

Mira CFD

by Charles Thomas Randall

FILE	7607_CHARLES_THOMAS_RANDALL_MIRA_CFD_128450_600357818.DO CX (44)		
TIME SUBMITTED	28-JAN-2019 08:49AM (UTC+0000)	WORD COUNT	5289
SUBMISSION ID	99899180	CHARACTER COUNT	34045

Aerodynamic Vehicle Testing and Analysis

Lecturer: Mr Tim Tudor

Module Title: Advanced Computational Methods

Module Code: M6X00408

Programme: BEng Motorcycle Engineering

Name: Charles T Randall

P Number: 147153

Contents

Introduction	1
Aims	1
Research.....	2
Domain.....	2
Domains & Boundary Conditions	2
Domain Size.....	2
Flow Patterns	3
Mesh	4
Mesh Type.....	5
Mesh Sizing	6
Prism Layers	6
Orthogonal Quality and Skewness.....	7
Turbulence Models	8
Reynolds Number.....	9
Methodology/Results???	10
Domain Size Study.....	10
Car Placement Study.....	12
Car Mesh Study	14
Density Box Size Study	15
Density Box Position Study	17
Density Box Mesh Size Study	18
Prism Layer Study.....	20
Turbulence Model Study.....	21
Modified Mira Car	23
Efficiency between original and mod	Error! Bookmark not defined.
Conclusion	26
Bibliography	27
Appendix	Error! Bookmark not defined.

List of Figures

Figure 1 - Examples of Turbulent and Laminar Flow, (Day, 2018)	3
Figure 2 - Example of Turbulence, (Comsol, 2019)	4
Figure 3 - Examples of Octree Mesh (left) and Quick Delaunay Mesh (right), (Randall, 2018)	5
Figure 4 - Example of Prism Layers, (Chhunchha, 2018).....	6
Figure 5 - How Orthogonal Quality is Calculated, (Sharqnet, 2015)	7
Figure 6 - Skewness, (Ansys, 2012)	8
Figure 7 - Domain Size Study, (Randall, 2018)	11
Figure 8 - Geometry Placement Comparison, 6m from VI (left), 12m from VI (right), (Randall, 2018)	12
Figure 9 - Geometry Placement Comparison, 6m from VI (left), 1.5m from VI (right), (Randall, 2018)	13
Figure 10 - Turbulence Comparison between Geometry being placed 6m from VI (left) and 1.5m from VI (right), (Randall, 2018)	13
Figure 11 - Reference Geometry Placement in Domain Results, (Randall, 2018)	13
Figure 12 - Mesh Density Comparison, 0.7m (left) 0.4m (right), (Randall, 2018).....	15
Figure 13 - Car Mesh Study, (Randall, 2018).....	15
Figure 14 - Density Box Comparison, 4x Geometry Volume (left), 1.5x Geometry Volume (right), (Randall, 2018)	16
Figure 15 - Density Box Size Study Graph of Results, (Randall, 2018)	17
Figure 16 - Comparison of Density Box being placed 4m from the VI (left) and 2m from the VI (right), (Randall, 2018)	17
Figure 17 - Comparison of a fine mesh density (left) and a large mesh density (right), (Randall, 2018)	19
Figure 18 - Graph of Prism Layer Study Results, (Randall, 2018).....	20
Figure 19 - Pressure Taps for Upper and Lower Surfaces of Mira Reference Car, (Randall, 2018)	22

List of Tables

Table 1 - Example of Orthogonal Quality	7
Table 2 - Orthogonal quality spectrum	7
Table 3 - Skew Spectrum.....	8
Table 4 - Base settings used for simulations	10
Table 5 - Domain Size Study	10
Table 6 - Mira Car Placement Study.....	12
Table 7 - Mira Car Mesh Density Results Table	14
Table 8 - Density Box Size Study Results	16
Table 9 - Density Box Position Results	18
Table 10 - Density Box Mesh Size Study Results	18
Table 11 - Prism Layer Study Results	20
Table 12 - Turbulence Model Study Results	21

Abbreviations

CFD – Computational Fluid Dynamics

VI – Velocity Inlet

Introduction

The use of wind tunnels for aerodynamic research is often expensive and due to the limited number of test sties, can mean considerable travelling is required. The ability to conduct research on computers using computational fluid dynamics (CFD) software in a closer location will save on expenses, travel time, and can similarly teach students about the basic principles of aerodynamics with visual representations, from fluid dynamics results obtained. Using CFD is not as straightforward as it may first appear. Many things can distort results making them unusable as they can be non-representative of the real situation because the software relies on assumptions. This report will be used to conduct a number of studies, to conduct research on what domain size to use with a certain sized model and how much smaller the domain can be made without distorting results too much to save on computational time. Once the domain is complete, a mesh must be created in the domain. A mesh allows the simulation software to calculate and predict fluid flow properties at every point in the domain. Once the mesh is complete it is imported into Fluent, which is where conditions are set, boundary conditions are applied, and turbulence models are established. Once parameters are set such as the speed of the air/fluid, the simulation can be ran. The computational time will depend on how complex the geometry is, how refined the mesh is and what parameters have been set for convergence, if any. Within the automotive industry drag and lift coefficients are vitally important as they can determine the vehicles top speed and maximum cornering speed. Within MotoGP this has recently become important, as aerodynamics is considered a "dark art". Not as much time has been spent on development when compared to formula 1. The Mira geometry used in this project is a basic geometry which is ideal for beginners using CFD software to analyse the effects of fluid flow under set conditions.

Aims

The aims of this assignment are;

- To attempt to replicate the conditions used for the Mira reference using Ansys CFD software
- To display confidence in analysing CFD results as well as showing competence using the Ansys CFD package
- To correctly explain how geometry changes affect the fluid flow properties of the reference car

Research

Domain

Domains & Boundary Conditions

A domain is either a 2D or 3D space in which the analysis software uses to solve the numerical equations of fluid flow, either around a body or through an internal body. The domain is defined by several faces which connect, for example a cube, in which boundary conditions are set. (Yunus & Cimbala, 2010). Boundary conditions are necessary limitations set to solve a boundary value problem. Boundary value problems are differential equations to be solved in a domain with known boundaries conditions set. (Cheng & D, 2005)

A suitable sized domain is required to run CFD analysis on a model. Knowing what dimensions to make the domain is important, a compromise between accuracy and simulation time should be considered. As the computation time is exponentially proportional to the domain size, having a domain too large will cause simulation time to increase dramatically without necessarily improving the accuracy of results. On the contrary, if the domain is too small it can produce inaccurate results due to the boundaries effecting the fluid flow results around the body of interest.

Domain Size

The domain parameters should consider the size of model being analysed. As the Mira reference car is being used in this project, simulations will be conducted to determine what size domain is fit for the reference geometry. According to Lanfrit, the domain size should be at least 3 car lengths in front, 5 car lengths behind and the displacement of the car should equate to 1-1.5% of the total cross-sectional area. (Lanfrit, 2009). This approach is not relevant if the domain boundaries serve as walls of a real wind tunnel, if this is the case then wall effects should be considered (Lanfrit, 2009).

Contrary to the information above, a paper on building flow domains with Ansys claims that the domain of a given object using external flow analysis should be at least 2 lengths in front of the object, 10 lengths behind, 10x width and 5x the height (Building flow domains with ANSYS design modeler, 2013). As the two sources above are credible, domains will be created with both geometry parameters. If both domain sizes produce similar results (within 5% of each other), the smaller domain will be used, as larger domains require more time to solve.

Flow Patterns

Flow patterns represent the way a fluid flows around an object. Using external flow analysis, if nothing is obstructing the domain inlet, laminar flow is expected to be entering the domain, this is where the flow of fluid is regular, or travels in a smooth path. This is sometimes called streamline flow where the velocity, pressure and additional flow properties remain constant at each point in the fluid. If looking at a horizontal surface in a state of laminar flow, the fluid would consist of thin layers in parallel, the fluid in contact with the surface appears stationary with the other layers sliding over each other in constant state, commonly referred to as boundary layers. (The Pillars Curriculum for chemical engineering, n.d.)

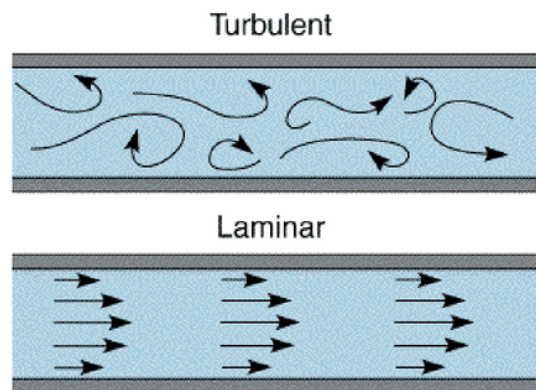


Figure 1 - Examples of Turbulent and Laminar Flow, (Day, 2018)

Just before laminar flow becomes turbulent it enters a state of transition, this is called transitional flow. At this point the flow is neither purely laminar nor turbulent, the fluid is behaving in different manners at different points, therefore their terms of frictional energy loss use different equations to anticipate their behaviour. (Jeffords, 2004). Turbulent flow happens when each element's velocity is fluctuating, this generally occurs at higher flowrate velocities. One way to view turbulence is by looking at fluid flowing through an object with obstructions in the shape of walls, see figure 2, it can be seen that the fluid is trying to take the easiest path (straight) but is getting obstructed by the walls, this along with the roughness of the walls and the free stream of the fluid is causing turbulence to occur. In the model below the fluid speed is 1m/s and the inlet diameter is 0.4m, so the Reynolds number is relatively low at 400,000. The Reynolds number is a good indication of how turbulent a body will be, generally larger bodies have higher Reynolds numbers, common sense would tell a person that a larger frontal area would have more of an effect on the surrounding fluid than a smaller frontal area. At a low Reynolds number, the inertia forces are smaller than the viscous forces, this makes it possible for the disturbances to dissipate easily which helps laminar flow remain, whereas at a high Reynolds number the inertia forces are adequately large to increase disturbances, so the motion becomes increasingly unstable. (Tu, Yeoh, & Liu, 2013). However just because an object is small it doesn't necessarily mean turbulence will not occur because there will always be disturbances if an object is

travelling through a fluid, these disturbances generally increases with velocity. Although this may be bad for a race team trying to increase the speed of their vehicle, in combustion chambers turbulence helps the flame speed increase to make it possible for engines to run at higher RPMs due to the increase of energy in the fuel/air mix which means the ignition timing doesn't have to be retarded too much, thus turbulence is not always bad. (Glassman, Yetter, & Glumac, 2014)

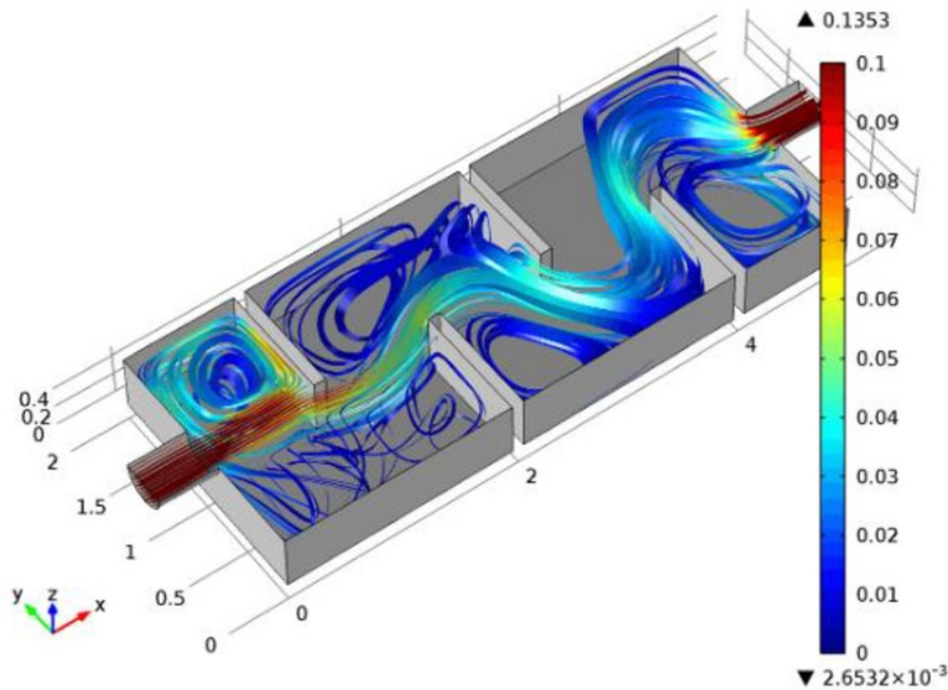


Figure 2 - Example of Turbulence, (Comsol, 2019)

Mesh

Computational fluid dynamics uses a finite difference method to solve numerical equations. This means the software uses numerical methods to solve equations by approximation by converting non-linear equations into linear ones, which are solved using algebraic techniques. CFD software uses this method to provide a number of locations in the build grid, the arrangement of these locations is called the mesh. The type and density of the mesh used is significant to the simulation, which includes the results, as a given geometry must be well defined by the mesh so the numerical simulation can accurately represent the geometry. (Munson, Okiishi, Huedsch, & Rothmayer, 2013).

Mesh Type

When creating a volume mesh in ICEM CFD the two common mesh options are “Robust Octree” and “Quick Delaunay”. A robust octree mesh works by using a top down method, which fills in the volume mesh by working from the top of the domain down. Input parameters are set, for example the maximum element size of different components including the boundary conditions, this tells the meshing software that no element can be larger than the set inputs. The octree method does not require a surface mesh as it creates its own during the meshing process. The algorithm used refines the mesh where required and maintains large elements where possible to save on computational time. The “Tetra” mesher then balances the mesh so edges of faces that share elements don’t vary by a factor of 2. At this point the mesh still doesn’t match the geometry so nodes are rounded to fit the geometry better and uses a “cutting” method to clear any mesh that cannot meet the required inputs without intersecting a surface. Lastly the nodes are moved around, joined together or even deleted to create a smoother mesh. (Sharcnet, 2015)

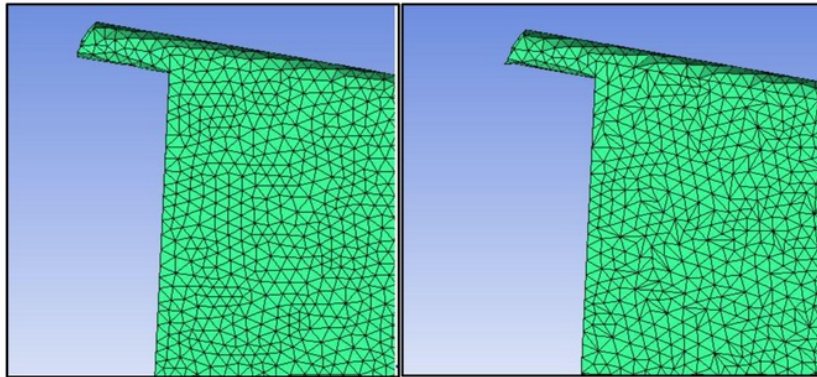


Figure 3 - Examples of Octree Mesh (left) and Quick Delaunay Mesh (right), (Randall, 2018)

The Delaunay method uses the opposite approach to Octree, it uses a “bottom-up” approach to build the mesh using the “Delaunay Tetra Algorithm”. For this method to work an existing surface mesh is required, it uses this surface mesh and works outwards to form the volume mesh and gradually increases the element size until maximum specifications are met. This can be useful if importing geometry with an already existing mesh, as the user can select the existing mesh as an input. (sharcnet, 2015). However, a “Quick Delaunay” mesh doesn’t appear as structured as a “Robust Octree” mesh (see figure 3) and requires the geometry to be of a higher quality.

Mesh Sizing

To create an accurate representation of the geometry being used for analysis, mesh sizing must be considered carefully, as mentioned previously, if the mesh is not refined enough to visually represent the geometry then results will be inaccurate, see figure 12. However, creating a mesh that is too refined is also unacceptable as computational time will increase. Therefore, correct element sizing is completely dependent on the size and shape of the geometry. (Ozen, 2014)

Prism Layers

Prism layers are used in ICEM CFD to represent boundary layers which naturally occur when bodies are passing through fluids. ICEM CFD uses “prism and tetra generation” to create these boundary layers which is necessary to capture what is happening perpendicular to surfaces. This allows more elements and nodes to be perpendicular to the surface which allows for more calculations closer to the surface giving more resolution to what is happening nearer the surface without increasing the number of elements, which in turn gives more accurate results without increasing computation time. (Sharcnet, 2015).

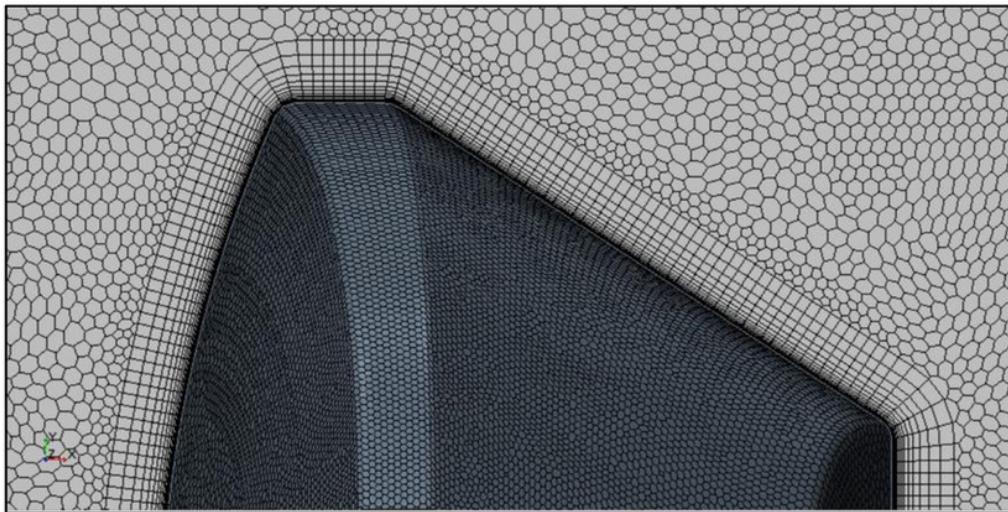


Figure 4 - Example of Prism Layers, (Chhunchha, 2018)

Orthogonal Quality and Skewness

Orthogonal quality represents how tidy cells are, adjacent to each other. It works by using vectors from the centre of one element and seeing how close that vector is to the centre of the elements around it, see figure 5. The range for orthogonal quality is 0-1 where 1 is good and 0 is poor. Skewness and Orthogonal quality work hand in hand as the skewness is determined by subtracting the orthogonal quality from 1, therefore a skewness of 1 is poor and 0 is good, see table 3. (Sharqnet, 2015). However, this doesn't mean the orthogonal quality has to be 1 or skewness must be 0, there are acceptable figures which is shown in the table below.

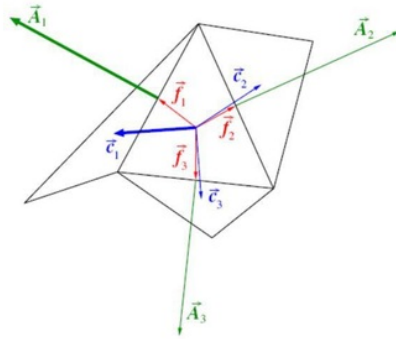
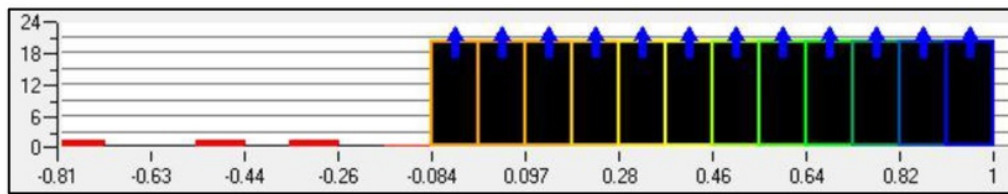


Figure 5 - How Orthogonal Quality is Calculated, (Sharqnet, 2015)

Table 1 - Example of Orthogonal Quality



(Randall, 2018)

Table 2 - Orthogonal quality spectrum

Unacceptable	Bad	Acceptable	Good	Very good	Excellent
0-0.001	0.001-0.14	0.15-0.20	0.20-0.69	0.70-0.95	0.95-1.00

(Ansys, 2012)

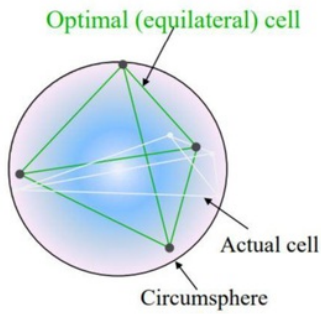


Figure 6 - Skewness, (Ansys, 2012)

Table 3 - Skew Spectrum

					
Excellent	Very good	Good	Acceptable	Bad	Unacceptable
0-0.25	0.25-0.50	0.50-0.80	0.80-0.94	0.95-0.97	0.98-1.00

(Ansys, 2012)

Turbulence Models

The most commonly used turbulence model today for bluff bodies is the K-epsilon (realizable) model which is a variant of the original k-epsilon model. The standard model solves for two variables, the rate of dissipation of kinetic energy (epsilon) and the turbulent kinetic energy (k). This model analytically takes into account the “fluid velocity in the viscous sublayer near the wall” using wall functions (Wasserman, 2016). This method is good for general purpose CFD with external flow properties and typically isn’t too memory intensive. However, this model does have some limitations which include no-slip walls, strong curvatures and occasionally difficulty solving for epsilon, so variants of this model have been made. Most famous of which is the realizable K-epsilon model which helps epsilon solve by introducing “mean flow distortion on turbulent dissipation” which is now the default model in main CFD packages.

Another popular turbulence model is k-omega, the difference between this and k-epsilon is that k-omega picks up more accuracy at near-wall surfaces, however overall is not as reliable as k-epsilon and sometimes has difficulty converging. (Wasserman, 2016) Even so, a variant of this model has gained acceptance in the aeronautical industry due to its ability to foresee separation and attachment more than the k-epsilon and standard omega models, its named SST K-omega. This model is similar to both k-epsilon and k-omega therefore has been a contender for a general use turbulence model as it works like k-epsilon in the free stream and like k-omega close to walls. (Wasserman, 2016)

Reynolds Number

The Reynolds number (Re) has a vital part in predicting a fluids behaviour. It determines if the fluid is in a laminar or turbulent state and is fundamental in viscous flows using numerical models with a pre-determined Reynolds number. "The dimensionless Reynolds number" relates to properties such as length, viscosity, velocity and the type of flow (SimScale, 2017). It can easily be expressed as the ratio of inertia forces to viscous forces which can be seen below.

Equation 1 - Reynolds Number Equation

$$Re = \frac{\rho V L}{\mu} = \frac{V L}{\nu}$$
$$Re = \frac{F_{inertia}}{F_{viscous}} = \frac{\frac{kg}{m^3} \times \frac{m}{s} \times m}{Pa \times s} = \frac{F}{F}$$

(SimScale, 2017)

Methodology and Results

It should be noted that a base set up was used for preliminary tests to be able to obtain results and that chosen results have been highlighted in red for ease of reading. The settings used for this are below in table 4. From then on, the best settings from the previous results were used. Also, any simulations requiring more than twenty hours of computational time are considered too long therefore the analyst chose not to run these scenarios.

Table 4 - Base settings used for simulations

Base Settings	
Fluid Velocity (m/s)	30
Domain Mesh Size (m)	1
Density Box Size	1 Part in-front, 4 behind
Density Box Mesh Size (m)	0.1
Myra Car Mesh Size (m)	0.1
Number of Prism Layers	3
Model	Viscous
Turbulence Model	K-epsilon (realizable)
Wall Treatment	Non-equilibrium wall functions

Domain Size Study

For the domain size study, a total of eleven sets of dimensions have been analysed for which can be seen in the table below. (See table 5)

Table 5 - Domain Size Study

Domain Size	cd	cl	Datum cd
15X, 7.9Y, 4.4Z (Mira Tunnel Size)	0.373	-0.0257	0.2963
38X, 4Y, 4Z (9x Mira, 3x Height)	0.371	-0.0267	0.2963
25X, 4Y, 4Z (6x Mira, 3x Height)	0.383	-0.071	0.2963
35X, 5Y, 4Z (8x Mira, 4x Height)	0.386	-0.0394	0.2963
37.8X, 8Y, 6Z (9x Mira, 6x Height)	0.349	-0.0121	0.2963
21X, 6.5Y, 9.25Z (5x Mira All)	0.353	-0.0771	0.2963
25x, 8Y, 10Z (6x Mira All)	0.365	-0.01	0.2963
29X, 9Y, 12Z (7x Mira All)	0.346	-0.0394	0.2963
33X, 10.5Y, 13Z (8x Mira All)	0.335	-0.545	0.2963
36X, 12Y, 14.5Z (9x Mira All)	0.364	-0.0482	0.2963
50X, 8Y, 16Z (Design Modeler 2013)	0.362	-0.0744	0.2963

Datum cl	Difference cd	Difference cl	Δ of cd/cl for Best Results
-0.0845	0.0767	0.0588	
-0.0845	0.0747	0.0578	
-0.0845	0.0867	0.0135	
-0.0845	0.0897	0.0451	
-0.0845	0.0527	0.0724	
-0.0845	0.0567	0.0074	0.0641
-0.0845	0.0687	0.0745	
-0.0845	0.0497	0.0451	
-0.0845	0.0387	0.4605	
-0.0845	0.0677	0.0363	
-0.0845	0.0657	0.0101	

(Randall, 2018)

It was found that for the domain size, the best results obtained were by setting the dimensions of the domain to 21m in the X axis, 6.5m in the Y axis and 9.25m in the Z axis, which is roughly 5x the dimensions of the Mira car. There are other dimensions which worked better for either drag or lift coefficients but overall these dimensions give the closest results to the datum inputs which were supplied by Tim Tudor. It can be seen in the graph below (See figure 7) that making the domain larger doesn't necessarily increase the accuracy of results.

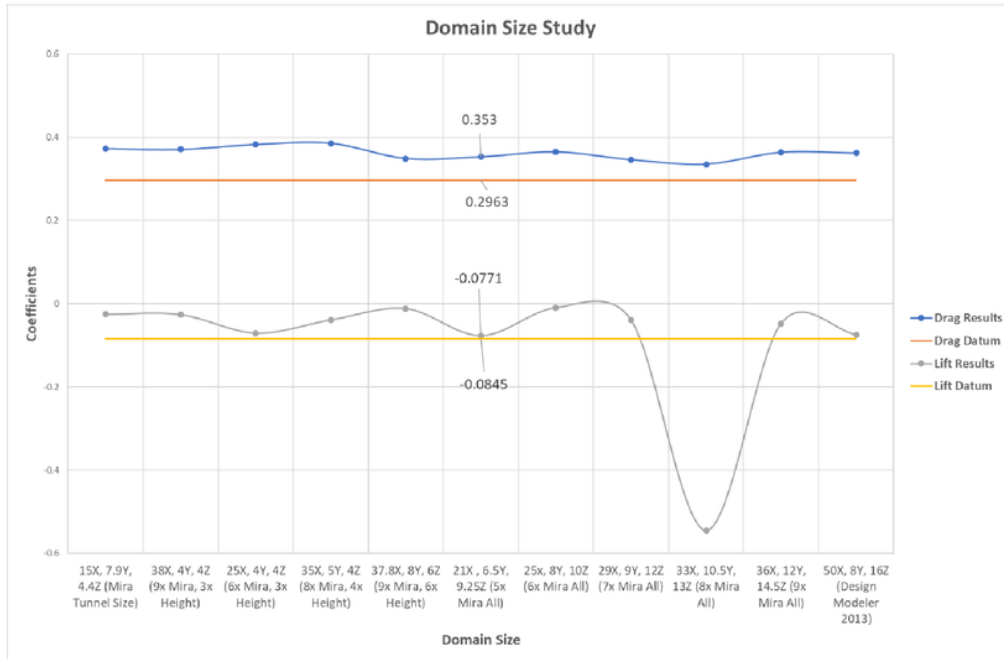


Figure 7 - Domain Size Study, (Randall, 2018)

Car Placement Study

For the car placement study, the most accurate run from the domain size study has been used for the domain settings. A total of ten simulations were completed to find the best location in the domain to place the reference geometry, the results can be seen in table 6 below.

Table 6 - Mira Car Placement Study

Myra Car Placement Study	Car Position	cd	cl	Datum cd
1	1.5m from VI	0.4417	-0.0428	0.2963
2	3m from VI	0.3619	-0.072	0.2963
3	4.5m from VI	0.3664	-0.0234	0.2963
4	6m from VI	0.341	-0.0556	0.2963
5	7.5m from VI	0.346	-0.00345	0.2963
6	9m from VI	0.366	-0.016	0.2963
7	10.5m from VI	0.373	-0.0615	0.2963
8	12m from VI	0.374	-0.0087	0.2963
9	13.5m from VI	0.362	-0.0235	0.2963
10	15m from VI	0.378	-0.0582	0.2963

Datum cl	Difference cd	Difference cl	Δ of cd/cl for Best Results
-0.0845	0.1454	0.0417	
-0.0845	0.0656	0.0125	0.0781
-0.0845	0.0701	0.0611	
-0.0845	0.0447	0.0289	0.0447
-0.0845	0.0497	0.08105	
-0.0845	0.0697	0.0685	
-0.0845	0.0767	0.023	
-0.0845	0.0777	0.0758	
-0.0845	0.0657	0.061	
-0.0845	0.0817	0.0263	

(Randall, 2018)

The study found that the best location for the Mira car was for the front face to be allocated 6m from the velocity inlet in the X axis, see figure 11. Placing the car there allowed the wake region to be better captured. When looking at a comparison of the car being placed further from the velocity inlet it didn't have much effect on the flow around the front of the car, see figure 8. Also looking at a comparison between the car being placed 1.5m and 6m away from the velocity inlet shows that the velocity of the fluid travelling under and over the car is generally higher and is also causing more turbulence around the car which is increasing the vortices formed in the wake region subsequently causing a larger difference in drag to the datum results, see figures 9 and 10.

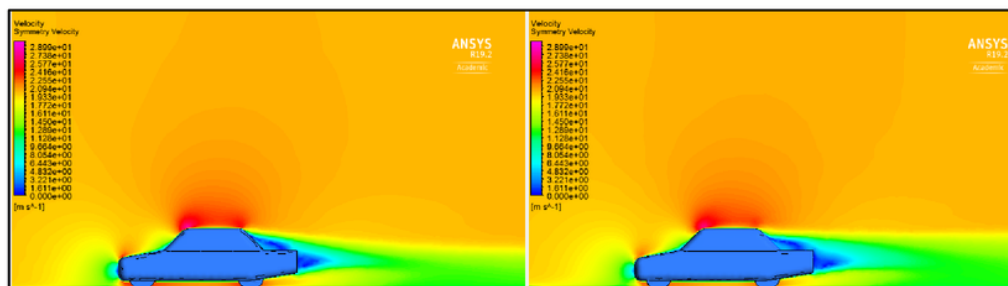


Figure 8 - Geometry Placement Comparison, 6m from VI (left), 12m from VI (right), (Randall, 2018)

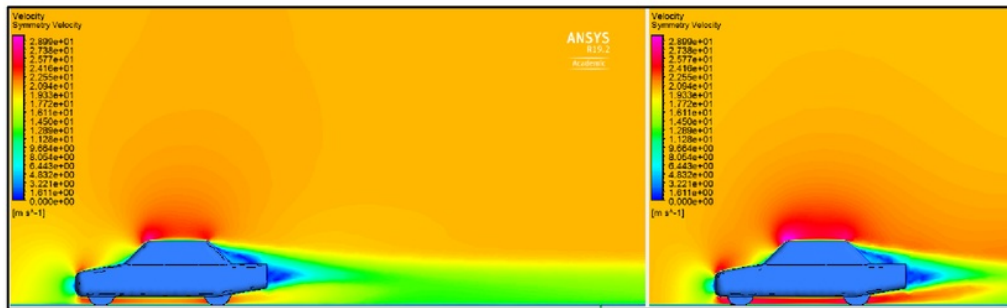


Figure 9 - Geometry Placement Comparison, 6m from VI (left), 1.5m from VI (right), (Randall, 2018)

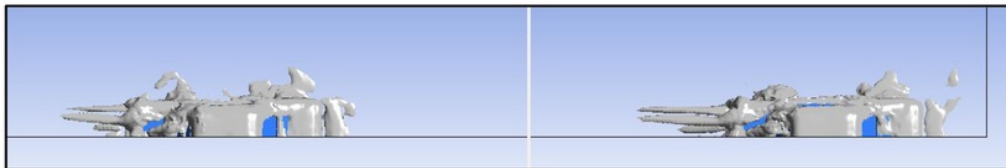


Figure 10 - Turbulence Comparison between Geometry being placed 6m from VI (left) and 1.5m from VI (right), (Randall, 2018)

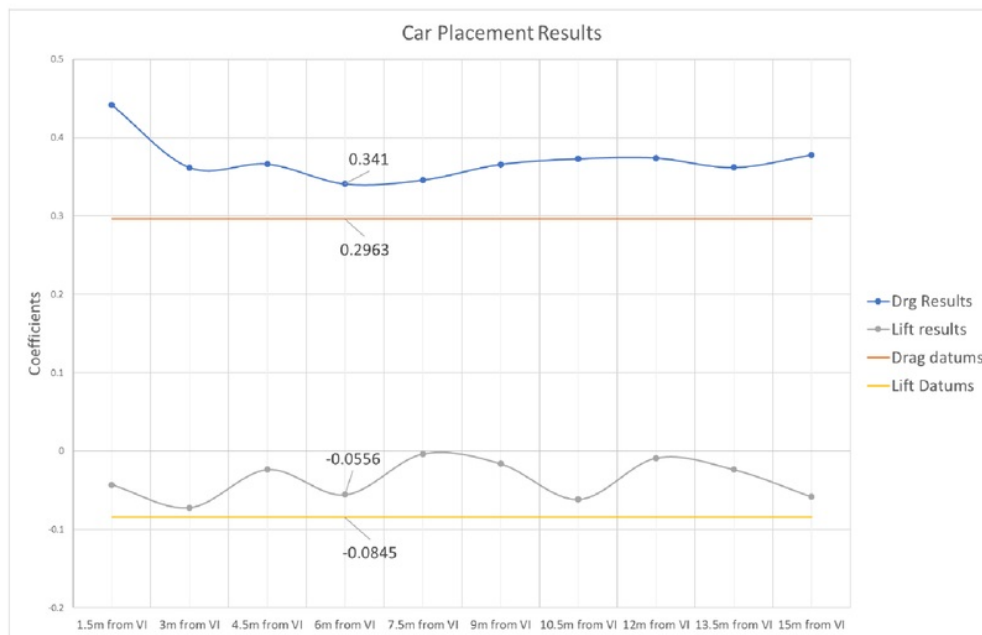


Figure 11 - Reference Geometry Placement in Domain Results, (Randall, 2018)

Car Mesh Study

A total of ten simulations were calculated to obtain the closest results to the drag and lift coefficients obtained from the datum results. It was found that the most accurate mesh density for the reference geometry is 0.04m, see table 7. However, looking at the lift results a mesh density of 0.07m produced a closer lift coefficient. As the whole study had not been completed by this point it is most likely that this was by chance, as a mesh density of 0.7 does not accurately represent the geometry of the Mira car, see figure 12. With a setting of 0.4m for the mesh density, the representation of the Mira car is much better although still not perfect, however refining the mesh further produced less accurate results (See figure 13) so a choice was made to continue with 0.4m.

Table 7 - Mira Car Mesh Density Results Table

Mira Car Mesh Size (m)	cd	cl	Datum cd
0.1	0.341	-0.0556	0.2963
0.09	0.335	-0.00295	0.2963
0.08	0.3259	-0.0972	0.2963
0.07	0.329	-0.08	0.2963
0.06	0.324	-0.185	0.2963
0.05	0.316	-0.164	0.2963
0.04	0.303	-0.11425	0.2963
0.03	0.2825	-0.119	0.2963
0.02	0.287	-0.1306	0.2963
0.01	0.2802	-0.1098	0.2963

Datum cl	Difference cd	Difference cl	Δ of cd/cl for Best Results
-0.0845	0.0447	0.0289	
-0.0845	0.0387	0.08155	
-0.0845	0.0296	0.0127	
-0.0845	0.0327	0.0045	0.0372
-0.0845	0.0277	0.1005	
-0.0845	0.0197	0.0795	
-0.0845	0.0067	0.02975	0.03645
-0.0845	0.0138	0.0345	
-0.0845	0.0093	0.0461	
-0.0845	0.0161	0.0253	

(Randall, 2018)

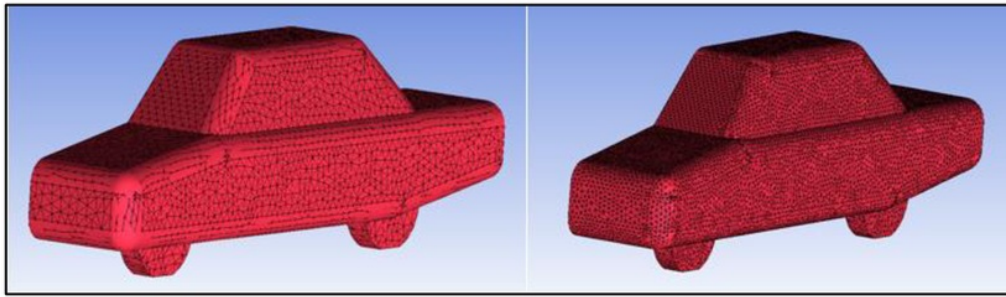


Figure 12 - Mesh Density Comparison, 0.7m (left) 0.4m (right), (Randall, 2018)

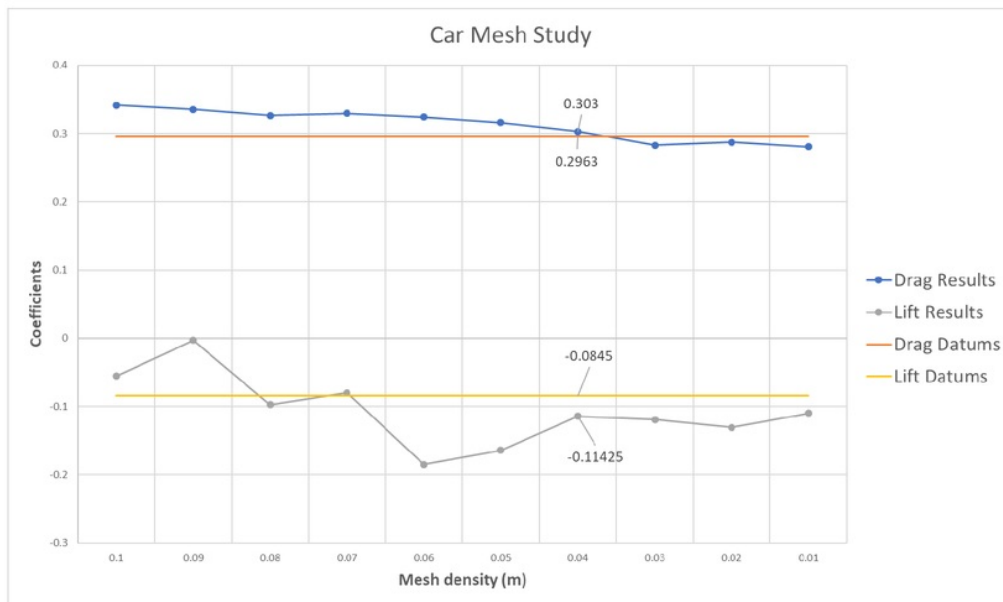


Figure 13 - Car Mesh Study, (Randall, 2018)

Density Box Size Study

Choosing the size of the density box is an important part of the mesh process. The idea of a density box is to allow fluent to capture a higher percentage of detail in selected areas instead of having to refine the whole domain, to save on computational time. A total of ten simulations were completed to see what size density box is best suited to increase the level of detail whilst at the same time capturing the wake region without causing simulation times to increase, see table 8. It was found that using a density box four times the size of the Mira reference car (from the centroid) had the best results in terms of maintaining run time efficiency without compromising the accuracy of the produced results. A density box 1.5 times the size of Mira produced the best results for Cd, meaning it was the closest result to the datum Cd figure, however it did not completely capture the wake region, see figure 14, also, a density box five times the size of the Mira car (from the centroid) produced the best

results for C_l , however the difference in result accuracy between the run at four times and five times the size are minimal (0.0002), so the choice was made to use a density box four times the size of the reference geometry.

Table 8 - Density Box Size Study Results

Density Box Size	cd	cl	Datum cd
1x Mira	0.3256	-0.1158	0.2963
1.5x Mira	0.30311	-0.1142	0.2963
2x Mira	0.30311	-0.1142	0.2963
3x Mira	0.3043	-0.1587	0.2963
4x Mira	0.30318	-0.11442	0.2963
5x Mira	0.303	-0.114	0.2963
6x Mira	n/a	n/a	0.2963
7x Mira	n/a	n/a	0.2963
8x Mira	n/a	n/a	0.2963
9x Mira	n/a	n/a	0.2963

Datum cl	Difference cd	Difference cl	Δ of cd/cl for Best Results
-0.0845	0.0293	0.0313	
-0.0845	0.00681	0.0297	
-0.0845	0.00681	0.0297	0.03651
-0.0845	0.008	0.0742	
-0.0845	0.00688	0.02992	0.0368
-0.0845	0.0067	0.0295	0.0362
-0.0845	0	0	Too Long (20+ hours)
-0.0845	0	0	Too Long (20+ hours)
-0.0845	0	0	Too Long (20+ hours)
-0.0845	0	0	Too Long (20+ hours)

(Randall, 2018)

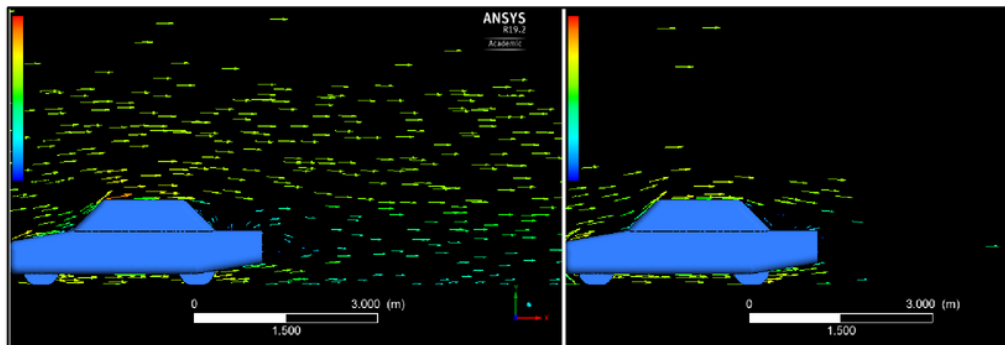


Figure 14 - Density Box Comparison, 4x Geometry Volume (left), 1.5x Geometry Volume (right), (Randall, 2018)

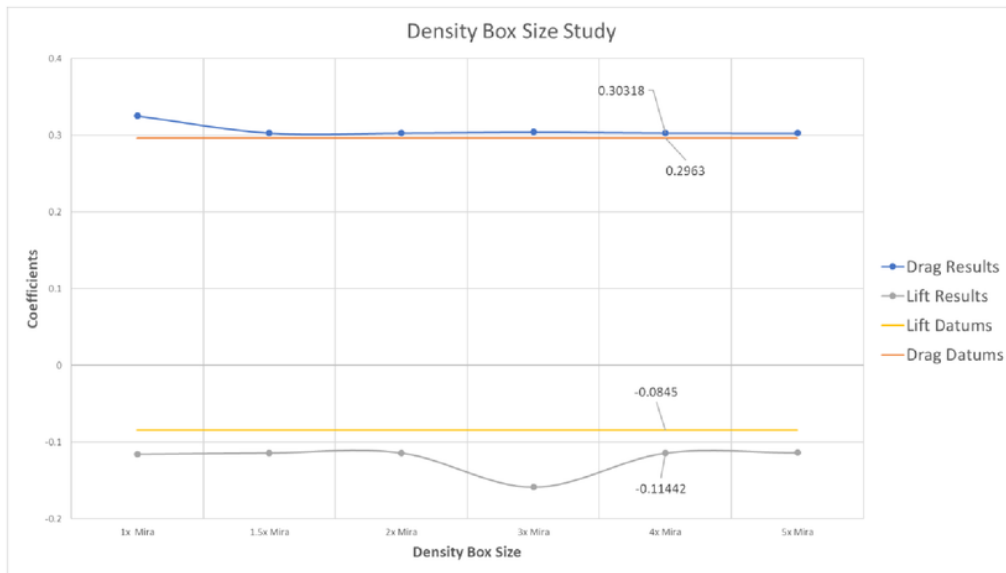


Figure 15 - Density Box Size Study Graph of Results, (Randall, 2018)

Density Box Position Study

The placement of the density box in domain is dependent on what region the analyst deems an area of interest. A maximum of seven simulations were completed, with differences of one meter in the x axis. As the study with the density box placed four meters from the velocity inlet produced results closest to the datums, this study was chosen to move forward with, see table 9. As the density box needed to surround the reference geometry, no more than seven simulations could be ran. This was due to the front of the density box eventually passing the front of the car, therefore defeating the object of the density box. Looking at the comparison below, see figure 16, capturing detail both in front and behind the geometry required the front face of the density box to be placed four meters from the velocity inlet, which also allowed more wake region to be captured, apart from the wake region there is no noticeable difference of fluid flow between the studies with the density box being placed two meters and a density box being placed four meters from the velocity inlet.

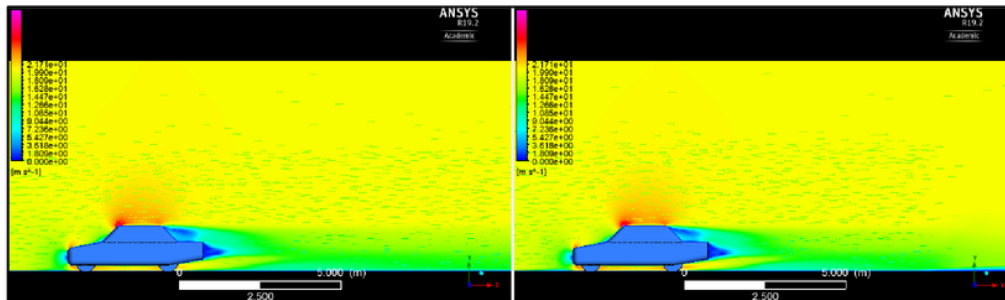


Figure 16 - Comparison of Density Box being placed 4m from the VI (left) and 2m from the VI (right), (Randall, 2018)

Table 9 - Density Box Position Results

Density Box Position	cd	cl	Datum cd
0m from VI	0.298	-0.142	0.2963
1m from VI	0.303	-0.136	0.2963
2m from VI	0.288	-0.112	0.2963
3m from VI	0.301	-0.13	0.2963
4m from VI	0.304	-0.0804	0.2963
5m from VI	0.31	-0.089	0.2963
6m from VI	0.312	-0.136	0.2963

Datum cl	Difference cd	Difference cl	Δ of cd/cl for Best Results
-0.0845	0.0017	0.0313	0.033
-0.0845	0.0067	0.052	
-0.0845	0.0083	0.0275	
-0.0845	0.0047	0.0455	
-0.0845	0.0077	0.0041	0.0118
-0.0845	0.0137	0.0045	
-0.0845	0.0157	0.0515	

(Randall, 2018)

Density Box Mesh Size Study

As mentioned previously, the density box is used to capture detail in specific areas. Therefore, creating a density box with a mesh that can adequately capture enough detail without increasing computations time was important, see figure 17. Ten studies were completed with the sixth study producing the closest results to the datum coefficients with the density box mesh size set to 0.09m, despite this, study five with a mesh density of 0.1m produced results with a difference of 0.001 therefore those settings were chosen to save time.

Table 10 - Density Box Mesh Size Study Results

Density Box Mesh Size	cd	cl	Datum cd
0.5	0.315	-0.094	0.2963
0.4	0.315	-0.0942	0.2963
0.3	0.307	-1.42	0.2963
0.2	0.308	-0.142	0.2963
0.1	0.304	-0.0804	0.2963
0.09	0.303	-0.0804	0.2963
0.08	0.309	-0.0804	0.2963
0.07	n/a	n/a	0.2963
0.06	n/a	n/a	0.2963
0.05	n/a	n/a	0.2963

Datum cl	Difference cd	Difference cl	Δ of cd/cl for Best Results
-0.0845	0.0187	0.0095	
-0.0845	0.0187	0.01	
-0.0845	0.0107	1.3355	
-0.0845	0.0117	0.0575	
-0.0845	0.0077	0.0041	0.0118
-0.0845	0.0067	0.0041	0.0108
-0.0845	0.0127	0.0041	
-0.0845	#VALUE!	#VALUE!	Too Long (20+ hours)
-0.0845	#VALUE!	#VALUE!	Too Long (20+ hours)
-0.0845	#VALUE!	#VALUE!	Too Long (20+ hours)

(Randall, 2018)

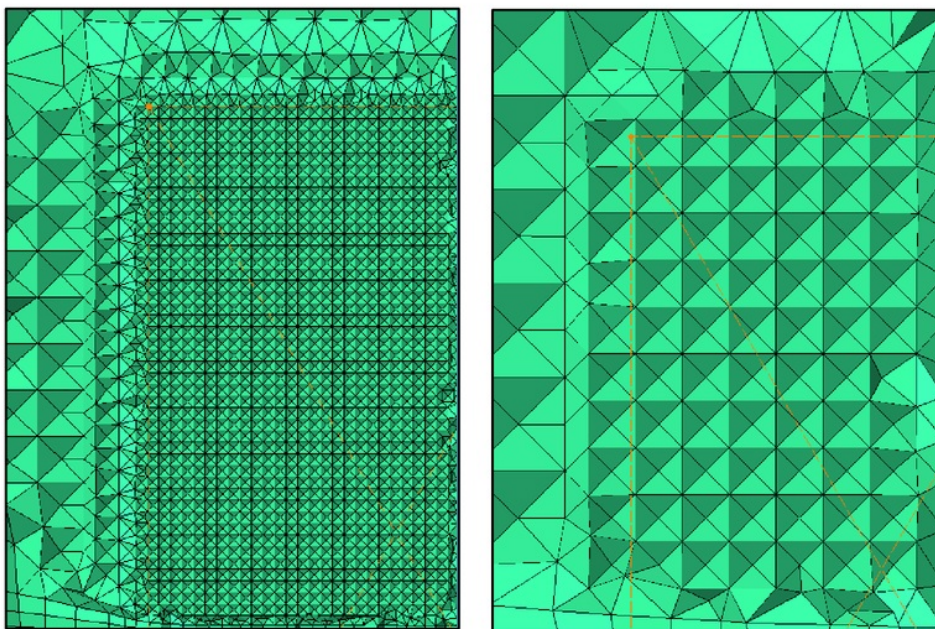


Figure 17 - Comparison of a fine mesh density (left) and a large mesh density (right), (Randall, 2018)

Prism Layer Study

As previously stated, prism layers are used to represent boundary layers which capture close to surface fluid movement. This study showed that using three prism layers presented the closest results to the datum coefficients, see table 11. However, the entire study thus far has been completed using a base set up with three prism layers. In hindsight this has tailored the previous studies settings to facilitate three prism layers to give the best results, so these results are questionable.

Table 11 - Prism Layer Study Results

Prism Layers	cd	cl	Datum cd
1	0.297	-0.0969	0.2963
2	0.311	-0.0934	0.2963
3	0.304	-0.0804	0.2963
4	0.301	-0.134	0.2963
5	0.299	-0.163	0.2963
6	0.312	-0.128	0.2963
7	0.2909	-0.1225	0.2963
8	0.292	-0.106	0.2963
9	0.291	-0.0911	0.2963
10	0.291	-0.0929	0.2963

Datum cl	Difference cd	Difference cl	Δ of cd/cl for Best Results
-0.0845	0.0007	0.0124	
-0.0845	0.0147	0.0089	
-0.0845	0.0077	0.0041	0.0118
-0.0845	0.0047	0.0495	
-0.0845	0.0027	0.0785	
-0.0845	0.0157	0.0435	
-0.0845	0.0054	0.038	
-0.0845	0.0043	0.0215	
-0.0845	0.0053	0.0066	
-0.0845	0.0053	0.0084	

(Randall, 2018)

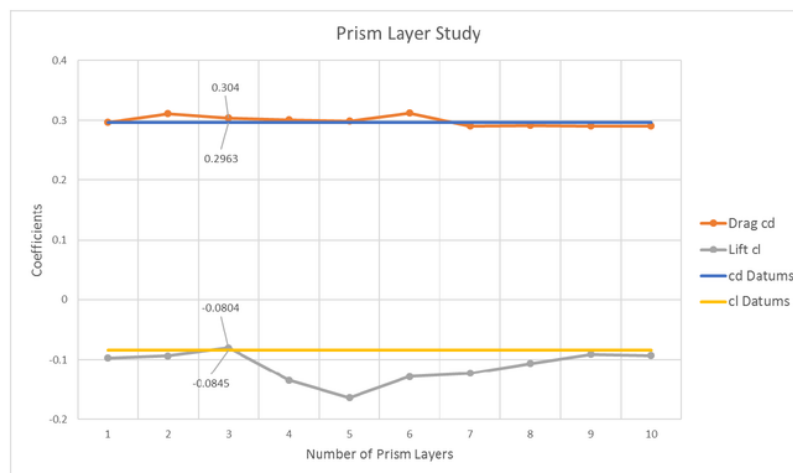


Figure 18 - Graph of Prism Layer Study Results, (Randall, 2018)

Turbulence Model Study

The turbulence model which produced the most accurate results was K-epsilon, realizable, standard wall functions. This model worked best for both situations where the top and sides of the domain are set to walls or symmetry, this can be seen in the tables below, see table 12. When looking at the pressure taps of the upper and lower surfaces of the reference geometry (See figure 19) it can be said that although the pressures do not match up exactly, the general flow is similar. This suggests the turbulence model used is mostly correct at picking up pressure distribution, however it struggles to detect where separation occurs on the rear slant of the reference car. The model seems to be over anticipating where separation occurs which suggests it also struggles to predict close to surface scenarios which backs up what has been previously mentioned about the k-epsilon turbulence model.

Table 12 - Turbulence Model Study Results

Turbulence Model	cd	cl	Datum cd
K-epsilon, standard, SWF	0.341	-0.038	0.2963
K-epsilon, standard, Non-eq wf	0	0	0.2963
K-epsilon, standard, evt	0.326	-0.0418	0.2963
K-epsilon, realizable, swf	0.303	-0.0823	0.2963
K-epsilon, realizable, Non-eq wf	0.304	-0.0804	0.2963
K-epsilon, realizable, evt	0.304	-0.0795	0.2963
K-omega, Standard	0.306	-0.0129	0.2963
K-omega, SST	0.32	-0.003	0.2963
Run 4 mira tunnel mods	0.395	-0.368	0.2963
Topside set as walls			
K-epsilon, realizable, swf	0.308	-0.085	0.2963
K-epsilon, realizable, evt	0.308	-0.0833	0.2963
K-epsilon, realizable, swf, With Mira wind tunnel Dimensions	0.324	-0.103	0.2963
K-epsilon, realizable, swf, With Mira wind tunnel Dimensions + Density around rear slant	0.319	-0.1	0.2963

Datum cl	Difference cd	Difference cl	Δ of cd/cl for Best Results
-0.0845	0.0447	0.0465	
-0.0845	0.2963	0.0845	FAIL
-0.0845	0.0297	0.0427	
-0.0845	0.0067	0.0022	0.0089
-0.0845	0.0077	0.0041	
-0.0845	0.0077	0.005	
-0.0845	0.0097	0.0716	
-0.0845	0.0237	0.0815	
-0.0845	0.0987	-0.2835	
Topside set as walls			
-0.0845	0.0117	0.0005	0.0122
-0.0845	0.0117	0.0012	
-0.0845	0.0277	0.0185	
-0.0845	0.0227	-0.0155	

(Randall, 2018)

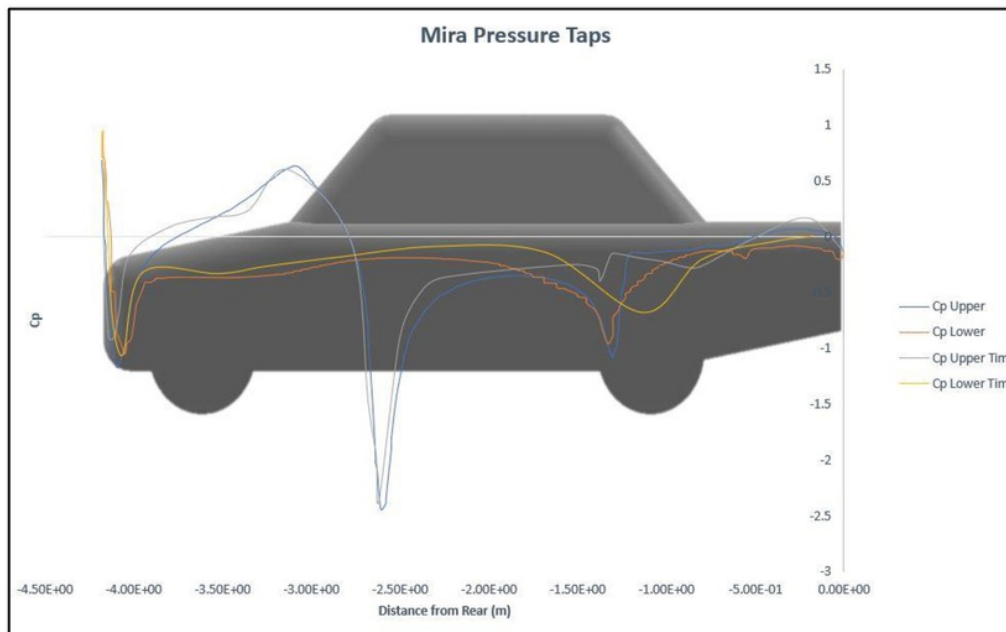


Figure 19 - Pressure Taps for Upper and Lower Surfaces of Mira Reference Car, (Randall, 2018)

Modified Mira Car

Some studies were conducted by adding a “rocket bunny” style spoiler to the reference geometry, as well as a bonnet scoop, to examine the effects they generate in terms of drag, downforce, and efficiency. It was found that the efficiency of the Mira car improved when compared to the original model by a factor of roughly 3.54, which can be seen below in table 13 below. The overall efficiency however still is not ideal, the table shows that even with the modifications, for every newton of drag, there is less than a newton of downforce.

Table 13 - Efficiency between Original model & Modified Model

Model	Drag Cd	Downforce (-Cl)	Efficiency (%)	Improvement Factor
K-epsilon, realizable, swf	0.303	0.08	26.40	
K-epsilon, realizable, swf, Modified	0.395	0.37	93.67	3.54778481

(Randall, 2018)

One result which was not expected is that less recirculation is occurring on the rear slant of the modified geometry. this could be due to the flow that previously was being recirculated being captured by the up wash from the spoiler therefore carrying it away. See figures 20 and 21

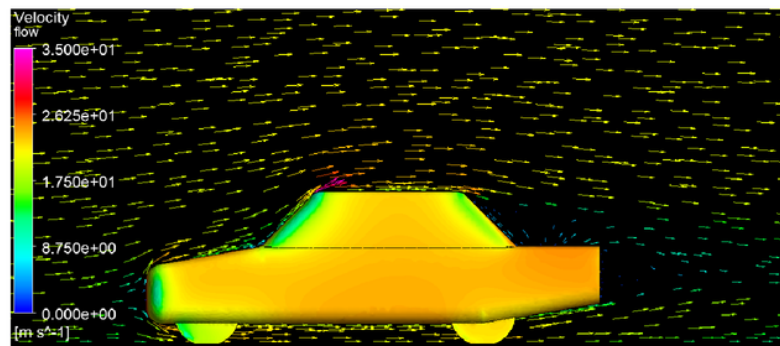


Figure 20 - Flow contours of Original Geometry, (Randall, 2018)

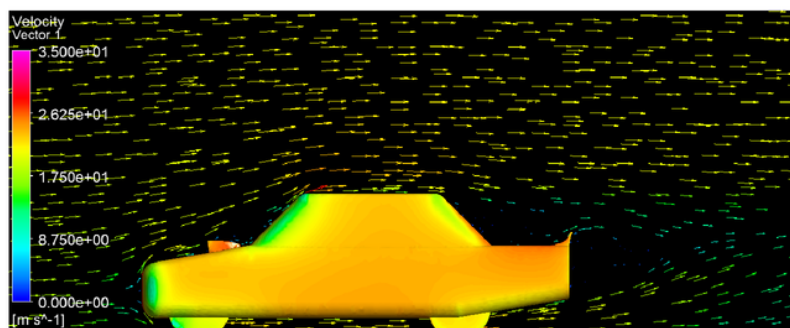


Figure 21 - Flow Contours of Modified Geometry, (Randall, 2018)

It can be seen from looking at figures 22 and 23 that the addition of a bonnet scoop has created a pressure build up on the bonnet of the Mira car, **in comparison to the original**. This, as well as the build up from the spoiler is the reason for the increase in drag. However separate simulations of both components should be ran to see how the individual components affect both lift and drag.

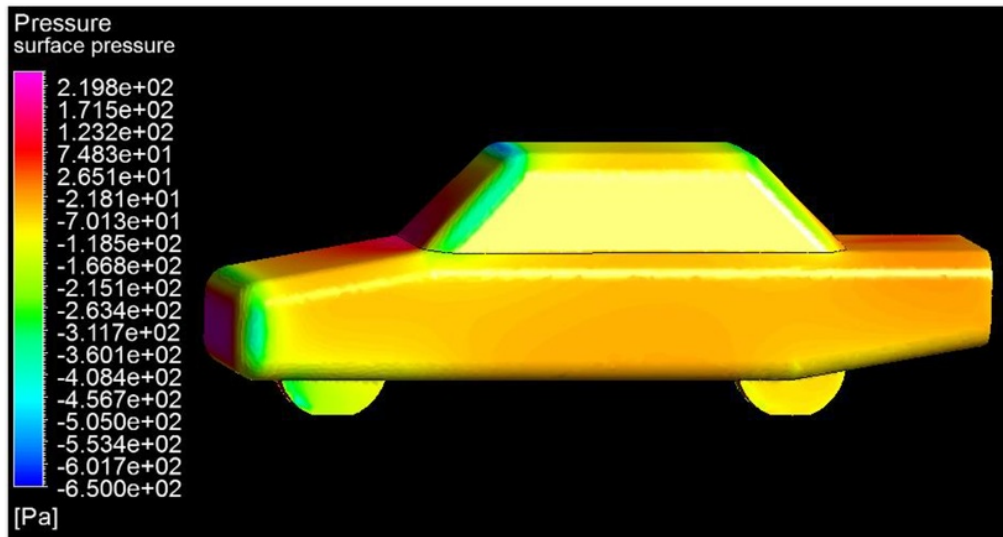


Figure 22 - surface Pressure contours of Original Geometry, (Randall, 2018)

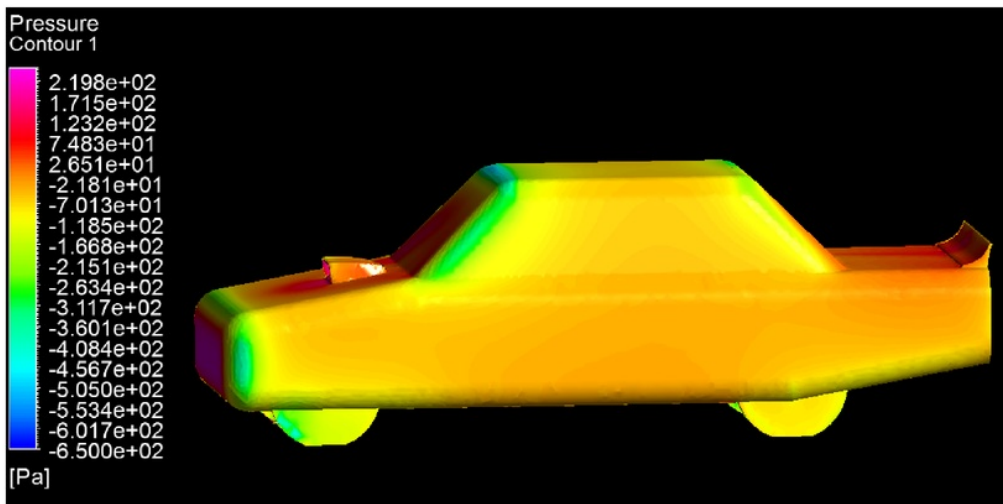


Figure 23 - Surface Pressure Contours of Modified Geometry, (Randall, 2018)

Below is an image showing how the spoiler is affecting the fluid flow over the tail of the Mira car. As expected it is influencing the flow to rise above the spoiler which is causing some downforce, however it is creating a larger region directly behind that car to have less of a laminar flow structure, causing drag, see figure 24.

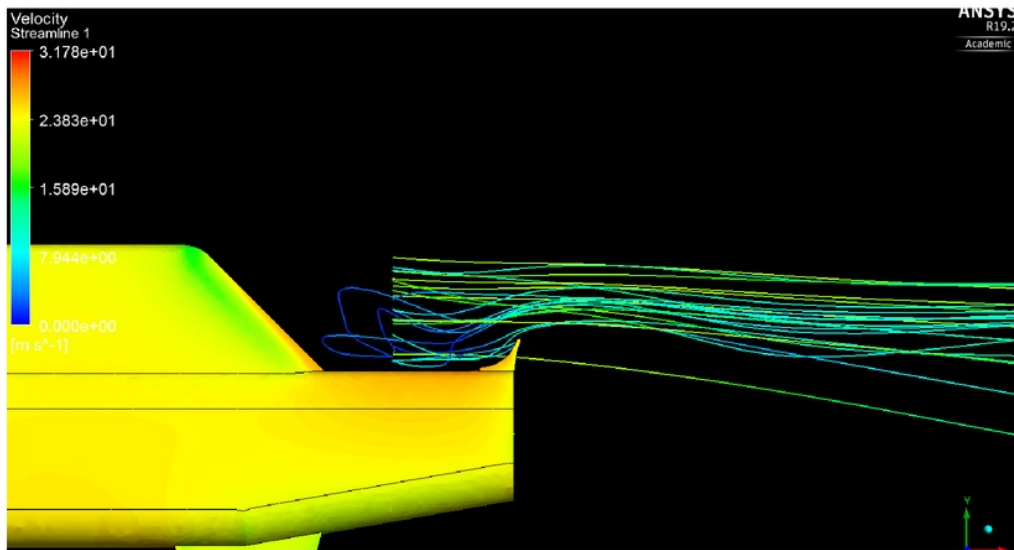


Figure 24 - Streamline effect of Rocket Bunny spoiler, (Randall, 2018)

Conclusion

It is inappropriate to comment on the credibleness and validity of this study, due to the fact that a selection of base settings were used that may have distorted results from the start of the study. However, the results produced from this study suggest a 2.2% margin of error for the coefficient of drag and approximately a 2.67% margin of error for the coefficient of lift which can be taken as a positive outcome. Further work could be completed to gain knowledge of more accurate settings to use for the same kind of body used in this study, which could produce findings which suggest the use of different settings.

As CFD simulation cannot be 100% accurate, it should be used alongside wind tunnel testing. It is relatively accurate at foreseeing how fluid will flow around a body. Therefore, CFD is useful at predicting the changes in fluid over a geometry caused by any adjustments, which can be confirmed in a wind tunnel or real-life testing.

The modified geometry results show a considerable increase in downforce. Unfortunately, this also results in a slight increase in drag, albeit not at the same rate. It is impossible to say whether this is beneficial as it would be determined on a case by case basis.

Bibliography

Ansys. (2012, March). Introduction to Ansys Meshing.

Building flow domains with ANSYS design modeler. (2013, February 7). Retrieved from CAE ASSOCIATES : https://caeai.com/sites/default/files/Flow_Domains_ANSYS_DM_CAEA_0.pdf

Cheng, A., & D, T. C. (2005). *Numerics*. Retrieved from Simscale:
<https://www.simscale.com/docs/content/simwiki/numerics/what-are-boundary-conditions.html>

Chhunchha, A. (2018, May). *AERODYNAMIC HEATING ANALYSIS OF RE-ENTRY SPACE CAPSULE USING COMPUTATIONAL FLUID DYNAMICS*. Retrieved from ResearchGate:
https://www.researchgate.net/figure/Close-up-view-of-the-near-wall-mesh-showing-prism-layers_fig3_326044537

Comsol. (2019). *Navier-Stokes Equations*. Retrieved from Comsol:
<https://www.comsol.com/multiphysics/navier-stokes-equations>

Day, S. (2018). *Fluid Flow*. Retrieved from A Level Notes:
<https://alevelnotes.com/notes/physics/fluids/fluid-flow>

Glassman, I., Yetter, R., & Glumac, N. (2014). *Combustion*. Academic Press.

Jeffords, A. (2004). *Laminar, Transitional or Turbulent Flow*. Retrieved from The Engineering Toolbox:
https://www.engineeringtoolbox.com/laminar-transitional-turbulent-flow-d_577.html

Lanfrit, M. (2009). *Best practice guidelines for handling Automotive External Aerodynamics with FLUENT*. Darmstadt.

Munson, B., Okiishi, T., Huedsch, W., & Rothmayer, A. (2013). *Fluid Mechanics*. John Wiley & Sons.

Ozen, M. (2014). *Meshing Workshop*. OZEN Engineering. Retrieved from
https://www.ozeninc.com/wp-content/uploads/2014/11/MESHING_WORKSHOP_2014.pdf

Randall, C. (2018). *Mesh Types*. Swansea.

SharcNet. (2015). *Octree Method*. Retrieved from
https://www.sharcnet.ca/Software/Ansys/16.2.3/en-us/help/icm_user/iuser_octree.html

Sharcnet. (2015). *Orthogonal Quality*. Retrieved from Sharcnet:
https://www.sharcnet.ca/Software/Ansys/16.2.3/en-us/help/wb_msh/msh_orthogonal_quality.html

SharcNet. (2015). *Prism Mesh*. Retrieved from https://www.sharcnet.ca/Software/Ansys/15.0.7/en-us/help/icm_user/iuser_prismmesh.html

sharcnet. (2015). *Quick (Delaunay)*. Retrieved from sharcnet:
https://www.sharcnet.ca/Software/Ansys/15.0.7/en-us/help/icm_help/imesh_tetraquick.html

SimScale. (2017). *What is the Reynolds Number?* Retrieved from Docs:
<https://www.simscale.com/docs/content/simwiki/numerics/what-is-the-reynolds-number.html>

The Pillars Curriculum for chemical engineering. (n.d.). *Introduction To Transport Processes*.
Retrieved from http://pillars.che.pitt.edu/student/slide.cgi?course_id=10&slide_id=1.0

Tu, j., Yeoh, G., & Liu, C. (2013). *Computational Fluid dynamics*. Elsevier.

Wasserman, S. (2016). *Choosing the Right Turbulence Model for Your CFD Simulation*. Retrieved from
engineering.com:
<https://www.engineering.com/DesignSoftware/DesignSoftwareArticles/ArticleID/13743/Choosing-the-Right-Turbulence-Model-for-Your-CFD-Simulation.aspx>

Yunus, A. Ç., & Cimbala, J. M. (2010). *Fluid Mechanics: Fundamentals and Applications*. McGraw-Hill.