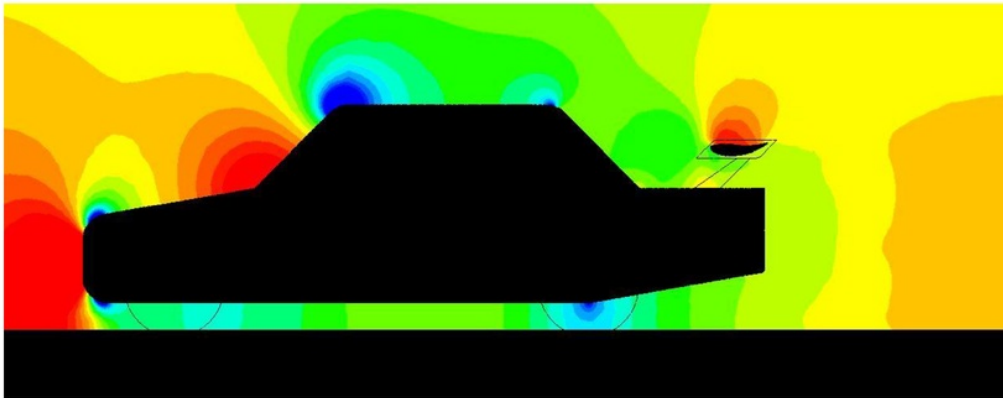


Figure 37: Modified MIRA Notchback pressure contours

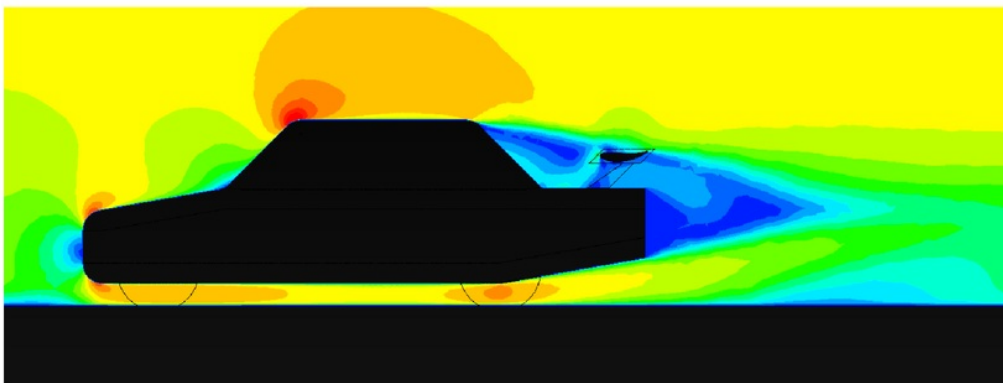


Figure 38: Modified MIRA Notchback Velocity contours

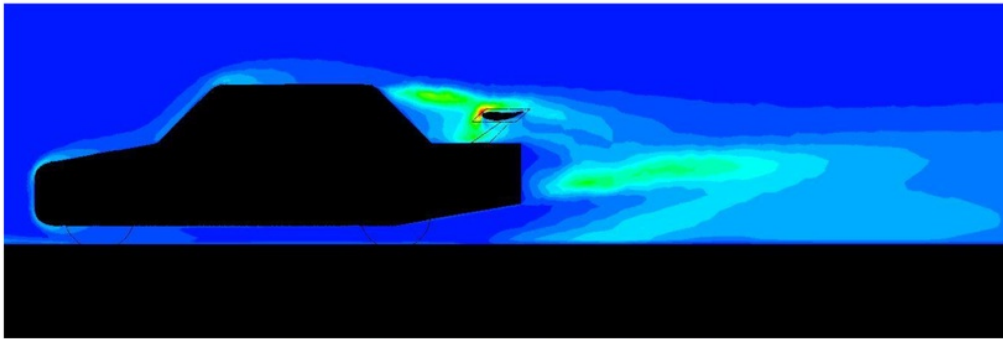


Figure 39: Modified MIRA Notchback Turbulence profile

Figure 39 shows how the wing breaks up turbulence at the rear slant of the vehicle and how flow is directed rearwards of the vehicle. As a result flow is more turbulent in the initial wake region of the car due to the flow separation caused by the wing.

Aerodynamic devices that aid in downforce, such as rear wings, can often lead to increased drag. It's therefore beneficial for an aerodynamicist to work on the underbody of the car to produce downforce because the air flow is already there, as opposed to adding a rear wing which will aid in downforce at the cost of potentially increasing drag.

5. Conclusion

The methods followed in this study to validate the simulation align with methods followed in previous studies. Upon reflection the results don't necessarily correlate with what is known about CFD models. For example, the result of this study showed that the turbulence model that output the best results was the Standard K-Epsilon with Standard wall functions, however the developments made to this model by the introduction of the Realizable form and wall treatments aim to improve the accuracy of the standard version, therefore the fact that the realizable model reduced the accuracy of the drag and lift values may imply that the model still needs development.

The domain study concluded that both excessively small and excessively large domains were of detriment to the resultant drag and lift values, whereas a domain volume of 515m^3 produced the most accurate results. Since the domain study primarily focused on the impact of domain length, further investigation may look into the effects of taller and wider domains.

Prism layers were only added once the domain study had been completed. This method ensured that no other variable was affecting the results of the domain study. The separate inflation layer study measured how aerodynamic values changed with increasing prism layers, which help give detail to boundary layer flow. Accuracy increased up to 6 prism layers, whereby after that cells beneath the car became too skewed for the software to accurately model.

Results of the mesh study showed that a part mesh of 0.02 offered a suitable combination of accuracy and computational time. The part mesh was fine enough to model the geometry of the MIRA car without compromising time constraints too much. With finer meshes than this the accuracy did not get much better, yet it took a lot longer to solve. However with more complex geometry such as that of the addition of a rear wing, the 0.02 mesh needed further refinement before inputting into Fluent.

6. Further Work

Multiple other variables are at play in CFD and other studies could have been conducted were time not one of the main limiting factors. A study of density box dimension and mesh size would have been interesting. This was the main constant throughout the study, therefore noting the effects of altering the density box properties may have increased accuracy.

Multiple density boxes around areas of significant reduced quality would have also been beneficial to test, for example around the wheels. Other studies blocked off the wheels to better the orthogonal quality, however this isn't representative of real geometry.

P150093

7. References

- Anderson, John D. 1984. "Fundamentals of Aerodynamics." *McGraw-Hill Series in Aeronautical and Aerospace Engineering* (McGraw-Hill Inc).
- ANSYS. 2009. *ANSYS Training Manual*. ANSYS, Inc.
- . 2006. "Non-Equilibrium Wall Functions." *ANSYS Fluent*. Accessed January 2019. <https://www.sharcnet.ca/Software/Fluent6/html/ug/node513.htm>.
- . 2006. "Standard Wall Functions." *ANSYS Fluent*. Accessed January 2019. <https://www.sharcnet.ca/Software/Fluent6/html/ug/node512.htm>.
- . 2012. *Turbulence Near the Wall*. ANSYS, Inc.
- Bernard, H. 2001. *Road Vehicle Aerodynamic Design*. London: MechAero Publishing.
- Cakir, Mustafa. 2012. *CFD study on aerodynamic effects of a rear wing/spoiler on a passenger vehicle*. Santa Clara, California: Santa Clara University.
- CFD-Online. n.d. *RNG k-epsilon model*. Accessed January 2019. https://www.cfd-online.com/Wiki/RNG_k-epsilon_model.
- Dobrev, Ivan, and Fawaz Massouh. 2014. *Investigation of relationship between drag and lift coefficients for a generic car model*. Sozopol, Bulgaria: Arts et Métiers ParisTech.
- Elger, Donald F., Barbara A. LeBret, Clayton T. Crowe, and John A. Roberson. 2013. *Engineering Fluid Mechanics*. Singapore: John Wiley & Sons Singapore Pte. Ltd.
- Engineers Edge. 2000. "Viscosity of air, Dynamic and Kinematic." *Engineers Edge*. https://www.engineersedge.com/physics/viscosity_of_air_dynamic_and_kinematic_14483.htm.
- Formula1. n.d. "Overtaking and the DRS." *Inside F1*. Accessed January 2019. https://www.formula1.com/en/championship/inside-f1/understanding-f1-racing/Overtaking_and_the_DRS.html.
- Frei, Walter. 2017. "Introduction to Turbulence Modeling." *COMSOL*. 6 July. <https://uk.comsol.com/blogs/which-turbulence-model-should-choose-cfd-application/>.
- Gaylard, A P, and A J Baxendale. 1998. *The Use of CFD to Predict the Aerodynamic Characteristics of Simple Automotive Shapes*. Danvers: Society of Automotive Engineers.
- HORIBA MIRA. n.d. *Full Scale Aerodynamic Wind Tunnel*. Accessed November 2018. [https://www.horiba-mira.com/our-services/full-scale-wind-tunnel-\(fswt\)](https://www.horiba-mira.com/our-services/full-scale-wind-tunnel-(fswt)).
- Katz, Joseph. 2006. *Race Car Aerodynamics*. Cambridge, Massachusetts: Bentley Publishers.
- Lanfrit, Marco. 2005. *Best practice guidelines for handling Automotive External Aerodynamics with FLUENT*. Technical Paper, Darmstadt, Germany : Fluent Deutschland GmbH.
- McBeath, Simon. 1999. *Competition Car Downforce*. Yeovil: Haynes Publishing.
- Palaskar , Pruthviraj Mohanrao, Vivek Kumar, and Rohit Vaidya. 2016. *Methodology Development to Accurately Predict Aerodynamic Drag and Lift for Passenger Vehicles Using CFD*. SAE Technical Paper 2016-01-1600, Society of Automotive Engineers. doi:10.4271/2016-01-1600.

P150093

sharcnet. n.d. *Orthogonal Quality*. SAS IP, Inc.

—. n.d. *Shear-Stress Transport (SST) k - ω Model*. SAS IP, Inc. Accessed January 2019.
https://www.sharcnet.ca/Software/Ansys/16.2.3/en-us/help/flu_th/flu_th_sec_turb_kw_sst.html.

Tucker, Paul. 2016. *Advanced Computational Fluid and Aerodynamics*. New York: Cambridge University Press.

User:Chb. 2005. "Ahmed Body." *CFD-Online*. 21 November. https://www.cfd-online.com/Wiki/Ahmed_body.

Wasserman, Shawn. 2016. *Choosing the Right Turbulence Model for Your CFD Simulation*. 22 November.
<https://www.engineering.com/DesignSoftware/DesignSoftwareArticles/ArticleID/13743/Choosing-the-Right-Turbulence-Model-for-Your-CFD-Simulation.aspx>.