

University of Wales Trinity Saint David School of Engineering

CFD Ahmed Body

Advanced Computational Methods
M6X00408

Nicholas David Thomas

1702567

27/01/2020

Contents

1	Introduction	i
2	Method	i
3	Results	iii
3.1	25°	iii
3.1.1	Base	iii
3.1.2	Domain	vi
3.1.3	Global Mesh size	vii
3.1.4	Density Refinement.....	viii
3.1.5	Prism Layers	ix
3.1.6	Meshing Strategies	ix
3.1.7	Further density Refinement	x
3.2	35°	xi
4	Evaluation	xv
5	Conclusion	xvi
6	References	xviii

1 Introduction

Computational Fluid Dynamics (CFD) can be used in many fields from aerospace, electronics and motorsport. It is used to make quantitative predictions of fluid-flow using the conservation equations (mass, momentum and energy) (Hu, 2012) to calculate how any fluid travels. So, this could be in this case a body travelling in air or it could be water travelling through a valve or it could be heat travelling across a heat sink.

The aim of this report is to create a case using the ANSYS CFD package to model the flow over an Ahmed body (Simscale, n.d.), with a 25° slant angle at an airflow of 40m/s. Then this case will be developed so that it correlates with the results found in previous studies. The objective of this is to be close to as possible if not 5% of the results given. Generally, aerodynamic problems are deemed accurate if they are validated within 5%. Once this has been done a follow up case of the 35° slant will be performed to further validate the methodology.

2 Method

The CFD process can be split into three sections, pre-processing, processing and post-processing. Pre-processing can be further split into two sections, geometry and meshing. Geometry is where the domain is made. This is the working area that is used so it could be the sides of the wind tunnel. If it the situation is not a wind tunnel it also limits where the calculations are taking place as there is not much use modelling something that it is happening a mile away where the air is not getting much of an effect from the problem. This is also the stage were the shape of the object is also brought in e.g. car, wind turbine. To add to that at this point any hole in the geometry can be filled so if the model can be simplified it is at this stage it can be done. To add to that this it is also the stage that the body is removed from the domain. This essentially leaves a hole in the domain where the body is so that the air flows around that section.

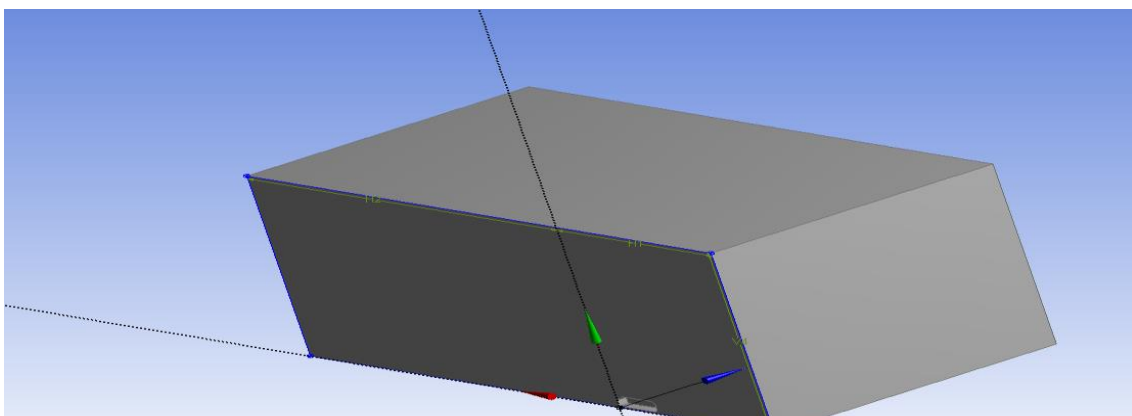


Figure 1-Example of a domain

The next stage is meshing. This is where the domain is split into smaller sections called elements. The elements are the shapes which will be where the calculations will take place. This is one of the most important steps as this will massively affect the end result. Meshing comes in a number of forms and the shape itself can be changed such as polyhedral, tetrahedra, quadrilateral and there 2D counterparts if it is a 2D problem. To add to that the meshing strategy can also be changed so top to bottom, bottom to top and variations of.

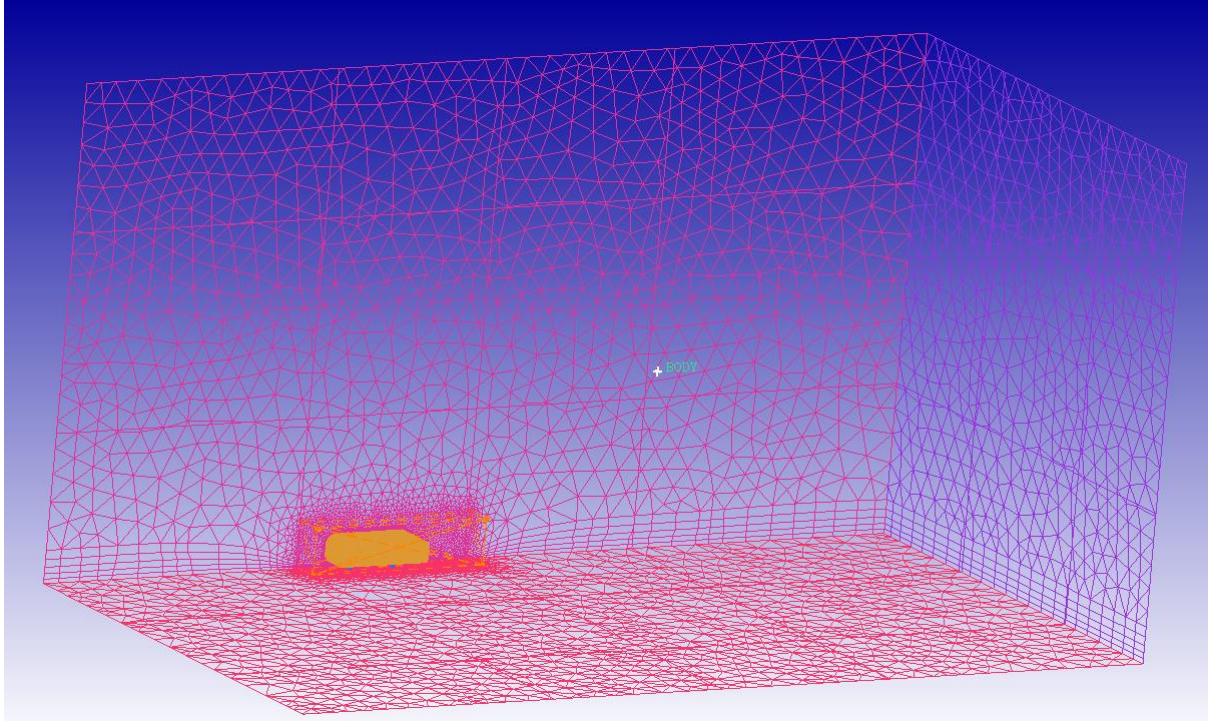


Figure 2-Example of a mesh around an Ahmed body

The next stage is solving. This is where the actual results are obtained using the Navier-Stokes Equation and a turbulence model. The Navier-stokes equation are five equations that need to be simultaneously calculated to measure the movement of fluid (NASA, 2015). This can be split into 5 equations; 3 of them calculate the momentum of the fluid in the X,Y,Z Cartesian format. One of them calculates the kinetic energy of the fluid and the other is to calculate the continuity of the system to make sure that the continuity of mass is being followed.

$$\text{Continuity: } \frac{dp}{dt} + \frac{d(\rho u)}{dx} + \frac{d(\rho v)}{dy} + \frac{d(\rho w)}{dz} = 0 \quad \text{Equation 1-Continuity Equation}$$

$$X - \text{Momentum: } \frac{d(\rho u)}{dt} + \frac{d(\rho u^2)}{dx} + \frac{d(\rho uv)}{dy} + \frac{d(\rho uw)}{dz} = -\frac{dp}{dx} + \frac{1}{Re_r} \left(\frac{d\tau_{xx}}{dx} + \frac{d\tau_{xy}}{dy} + \frac{d\tau_{xz}}{dz} \right)$$

$$\text{Equation 2-X-Momentum Equation}$$

$$Y - \text{Momentum: } \frac{d(\rho v)}{dt} + \frac{d(\rho uv)}{dx} + \frac{d(\rho v^2)}{dy} + \frac{d(\rho vw)}{dz} = -\frac{dp}{dy} + \frac{1}{Re_r} \left(\frac{d\tau_{xy}}{dx} + \frac{d\tau_{yy}}{dy} + \frac{d\tau_{yz}}{dz} \right)$$

$$\text{Equation 3-Y-Momentum Equation}$$

$$Z - \text{Momentum: } \frac{d(\rho w)}{dt} + \frac{d(\rho uw)}{dx} + \frac{d(\rho vw)}{dy} + \frac{d(\rho w^2)}{dz} = -\frac{dp}{dz} + \frac{1}{Re_r} \left(\frac{d\tau_{xz}}{dx} + \frac{d\tau_{yz}}{dy} + \frac{d\tau_{zz}}{dz} \right)$$

Equation 4-X-Momentum Equation

$$\text{Energy: } \frac{d(E_r)}{dt} + \frac{d(uE_r)}{dx} + \frac{d(vE_r)}{dy} + \frac{d(wE_r)}{dz} = -\frac{d(up)}{dx} - \frac{d(vp)}{dy} - \frac{d(wp)}{dz} - \frac{1}{Re_r Pr_r} \left(\frac{dq_x}{dx} + \frac{dq_y}{dy} + \frac{dq_z}{dz} \right) + \frac{1}{Re_r} \left(\frac{d}{dx} (u\tau_{xx} + v\tau_{xy} + w\tau_{xz}) + \frac{d}{dy} (u\tau_{xy} + v\tau_{yy} + w\tau_{yz}) + \frac{d}{dz} (u\tau_{xz} + v\tau_{yz} + w\tau_{zz}) \right) \text{(NASA, 2015)}$$

Equation 5-Kinetic Energy Equation

The computational approach also varies, one of which is Direct Numerical Simulation (DNS) which does not use a turbulence model so is computationally very expensive. Large eddy simulation (LES) solves using an averaged Navier-Stokes equation so is less computationally expensive but still too complicated. Then there is the Reynolds Averaged Navier-Stokes Simulation which is the least computationally expensive and uses a turbulence model (ANSYS, 2012) which is generally used. A turbulence model is a non-linear mathematical model (SimScale, 2018). These come in different types and each have their positives and negatives (Wasserman, 2016). This is why certain turbulence models are best suited to certain types of flow.

The next stage is post processing. This is where the results from the solver are analysed using a number of tools. It is possible to see pressure distributions, flow lines, turbulence and plots of velocity and pressure. However, there is a risk in post processing and going too far and getting data, which is not really necessary, so the user has to be careful on how much they do.

3 Results

3.1 25°

3.1.1 Base

The author first set up a basic run on the 25° which had no prism layer or density boxes. This was to see where the general area the flow was in so that they could go from there. To add to that the author started on the 25° slant as from research he found that there was generally less separation on the 25° compared to the 35° so it would be an easier case to start with (Ghaedamini, 2010). For the base run the author used the suggested settings from (Lanfrit, 2005) as a starting point. This was chosen as this paper has been used as a basis to a number of successful projects which the author has seen.

Table 1-Base Settings

Body Used	Variable	Value
25	Domain x front	5
	Domain X Rear	6
	Domain Y lower	0.01
	Domain Y Upper	6
	Domain Z	6
	Repair Geomet	
	y Toleranc	0.005
	Scale Factor	1
	Max element size	1
	Models	
	Model	K-Epsilon
	K epsilon model	resizable
	Near wall Treatment	non-Equilibrium

Run	CD	%diff from given Cd	Cl	% diff from given CL	efficienc y	%diff from given efficienc y	min orth quality	max ortho skew	max Aspect ratio
Given	0.299		0.345		1.153846				
Base 0	6.08E-02	-92.051	7.64E-01	-93.9248	12.58016	6352.987	1.95E-01	6.4862E-01	1.79E+01

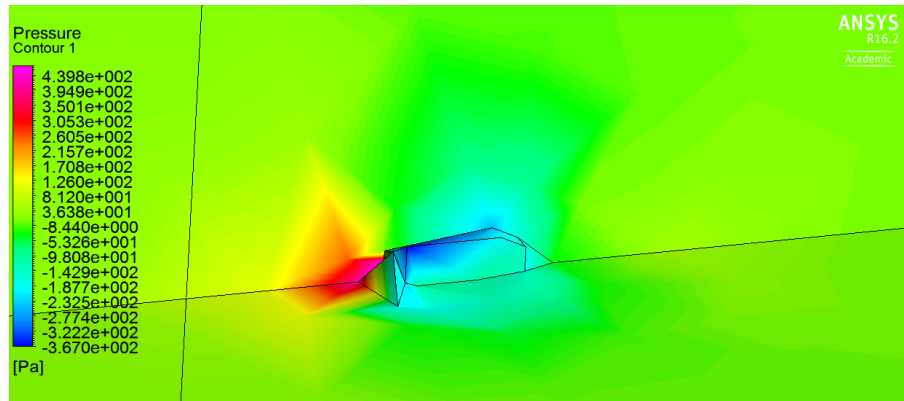


Figure 3- Initial Pressure Distribution

As can be seen from this the author gained very little information from this as the mesh was far too coarse to really gain much, as the body is relatively small compared to the mesh size. So, the author went onto adding density refinement and prism layers. Density refinement is to refine the region around the body to get a finer mesh around that section. This means that more detail can be picked up in certain areas without adding too much computing time. The prism layers are used to model turbulence near the walls of the body (Southampton University, n.d.).

Table 2-Base 2 Settings and Results

Body Used	Variable	Value
25	Domain x front	4.176
	Domain X Rear	5.22
	Domain Y lower	0.01
	Domain Y	4.99
	Domain Z	5
	Mesh	
	Scale Factor	1
	Max element size	0.25
	No. Layers	5
	Prism on	ahmed, road, feet
	Max Elemtn Size on part	ahmed-0.025 feet-0.015
	Density	0.03
	Density resize x	1.8
	density resize y	1.4
	density resize z	1.4
	desnity x offset	0.55
	Models	
	Model	K-Epsilon
	K epsilon model	resizable
	Near wall Treatment	non-Equilibrium

Run	Volume M^3	CD	%diff from given Cd	Cl	% diff from given CL	efficienc y	%diff from given efficiency	min orth quality	max ortho skew	max Aspect ratio
Given		0.299		0.345		1.153846				
Base	0	0.328530	9.87625418	0.352190	2.08405797	1.072018	-7.09179273	6.89E-07	9.9869E-01	3.59E+03

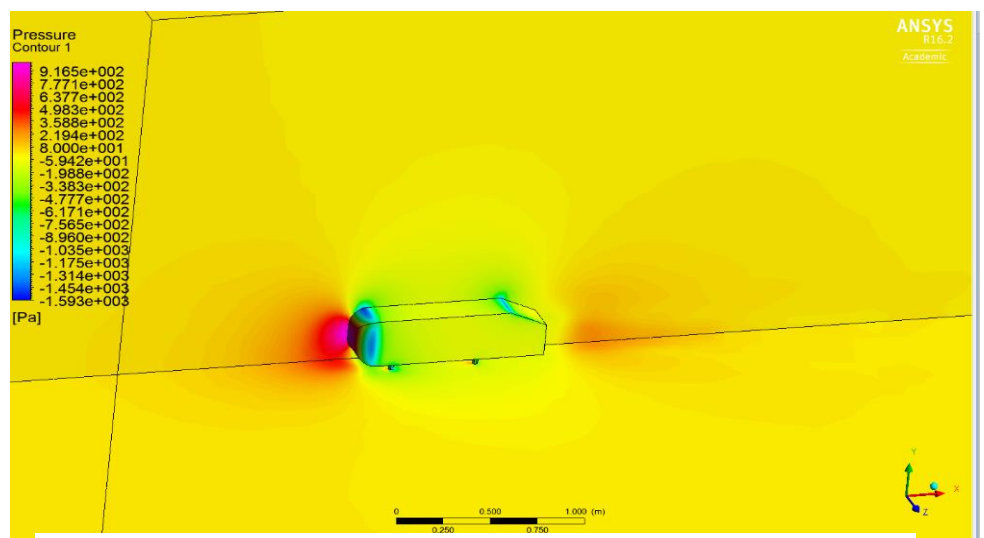


Figure 4-Base 2 Pressure Contour

This is what was expected that there is a pressure build up in front of the body. This is because as the air hits the body the air slows down therefore the pressure builds up. The opposite of this is also true around the edges and the kick point as the air speeds up and the pressure drops

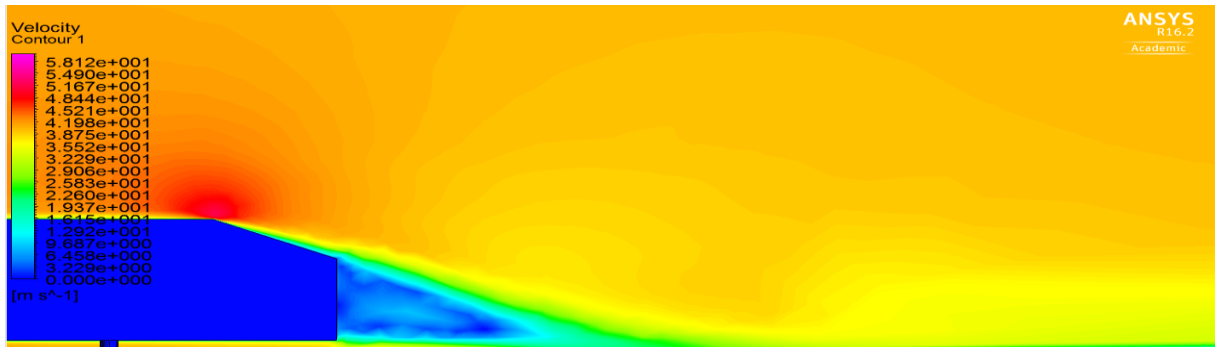


Figure 6-Base 2 Velocity at rear



Figure 5-Base 2 Pressure Contour of rear

The results also show that there is some interesting turbulent flow behind the body which may have a large effect on the results. To add to that there is also an interesting area of pressure around the kick point which will be further be refined as this also affects the flow.

3.1.2 Domain

The size of the domain is important as you want to capture all the flow but not have the domain too big as it will increase computing time. So, the author initially went through a sweep of changing the volume of the domain.

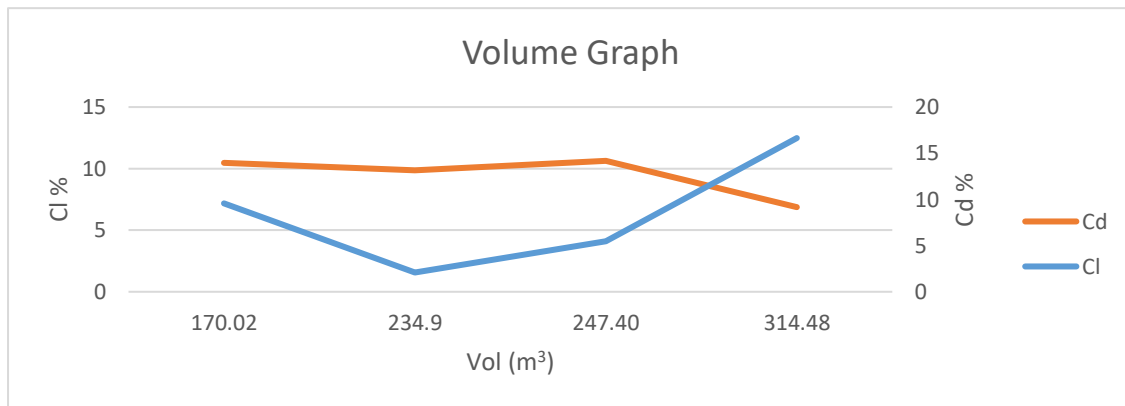


Figure 7-Volume Sweep of Domain

As can be seen the author could not find much of a trend changing volumes so they went onto changing individual dimensions. By changing each dimension, the author then managed to get the percentage difference of Cd down to 7.1% and the Cl to 3.3% difference.

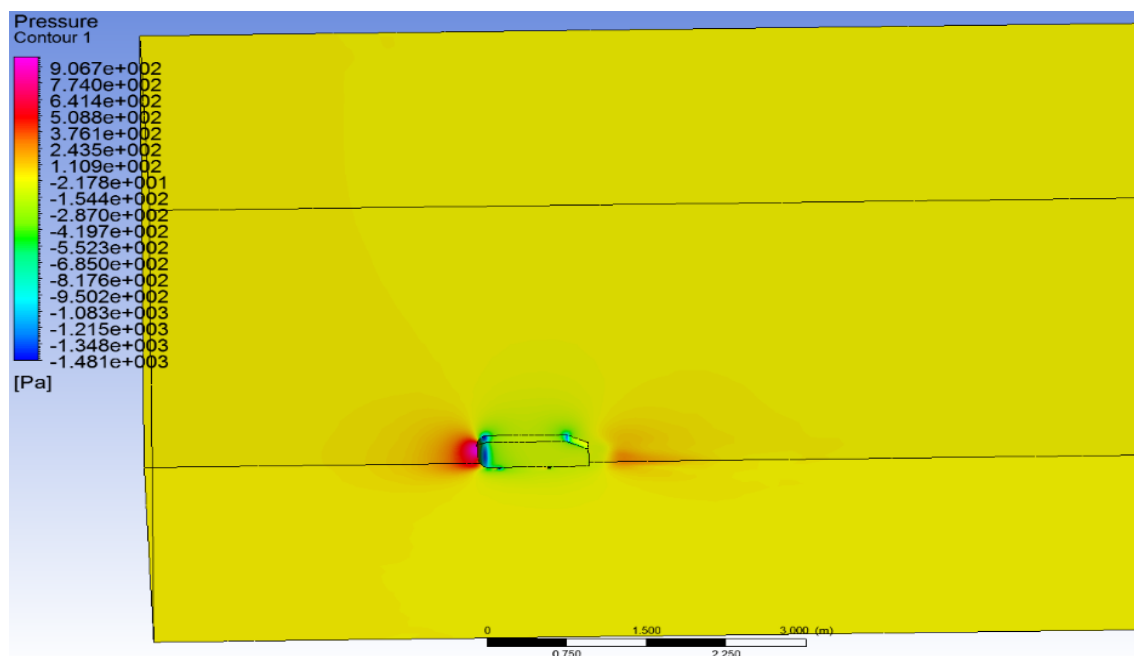


Figure 8-Pressure contour from 25 domain sweep

This shows that the domain is large enough to pick up all the details without missing anything. If there is any effect in the outer edges then the effect would be negligible.

3.1.3 Global Mesh size

The global mesh size is important as the smaller the elements the more detail will be picked up but on the other hand the smaller the more computationally expensive it is. So there needs to be a trade-off.

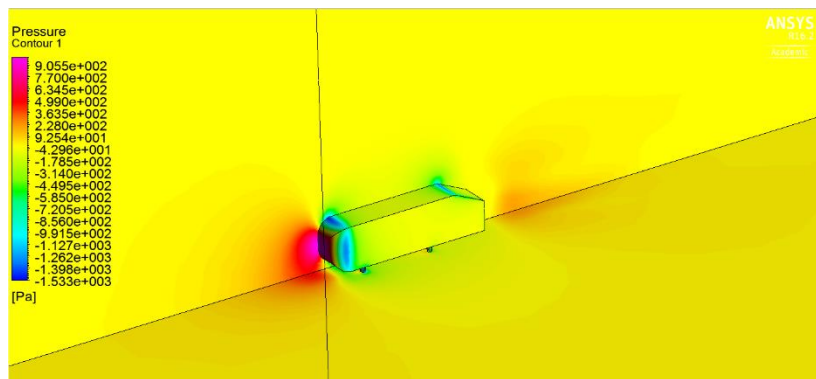


Figure 9-Best Mesh size Pressure Contour

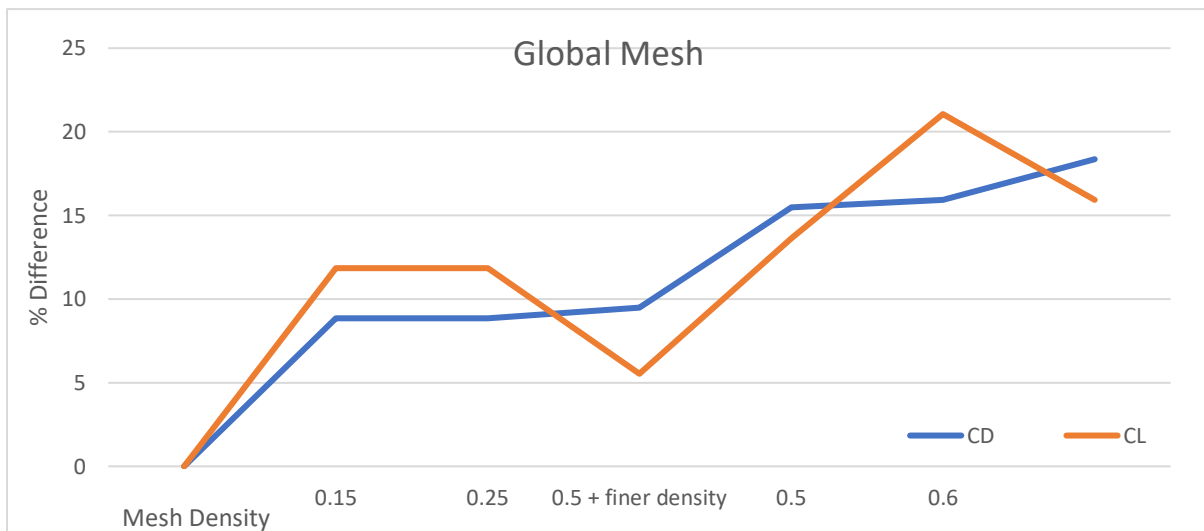


Figure 10-Global Mesh Sizer Sweep

There is a clear trend that the finer the mesh the more accurate the result. But there is also the same trend for computing time with the smaller the mesh the longer it takes. But what else this shows is that a similar global mesh can be used but with a finer refinement region can have better results (0.5+finer density). So, in the end the author deemed 0.25 as the best size as the processing time was not too long to compute but still had reasonable results.

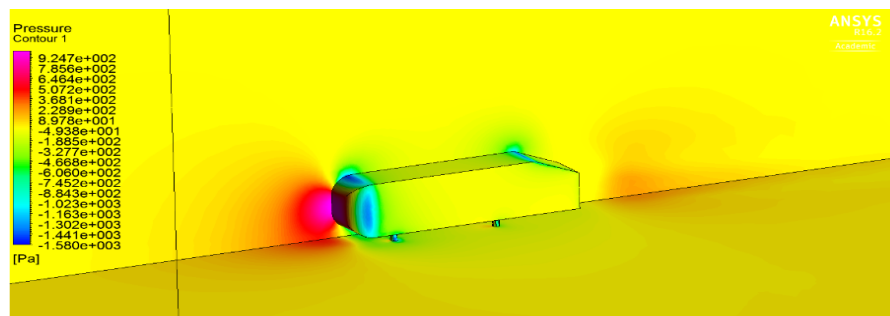


Figure 11-Smallest mesh size Pressure Contour

There is also a visible difference with much more smaller details being picked up by the smaller mesh size. However, the smaller the mesh the worse the orthogonal quality, with the minimum being $6.65\text{e-}07$ for the smallest mesh and $1.06\text{e-}05$ for the largest. The orthogonal quality is a measure of how different the shape of the element is compared to the original so effectively how much it has warped. The best orthogonal quality is 1 with it being the original shape and 0 being the worst (Davim, 2017). If the mesh it particularly bad it can go into negative values though.

3.1.4 Density Refinement

Generally, a base case is made and then the density box(es) are placed where there are key areas of interest such as turbulent areas or areas of large pressure gradients. Density refinement can also include the surface mesh size. This is the size of the mesh on individual part such as wheels or wings on the body which is being analysed. The author first started with one density box which they started by changing the volume of. From this the author came to a volume of 7.128m^3 as this picked up most of the important features around the car while it still not being too big as to not being computationally expensive. After this the author investigated the effect of the density of the density box.

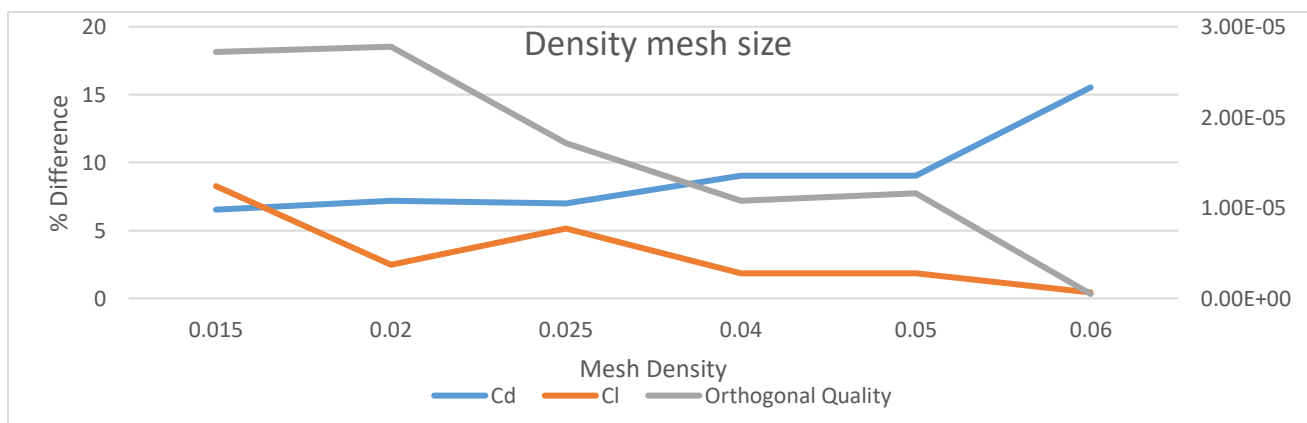


Figure 12-Density Refinement Mesh Size results

Interestingly the Cl and Cd act differently when the mesh density changes. This is averse to what the writer would expect as they expected the results to rise linearly with mesh density like the global mesh size did. This perhaps could be more to how it interacts with the rest of the mesh as it may be due to it skewing the mesh more which can be seen with the orthogonal quality because the greater the mesh the worse the orthogonal quality. This suggests that the orthogonal quality has more of an effect on the lift value than the drag value.

The author then investigated changing the surface mesh which refines the size of mesh on the surface of the body. This did follow the trend of the finer the mesh the more accurate the

results. However, it did have quite a large effect on computing power so the author opted for a compromise between the most accurate and the computing time.

The author then changed the location of the density box. This made large gains in the Cl value with it dropping from 6-7% down to 0.19% difference. However, it did not have much of an effect on the Cd. This maybe down to it picking up more flow in front of the body as the density box was initially further back so bringing it forward meant that more of the flow at the front was being picked up.

3.1.5 Prism Layers

The author then went into investigating the effect of the number of prism layers.

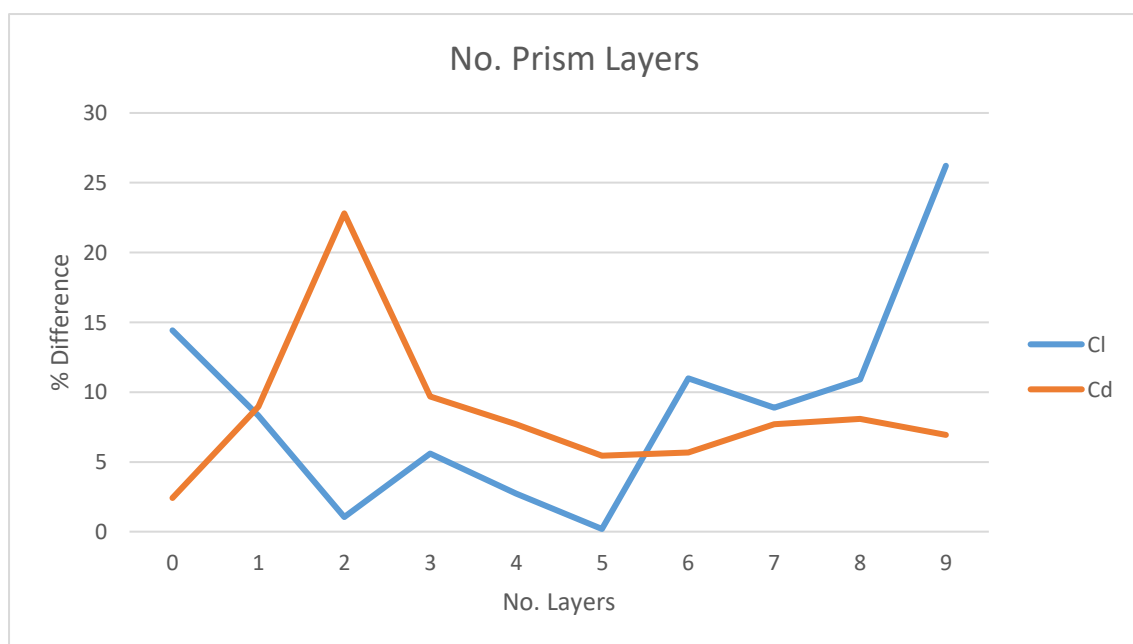


Figure 13-Number of Prism Layers Results

The author found that the accuracy increased with the number of prism layers to a point then got worse again. This maybe down to the prism layers modelling the near wall conditions up to a certain height (5 layers) then once it gets to its optimum it just starts to skew the rest of the mesh and worsen the quality. This is because the way the prism layers get put in is that it removes a layer of elements then replace them with quadrilateral cells instead in this case tetrahedral. This creates skew each time and the more layers inserted the more the cells skew.

3.1.6 Meshing Strategies

The writer then did a sweep of meshing strategies. They got running 6 different cases of meshing strategies. The first was Quick (Delaunay) Standard. This is a bottom up method which stems from a surface mesh but this resulted in the Cd to become 26% difference. The author had better luck with the TGlub version. This uses a similar method of bottom up but it

uses a more gradual transition rate near the surface of the body and a faster transition rate further out from the body. This means that the further away from the body the larger the element gets (Leap CFD team, 2012). This is quite good as it means the elements are smaller nearer the body where more detail is needed. However, this still did not offer any gain from the robust octree being used in the cases before. The author looked into polyhedral mesh which brought down the accuracy but halved the number of cells so the author doubled the number of elements and made them polyhedral which was quite close to the answer with Cl being 7% different and Cd being -0.85%. This took a long time to do as you have to essentially mesh the body twice and there was no gain.

The author then looked into turbulence models but because there was very little separation it did not make a difference and if it did it was a negative one.

3.1.7 Further density Refinement

At this point the values gained was a difference of Cd - 5.4% and Cl 0.55% so the author did further density refinement to get closer to the result. The author did this by putting an extra density box around the slant.

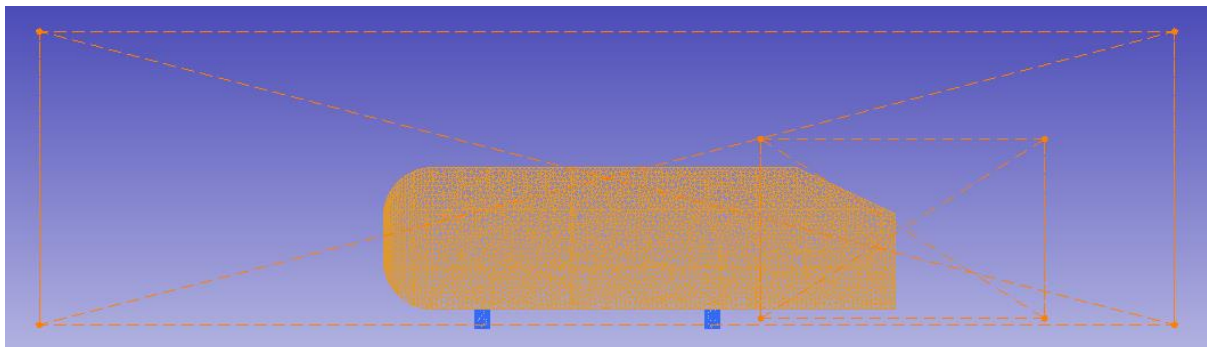


Figure 15-Secondary Refinement Box

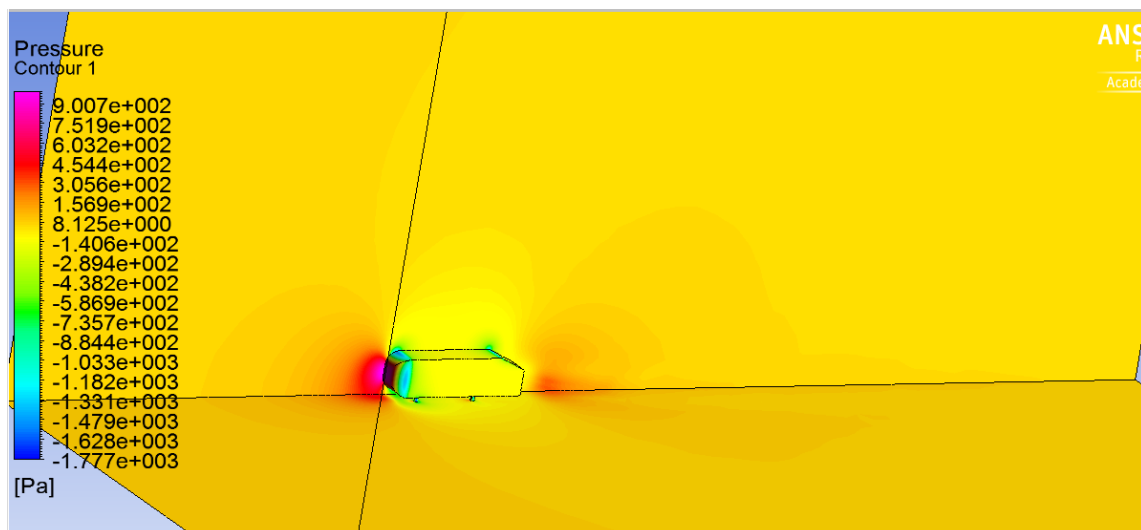


Figure 14-Pressure Contour with the added Second Refinement Box

This made a bit of a difference by taking CI to -0.66% and Cd to 4%. This is better as they are now both within the 5% tolerance.

3.2 35°

The author then went onto using the best settings with 1 density box from the 25° on the 35°. But the results were completely different (Cd-6.9% and the CI-4840.3%). This is due to the separation that occurs with the larger slant.

Run	CD	%diff from given Cd	CI	% diff from given CL	efficiency
Given	0.279000		0.004		0.01433692
X dimension					
Domain 1	0.323280	15.87096774	0.197840	4846	0.61197723
Domain 2	0.342760	22.85304659	0.284540	7013.5	0.83014354
Domain 3	0.318880	14.29390681	0.208240	5106	0.65303562
Domain 4	0.322190	15.48028674	0.231020	5675.5	0.71703032
Domain 5	0.328740	17.82795699	0.221100	5427.5	0.67256799
Y Dimension					
Domain 6	0.327300	17.31182796	0.223490	5487.25	0.68282921
Domain 7	0.333800	19.64157706	0.256980	6324.5	0.76986219
Domain 8	0.341440	22.37992832	0.220290	5407.25	0.64517924
Z Dimension					
Domain 9	0.332020	19.00358423	0.266620	6565.5	0.80302391
Domain 10	0.326150	16.89964158	0.234420	5760.5	0.71874904
Domain 11	0.310830	11.40860215	0.160220	3905.5	0.51545861
Domain 12	0.348770	25.00716846	0.353300	8732.5	1.0129885
Domain 13	0.342820	22.87455197	0.296400	7310	0.86459366

Figure 16-Domain sweep for 35

The author then put a secondary density box in. This brought the results slightly closer (Cd- 2.52% CI-1870.9%). But as can be seen the 35 degree has significant separation. As the wake was so much longer on the 35 the author then made another domain sweep.

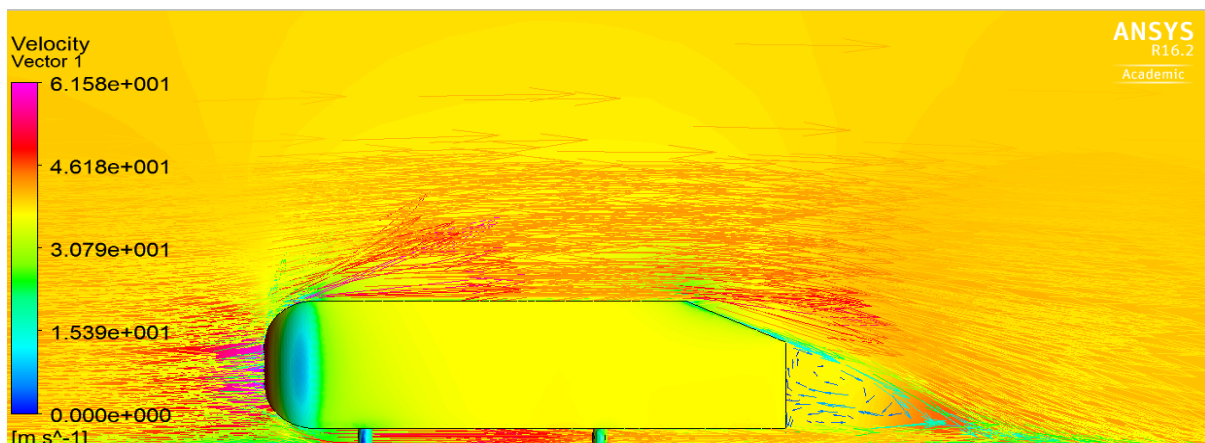


Figure 17-Velocity stream Lines 25

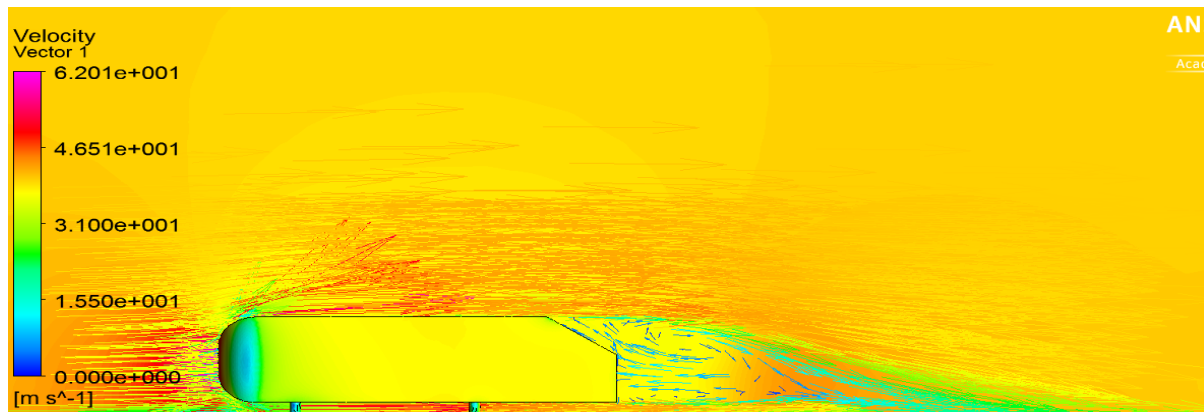


Figure 18-Velocity Stream Lines 35

This shown that the domain needed to be bigger than the 25° as expected. With the best being domain 11 which had dimensions X-12.516m Y-8.35m Z-6.044m.

Then the writer went onto the prism layers as they thought that there could be benefits, as separation is occurring therefore prism layer could be used to help model the near wall separation.

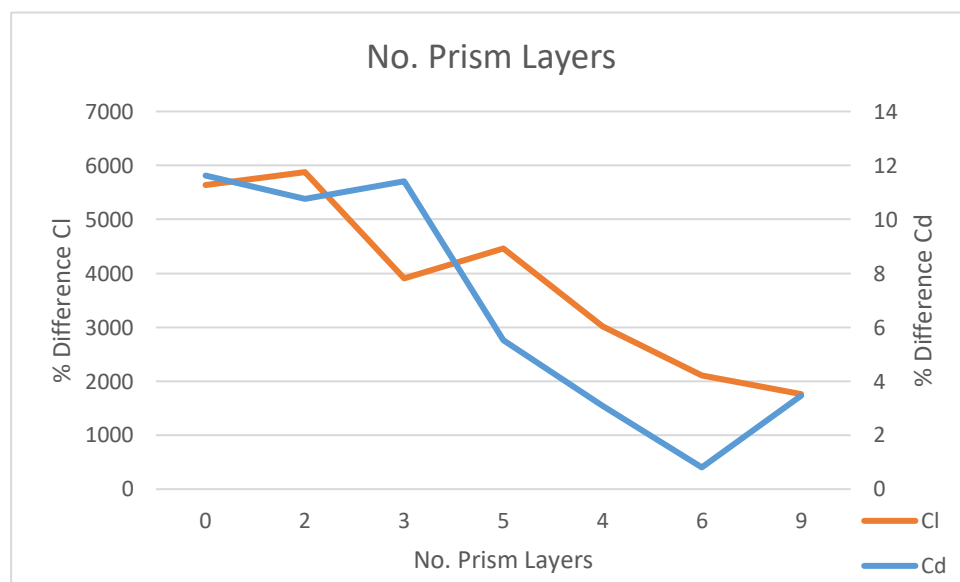


Figure 19-Prism Layer Sweep 35

This shows as the number of prism layers increase so does the accuracy to a point. This will be because it is able to model the boundary layer which is much larger when the air is turbulent (Epifanov, 2011). However as mentioned earlier the larger the number the layers the higher potential for skewed elements.

The author then looked through a number of turbulence models.

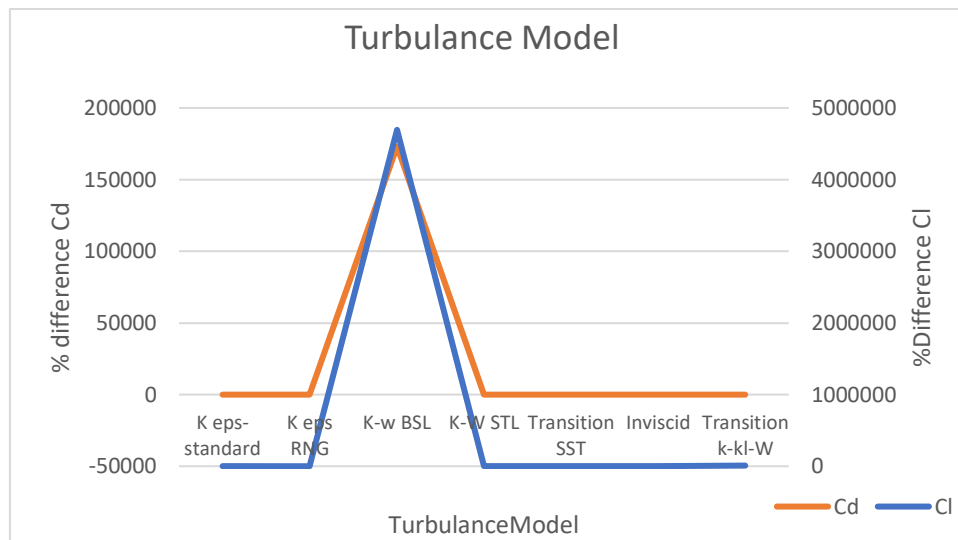


Figure 20-Turbulence Model Sweep 35

As can be seen some are significantly better than others but author found that the closest was K-Epsilon RNG (CI-223.425% Cd—8.548%). This may be because the RNG model enhances the swirling motion in turbulence some may take into the swirl behind the model better than the others (ANSYS, 2009).

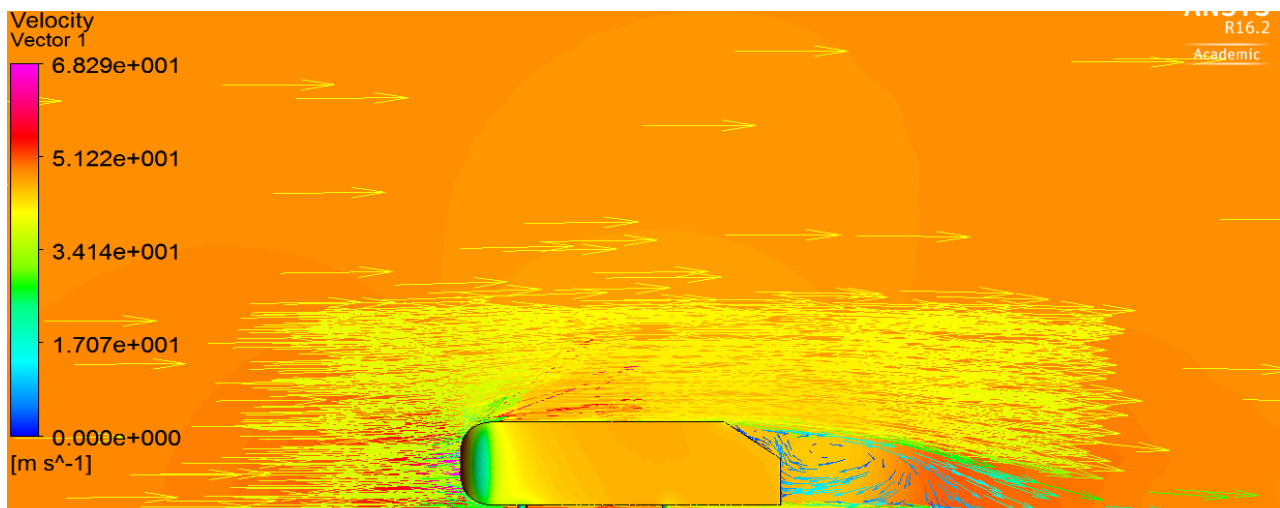


Figure 21-Velocity Stream Line with K-E RNG model

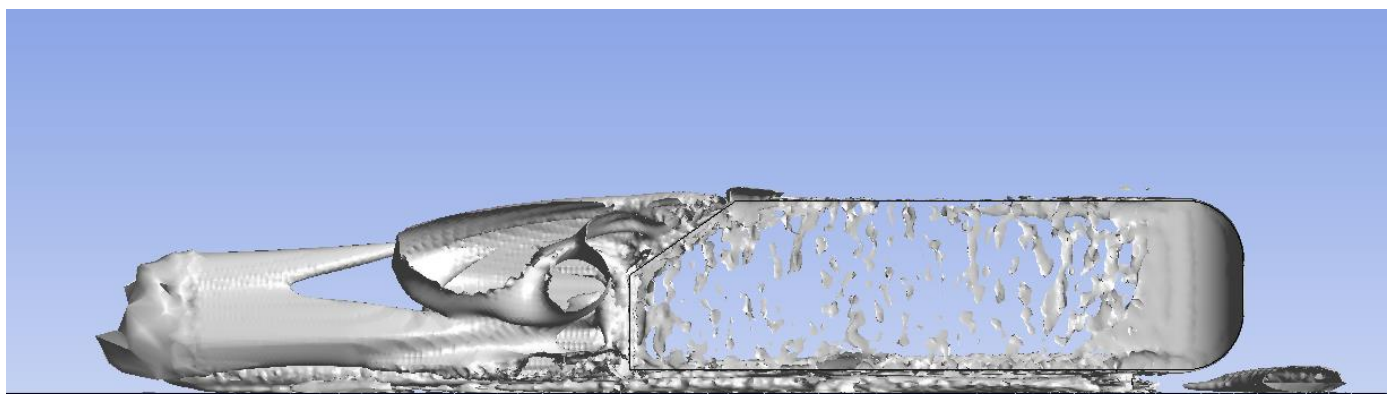


Figure 22-Vortices around 35

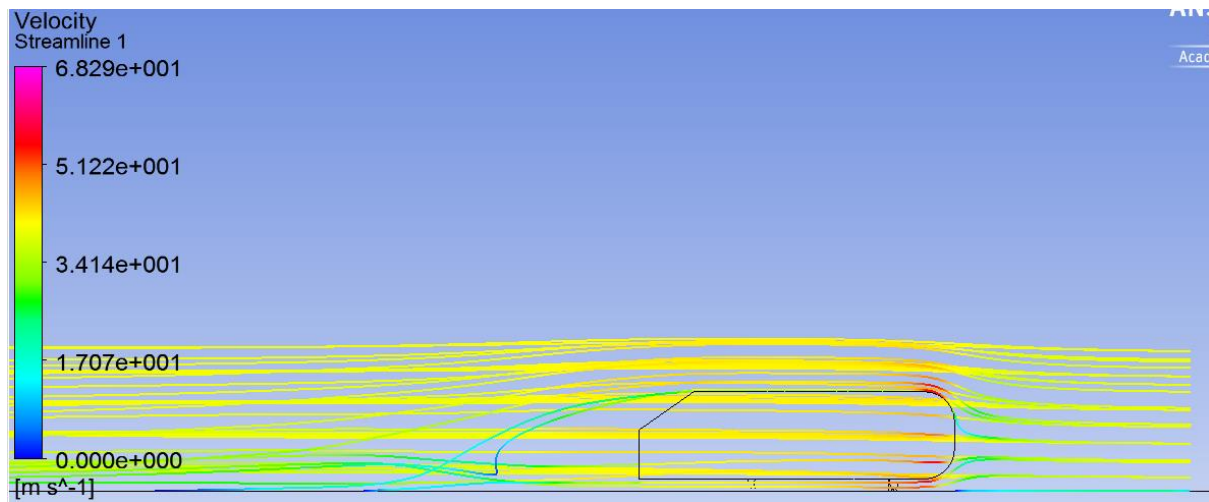


Figure 23-Stream Lines 35 Side View

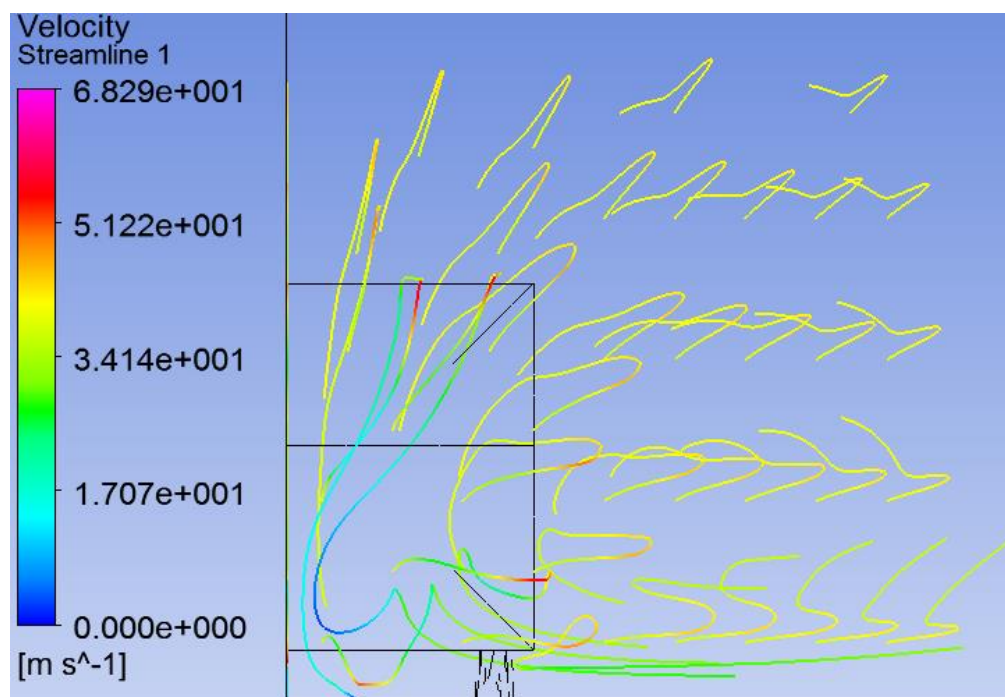


Figure 24-Stream Lines 35 Front View

4 Evaluation

The author if they were to do this study again would do some things differently. One thing would be changing individual dimensions on the domain on 25° study. It was done on the 35° but on the 25° it was a bit aimless so changing each individual part makes sure that the domain is at its minimum size. Another thing the author would do differently is on the density refinement possibly investigate increase the mesh in certain parts to see if the computation time could be reduced whilst keeping the same level of accuracy. The author would also would like to get the 35° slant closer to the value given. However, this was not possible at the time further density refinement possibly on the top or under the body could be investigated to see if that is what is affecting the results. This can be seen where there is a large vortex under the body. However, as the value of CI is so low is could be counted as 0 which mean getting the percent difference very difficult.

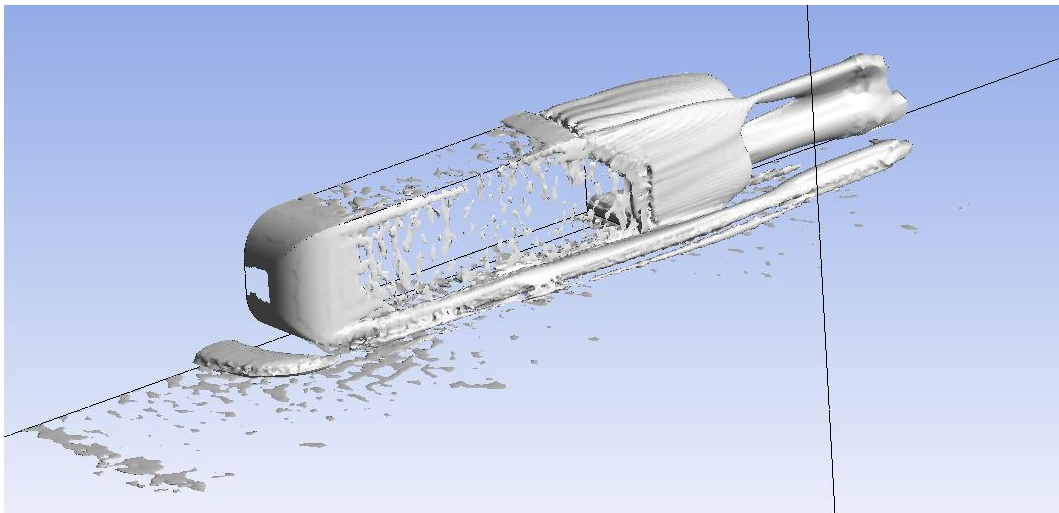


Figure 25-Vortices 35

5 Conclusion

In conclusion the best settings are different on a case by case basis. For 25° the author found that these settings were the best but admits that there is still room for improvement discussed in the evaluation section.

Table 3-Best Settings 25

Body Used	Variable	Value			
25	Domain x front	4.176	Max Elemtn Size on part	ahmed-0.035 feet-0.025	
	Domain X Rear	5.22	Density 1	0.015	
	Domain Y lower	0.01	Density 1 resize x	2.2	
	Domain Y Upper	4.99	density 1resize y	1.8	
	Domain Z	5	density 1resize z	1.8	
	Meshing		desnity 1 x offset	0.55	
	Scale Factor	1	Density 2	0.01	
	Max element size	0.25	Density 1 resize x	0.55	
	No. Layers	5	Solving		
	Prism on	ahmed, road, feet	Model	K-Epsilon	
			K epsilon model	reslizabl e	
			Near wall Treatem ent	non-Equilibri um	

Here are the settings the author used which was closest to the results for the 35°.

Table 4-Closest Settings 35

Body Used	Variable	Value			
35	Domain x front	4.176	Max Elemtn Size on part	ahmed-0.035 feet-0.025	
	Domain X Rear	8.34	Density	0.015	
	Domain Y lower	0.01	Density resize x	2.2	
	Domain Y Upper	4.99	density resize y	1.8	
	Domain Z	6.044	density resize z	1.8	
	Repair Geometr y		desnity x offset	0.55	
	Tolerenc e	0.005	Density 2	0.01	
	Scale Factor	1	Density 2 resize x	0.55	
	Max element size	0.25	Density 2 offset X	0.3	
	Edge Criterion	0.2	Density 2 Offset y	0.02	
	No. Layers	9	Models		
	Prism on	ahmed, road, feet	Model	K-Epsilon	
			K epsilon model	RNG	
			Near wall Treatem ent	non-Equilibri um	

These are the closest the author found but they believe that there could be more gains through density refinement. Perhaps by extending it further rearwards it maybe be able to pick up more of the turbulent area. To add to that perhaps more refinement underneath the body as this shows sines of flow underneath which has a high vorticity so may have an effect on the result. This study was made sequentially but each part may have an effect on the other so perhaps it could be a benefit to do the study iteratively instead

6 References

ANSYS, 2012. *Introduction to ANSYS FLUENT*. s.l.:s.n.

ANSYS, I., 2009. *RNG K- ϵ Model*. [Online]

Available at: <https://www.afs.enea.it/project/neptunius/docs/fluent/html/th/node59.htm>

[Accessed 24 January 2020].

Davim, J. P., 2017. *Computational Methods and Production Engineering Research and Development*. s.l.:s.n.

Epifanov, V. M., 2011. *Thermopedia*. [Online]

Available at: <http://www.thermopedia.com/content/595/>

[Accessed 20 January 2020].

Ghaedamini, H., 2010. *CFD analysis of Ahmed reference body by several turbulent models*, s.l.: s.n.

Hu, H. H., 2012. *Fluid Mechanics*. 5th ed. s.l.:Elsevier.

Lanfrit, M., 2005. *Best practice guidelines for handling Automotive External Aerodynamics with FLUENT*. 1.2 ed. Darmstadt: s.n.

Leap CFD team, 2012. *Tips & Trcks: INFLATION LAYER MESHING IN ANSYS*. [Online]

Available at: <https://www.computationalfluidynamics.com.au/tips-tricks-inflation-layer-meshing-in-ansys/>

[Accessed 17 January 2020].

NASA, 2015. *NASA Navier-Stoke Equation*. [Online]

Available at: <https://www.grc.nasa.gov/www/k-12/airplane/nseqs.html>

[Accessed 14 November 2019].

SimScale, 2018. *simscale.com*. [Online]

Available at: <https://www.simscale.com/docs/content/simwiki/cfd/whatis CFD.html>

[Accessed 09 January 2020].

Simscale, n.d. *Simcale.com*. [Online]

Available at:

<https://www.simscale.com/docs/content/validation/AhmedBody/AhmedBody.html>

[Accessed 08 January 2020].

Southampton University, n.d. *southampton.ac.uk*. [Online]

Available at: <http://www.southampton.ac.uk/~zxie/SESS6021/CCMP/online1/072->

[workingWithPrismMesh-02.html](#)

[Accessed 17 January 2020].

Wasserman, S., 2016. *Choosing the Right Turbulence Model for Your CFD Simulation*.

[Online]

Available at:

<https://www.engineering.com/DesignSoftware/DesignSoftwareArticles/ArticleID/13743/Choosing-the-Right-Turbulence-Model-for-Your-CFD-Simulation.aspx>

[Accessed 16 January 2020].