

Ahmed Assignment

by George Wright

FILE	11946_GEORGE_WRIGHT_AHMED_ASSIGNMENT_176684_1832380365.DOCX (5.46M)		
TIME SUBMITTED	29-APR-2019 08:48AM (UTC+0100)	WORD COUNT	4094
SUBMISSION ID	105626919	CHARACTER COUNT	21043



Prifysgol Cymru
Y Drindod Dewi Sant
University of Wales
Trinity Saint David

University of Wales Trinity Saint David

School of Engineering

Computational Fluid Dynamics: Ahmed
Body Analysis (M7X01380)

George Wright

Student ID: 1804298

17th April 2019

Table of Contents

1	Introduction.....	1
2	Literature Review.....	2
2.1	Computational Fluid Dynamics (CFD).....	2
2.2	The Ahmed body.....	3
3	Creating a Simulation Model	4
3.1	Importing and creating a Fluid Domain.....	4
3.2	Creating a Mesh	7
3.3	Solving and Analysing Results	8
4	Investigating Simulation Parameters	14
4.1	Domain Sizes	14
4.2	Prism Layers	15
4.3	Mesh Strategies.....	23
4.4	Turbulence Models	30
4.5	Overall Results.....	31
5	Conclusions.....	32
6	References.....	33

1 Introduction

This assignment's main aim was to generate a 3D simulation for a 25° slant Ahmed body travelling straight ahead at 40ms^{-1} using ANSYS Workbench 16.0 and Fluent computational fluid dynamics (CFD) solver. During this process a fluid domain geometry was created along with various meshing strategies and boundary conditions to simulate the Ahmed. The simulation would then be analysed to confirm results against experimental data, particularly for drag and lift coefficients, investigating CFD methodologies in the process. Several simulations were performed to study the effects of the domain size, number of prism layers, mesh strategy, solver settings and turbulent models on the Ahmed external aerodynamics along with drag and lift to obtain accurate results.

2 Literature Review

2.1 Computational Fluid Dynamics (CFD)

CFD can calculate fluid flow, heat and/or mass transfer, chemical reactions and other parameters using computer science. It is highly beneficial in exploring innovative ideas, developing products, optimisation, troubleshooting and re-designing. Additionally, CFD is an alternative way to carrying out experiments and collecting data making it advantageous in saving money, time and effort (ANSYS 2015). Linfield and Mudry (2008) state further advantages in that once a CFD model is produced it can be used numerous times and operated simultaneously on other machines whereas using a physical model can only be tested in one stage.

However, Linfield and Mudry (2008) argue disadvantages of CFD that it is mainly simulated in a steady state and so it is unable to forecast re-entrainment of particles in contrast to physical models. Additionally, CFD relies on computer storage limiting the number of elements implemented into a model and so producing inaccurate results that don't agree with physical models. CFD was used in this assignment as it can produce fluid flow around the Ahmed travelling straight ahead at 40ms^{-1} showing pressure and velocity distributions including calculations for drag and lift coefficients without the use of experiments. Figure 1 below shows the CFD process followed in this assignment:

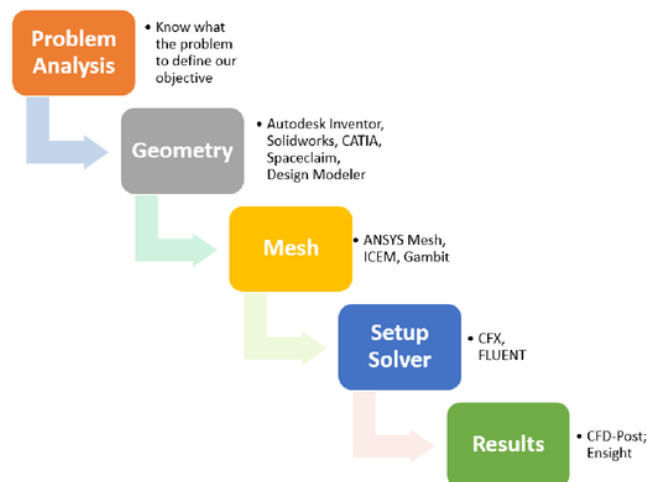


Figure 1: CFD Process (Process of CFD 2019).

This CFD process was adopted in this assignment using ANSYS Workbench 16.0 and Fluent. CFD uses discretization by creating a domain in geometry and then implementing a meshing strategy leaving a finite set of volume elements. Each volume element has energy conservation equations

applied for mass, momentum and energy generating solutions. Discretization is used on these non-linear differential equations into a structure of algebraic equations to which each are solved analytically.

The differential equations are very complex and are only solvable via human calculations for simple geometries (Linfield and Mudry 2008). Hence, CFD is a suitable tool to use in calculating these analytical equations. In addition, four viscous models (within Fluent) were used in this assignment; k-epsilon realizable, k-epsilon standard, k-omega standard and omega SST (Thabet and Thabit 2018). These turbulent models were essential to use in the Fluent to predict the turbulent flows around the Ahmed; as the eddy currents dissipate and get smaller than the mesh elements the turbulent model chosen calculates what energy it will have (Wu 2011).

2.2 The Ahmed body

The Ahmed body is used in this assignment to generate a simulation which is commonly employed in the vehicle industry as it represents a simplistic car geometry and confirms the external aerodynamics it imposes; particularly in the wake region (Thabet and Thabit 2018). It was created by Ahmed et al (1984) to study the effect of the rear slant angle on drag and the figure below shows the Ahmed used and provided in this assignment with dimensions:

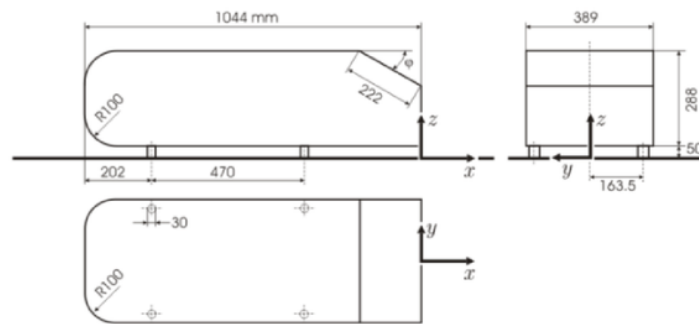


Figure 2: Ahmed Body Geometry (Ahmed et al 1984).

θ is called the critical angle of the Ahmed as changing this significantly affects the air flow structure at the wake along with drag and lift particularly at $\theta = 30^\circ$ where flow separation occurs (Lienhart, Stoots and Becker 2019). Through optimising θ the vehicle industry can reduce drag of a car saving fuel, reducing CO₂ emissions, improving stability and decreasing noise (Meile et al 2011). Furthermore, reducing lift is also optimised to enhance a vehicles performance around corners by applying the tyres to normal load of the car (Toet 2019).

In this assignment θ is analysed at 25° and table 1 below shows experimental data provided in a wind-tunnel for this geometry travelling at 40ms^{-1} :

Slant Angle (φ)	Lift Coefficient	Drag Coefficient
25	0.345	0.299

Table 1: Ahmed Body Experimental Data.

Moreover, figure 3 (provided in this assignment) shows the aerodynamic flow at the back of the Ahmed for a slant angle θ of 25-degrees:

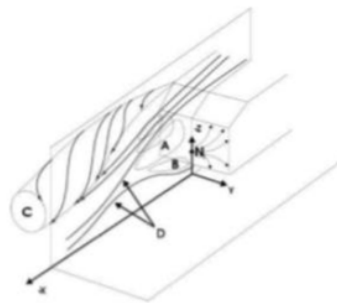


Figure 3: Ahmed 25° Slant Flow Structure in the wake.

The figure illustrates various flow structures for the Ahmed 25° slant; regions A and B represent the upper and lower recirculating areas with the former being the largest; C is a longitudinal vortex called a C-pillar; and D shows flow separation from the Ahmed (Rao et al 2018). The main aim of this assignment was to create an Ahmed 25° slant simulation that would agree with the experimental results in table 1 and visual results shown in figure 3.

3 Creating a Simulation Model

3.1 Importing and creating a Fluid Domain

The first stage in the CFD process was to import the 25° slant Ahmed body into ANSYS 'geometry' with θ equalling 25-degrees. It was important to ensure the Ahmed was confined to a volume called the domain which would be created around the body; this domain would be filled with elements. Prior to creating this domain, several assumptions were made:

1. Fluid domain will act as a wind tunnel
2. Floor is static
3. Ahmed body is symmetrical down the centre; in the x-y plane

The advantage of point three is that the domain can be halved to a size which the simulation needs to be and therefore halving the number of elements; fewer elements will produce a quicker simulation. The initial domain had dimensions specified as $x = 6\text{m}$ (-2m in front and 4m in back relative to the centre of the Ahmed), $Y=1\text{m}$, and $z = 1\text{m}$. This domain can be seen in figure 4:

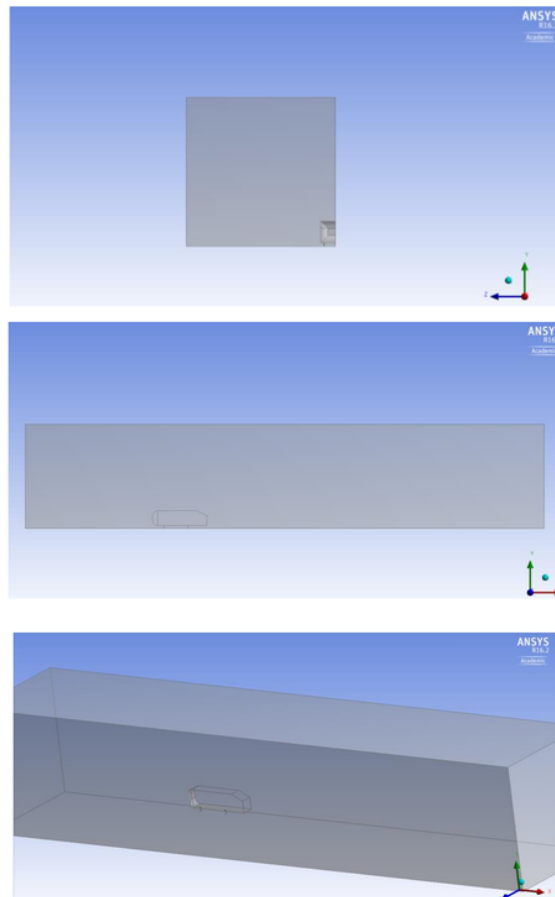


Figure 4: Domain box around the Ahmed body.

These dimensions were chosen as it is anticipated to see laminar flow approaching the Ahmed i.e. low Reynolds number; -2m in the x -axis ensuring there is enough volume within this region that the flow does not deflect from the Ahmed to the inlet and back again. In contrast, as figure 3 shows in regions D and C, generated from the Ahmed's slope and edges it is expected that there will be separation and a C-pillar from the back of the Ahmed to the domain outlet hence the larger $+4\text{m}$ x -axis to capture all this detail. In addition, we expect to see further vortices from the legs of the Ahmed. Regions A and B (in figure 3) are expected to be visible in the simulation showing two recirculation's in the wake with the former region bigger than the latter.

In addition, as the flow contacts the Ahmed in the initial stages it's expected that the velocity will decrease as it goes towards the Ahmed as it will act as a wall exerting a force i.e. no flow can pass through the Ahmed body and so leaving a high pressure. A further assumption to state at this stage is that the flow in this simulation is incompressible i.e. the flow's density is constant and uniform. Thus, the equation of continuity applies stating that the volume flow per second travelling through the domain must equal to the volume flow out of the domain; this is the same case from the front to back of the Ahmed. Therefore, it is predicted that as the cross-sectional area is decreased when the flow goes over the top and bottom of the Ahmed the velocity will increase. Furthermore, as the flow does not change in elevation, Bernoulli's equation leads to acceptance that the pressure will decrease at the top and bottom of the Ahmed due to increase in velocity (Halliday, Resnick and Walker 2010). Finally, it is anticipated that the airflow will gradually normalise to initial velocity and pressure values as it approaches the domain outlet.

The last set-up was to ensure each face of the domain and Ahmed were broken down by either single face or grouped together and renamed. These faces were to be used for specifying specific boundary conditions later and were named as the following; topside, floor, inlet, outlet, symmetry and Ahmed and Ahmed back as shown in the below figure:

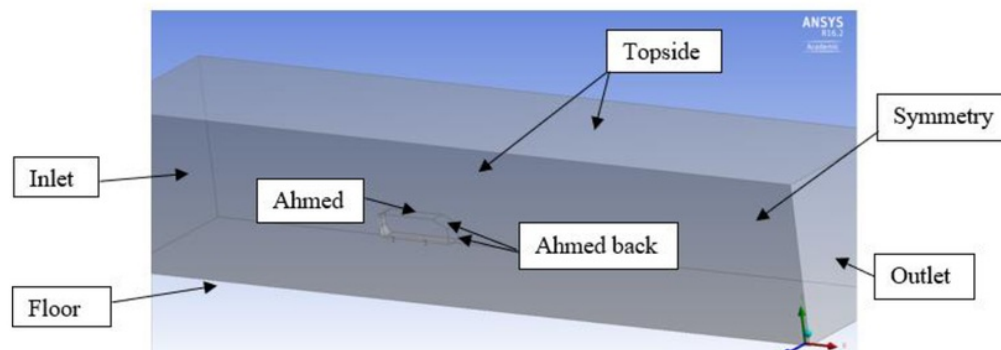


Figure 5: Simulation faces.

The sketch was saved in ANSYS and linked to ICEM which would upload the domain and Ahmed to allow a mesh to be created.

3.2 Creating a Mesh

The sketch created in 3.1 was checked for quality ensuring that all edges of the domain and Ahmed were well-defined and joined together; all edges showed red lines highlighting good quality. Before creating a mesh, ICEM was used to state where elements were to be filled and was done so with a body created in the domain centre.

Thereafter, it was important to specify the surface shell mesh parameters type and method path as triangles and path independent respectively; this will populate surface elements as triangles. In addition, a volume global mesh type Tetra/Mixed with method robust octree was selected as this is beneficial with poor quality geometries i.e. the Ahmed body. However, octree has its disadvantage that it creates a structural mesh that is not suitable for the fluent solver but still works.

The first simulation used a global volume mesh element size of 0.2m and a density box to capture the whole of the Ahmed and approximately 0.5m away from the back body with element size of 0.05m. This can be seen in figure 6 below:

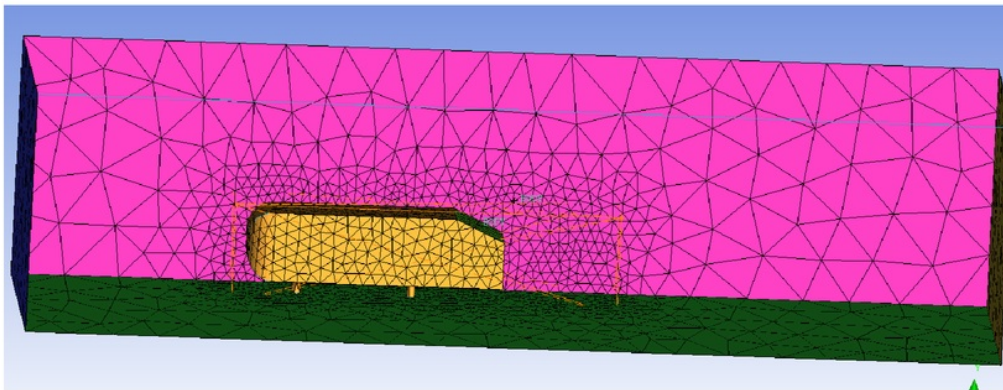


Figure 6: Simulation 1 mesh.

The density box was created around the Ahmed to include a detailed analysis of the wake region as discussed in 3.1 including obtaining accurate results for drag and lift. As the mesh was now populated at this stage a simulation would be possible to commence using Fluent.

3.3 Solving and Analysing Results

The mesh generated in 3.2 had to be set-up within the Fluent Solver where the viscous models were applied; K-epsilon realisable (standard wall function) was selected for the initial runs. Boundary conditions with faces created in 3.1 were stated in Fluent as the following; the Ahmed, Ahmed back and floor faces were 'walls'; the inlet face was a 'velocity inlet' with a x-component value of 40 ms^{-1} ; the outlet face was a 'pressure outlet' with a value 0 Pascal; and the symmetry and topside faces were 'symmetry'. The fluid in the model was checked to ensure it was air and that the body was the fluid.

Further set-up included the computation from the inlet face checking if the correct reference value for the velocity inlet was 40 ms^{-1} and specifying the Ahmed frontal area of 0.058 m^2 . Solution method 'coupled' was chosen as this calculates velocity and pressure simultaneously checking values one against the other giving a quicker convergence. After a hybrid initial solution was carried out for initial predictions the simulation was ready to start allowing 200 iterations if needed. Figure 7 below shows fluent solving the mesh:

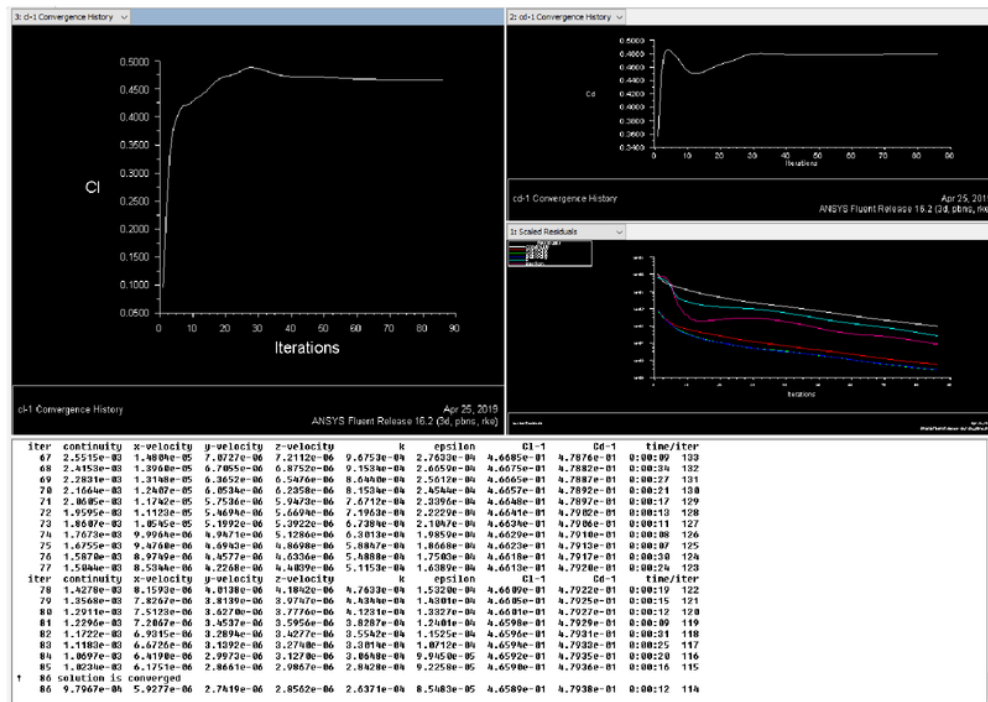
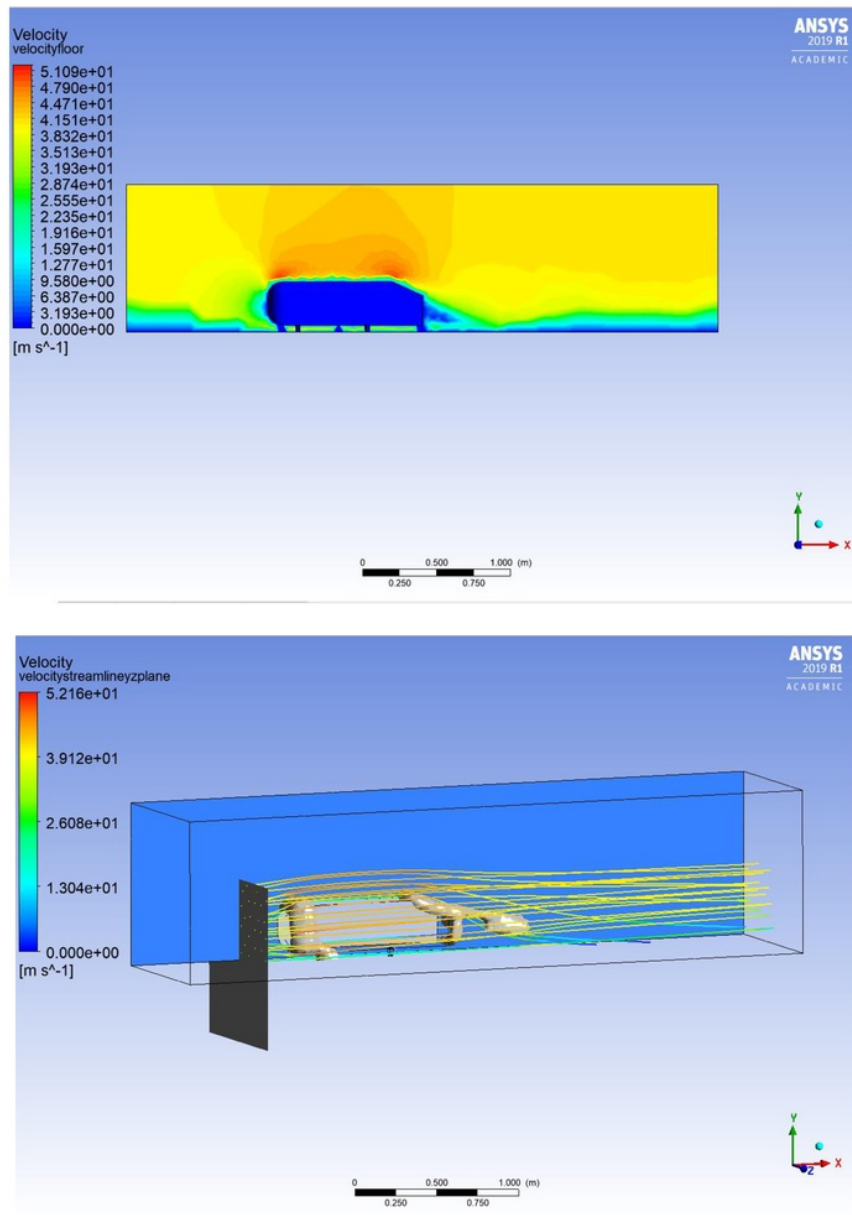
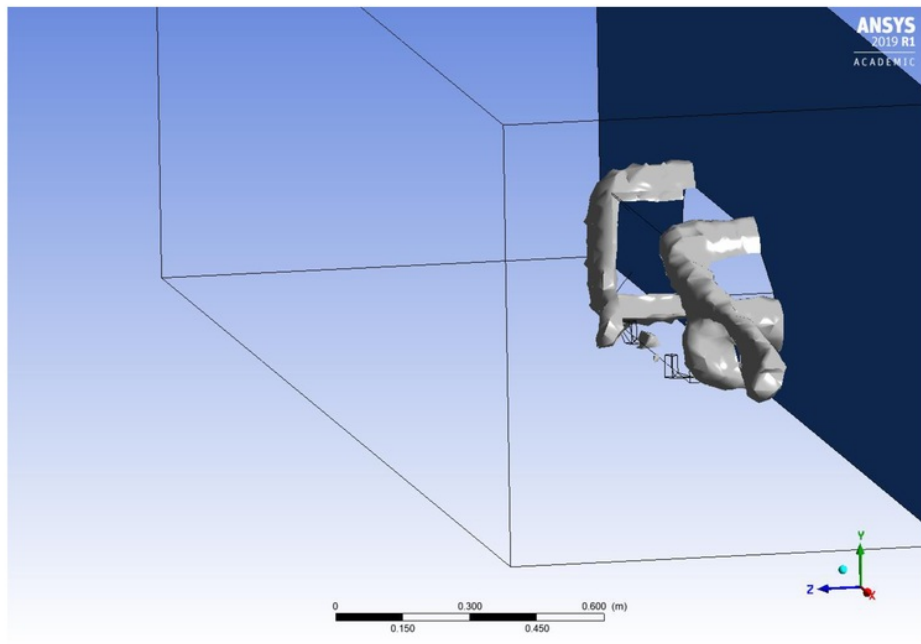


Figure 7: Simulation 1 Fluent results.

Referring to figure 7 simulation 1 converged quickly on the 86th iteration giving a 0.466 lift and 0.479 drag coefficient(s). Comparing this with experimental results in table 1 both the lift and drag are over predicted using this domain/ mesh/solver by 0.121 and 0.18 respectively. An analysis of the solution

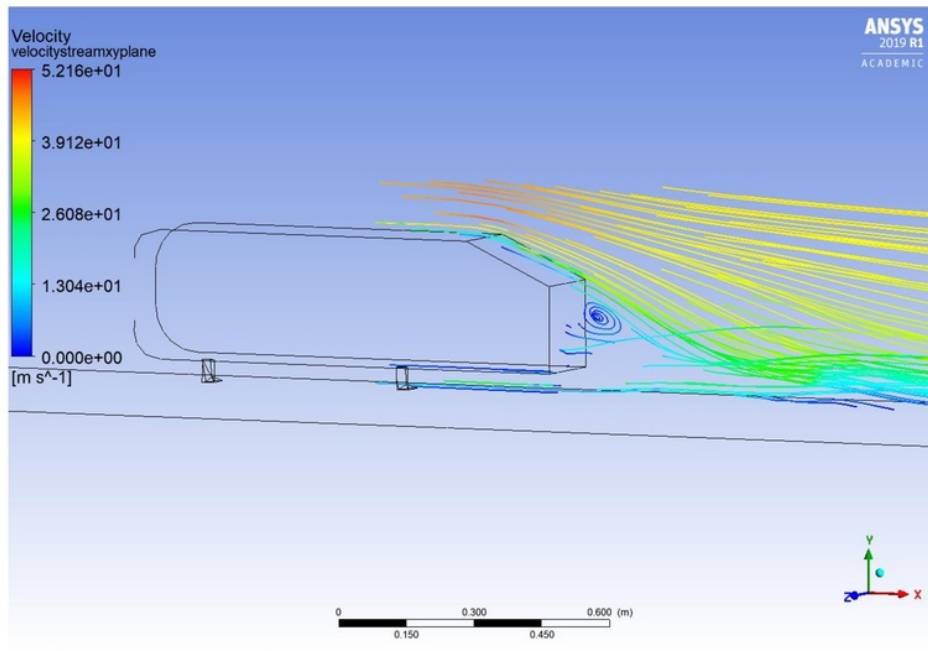
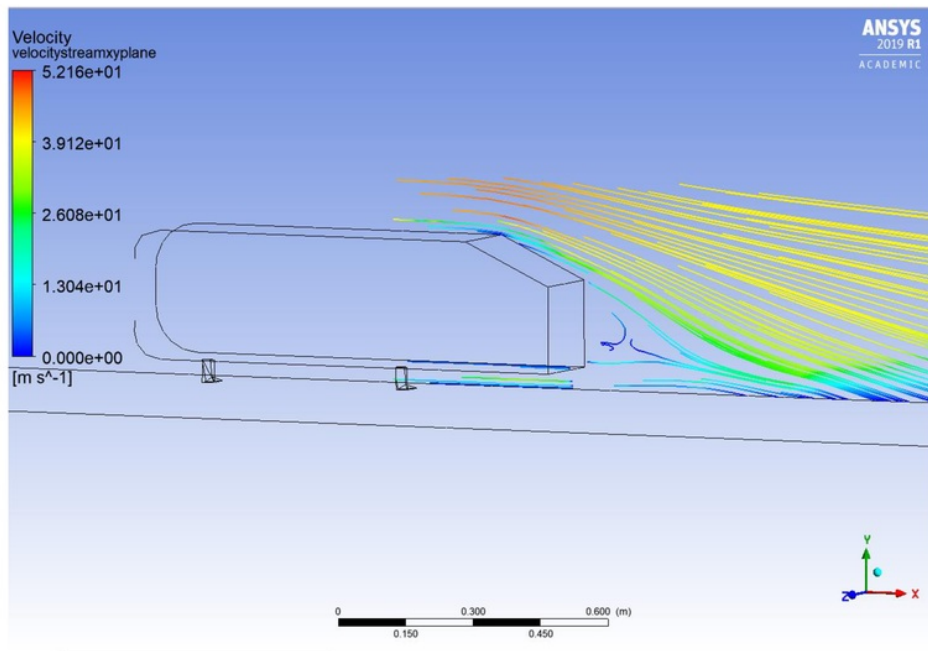
was carried out using ANSYS results where velocity, pressure and turbulent distributions were seen. Figure 8-10 below shows velocity distributions, streamlines and turbulence around the Ahmed:

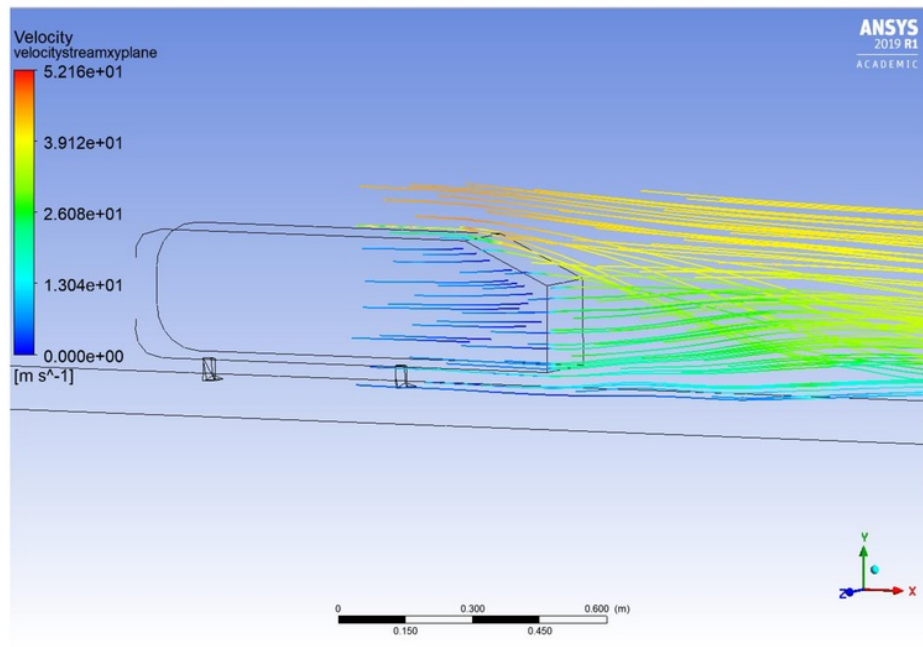




Figures 8, 9 & 10: *Simulation 1 - Velocity contours and streamlines from the y-z plane with turbulent flow strength ≥ 0.2 .*

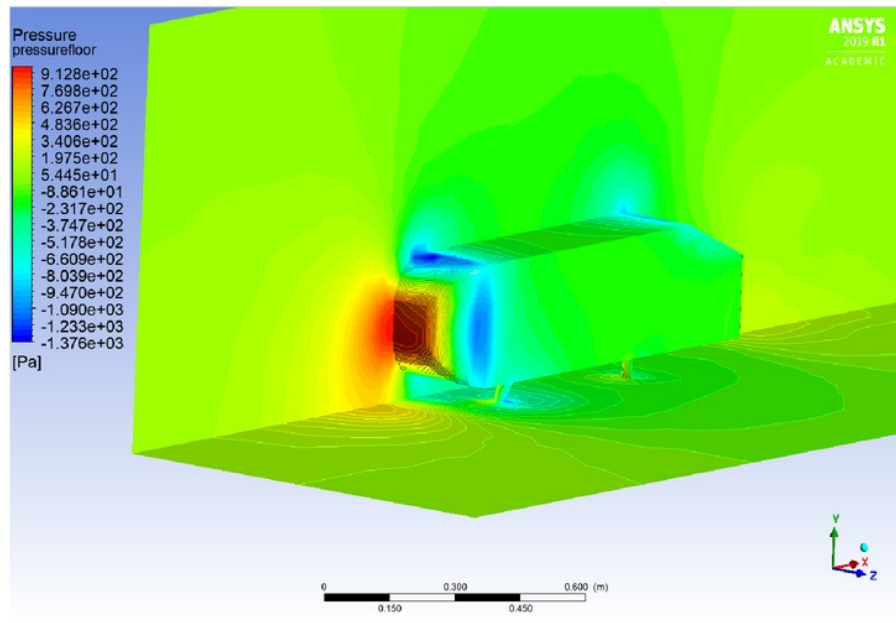
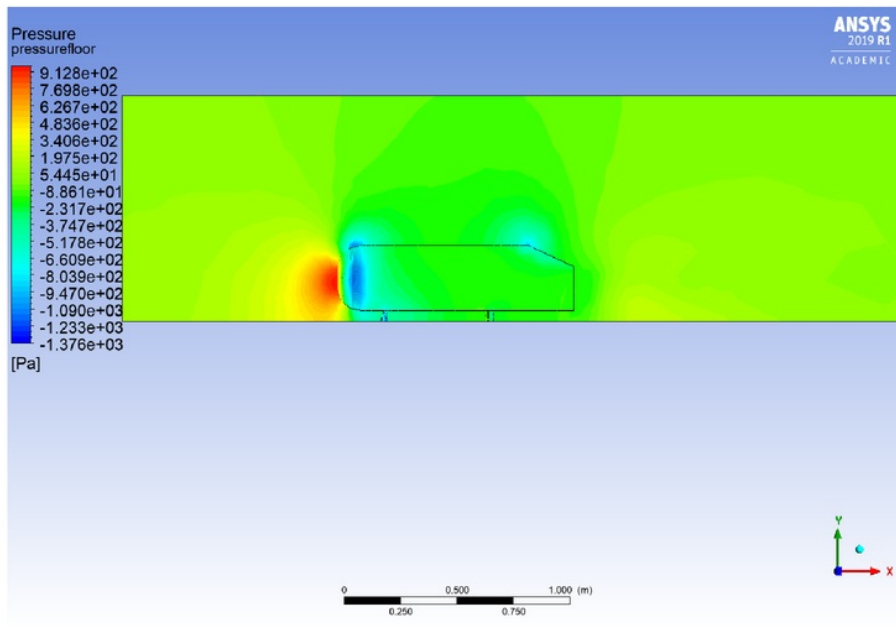
The contours and streamlines in figures 8 & 9 show that the velocity increases as the flow goes over the Ahmed body as predicted. The turbulent flow regions are seen on the Ahmed edges and a vortex is seen from the Ahmed back as predicted in figure 3 region C. However, the mesh created in 3.2 would need to be improved as the turbulent vortex seems to be increasing in strength and should be dissipating. Turbulent regions around the Ahmed legs were observed for strengths ≥ 0.1 . Furthermore, figures 11, 12 and 13 show velocity streamlines in the x-y plane for $z=0\text{m}$, 0.1m and 0.2m :





Figures 11, 12 & 13: Simulation 1 – Velocity streamlines from the x-y plane with $z=0m$, $0.1m$ and $0.2m$.

Figure 12 shows an eddy current on the Ahmed back as predicted in figure 3 region A. However, region B was not seen and so demonstrating further that the domain/mesh/solver needed to be improved. Finally, a pressure distribution was analysed:



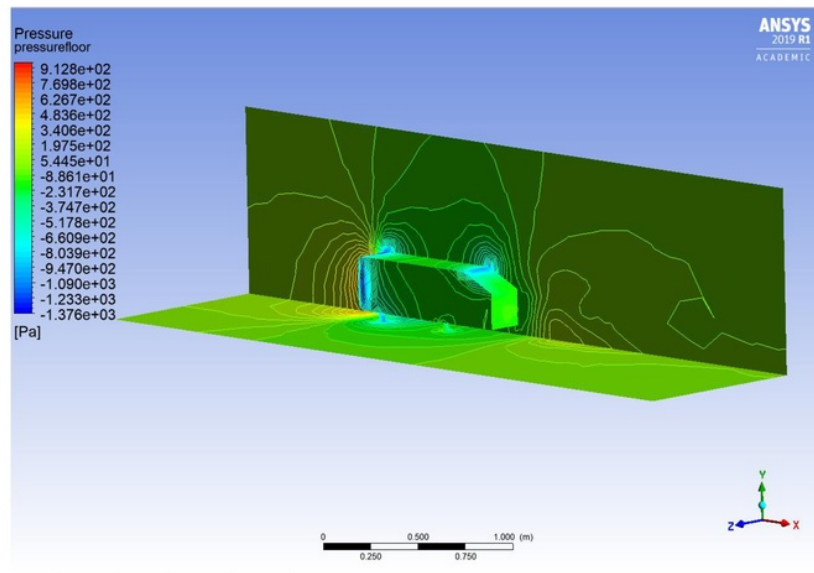


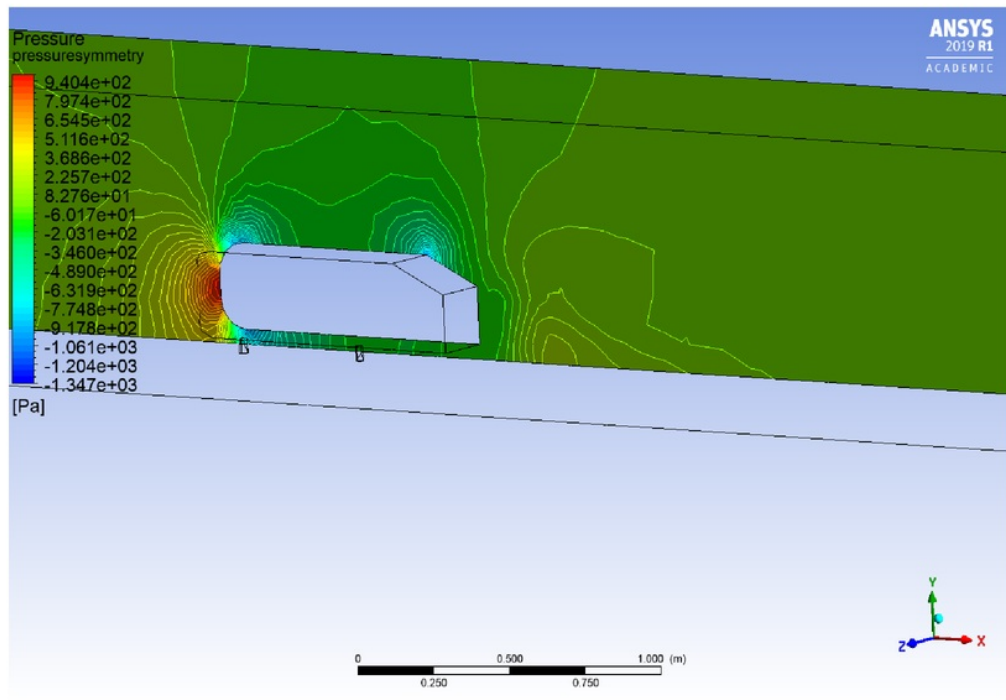
Figure 14, 15 & 16: Simulation 1 - Pressure distributions.

Figures 14 and 15 both confirm the prediction that the pressure increases as the flow meets the Ahmed body and decreases over the top and bottom edges as the velocity increases. Moreover, figure 16 shows that the different pressure contour lines vertically from the inlet to the Ahmed front and from the Ahmed back to the outlet. This suggests that the domain box chosen is not big enough as the pressure flow is not uniform in pressure from the inlet and does not normalise when travelling to the outlet. After analysis it became evident that several parameters could be improved on to produce accuracy in results.

4 Investigating Simulation Parameters

4.1 Domain Sizes

Simulation 2 was carried out using a bigger domain box from $x=6\text{m}$ to 8m ; an additional 2 meters in length at the back of the Ahmed body. Using ICEM the domain and Ahmed was re-meshed creating more volume elements and the solver was ran under this new change. The solution converged on the 90th iteration a little longer than simulation 1 producing lift and drag of 0.287 and 0.485; increasing the domain had improved lift but was still 0.058 out from experimental values whereas drag deteriorated with an increase from the previous simulation. Figure 17 below illustrates a visual change analysed:

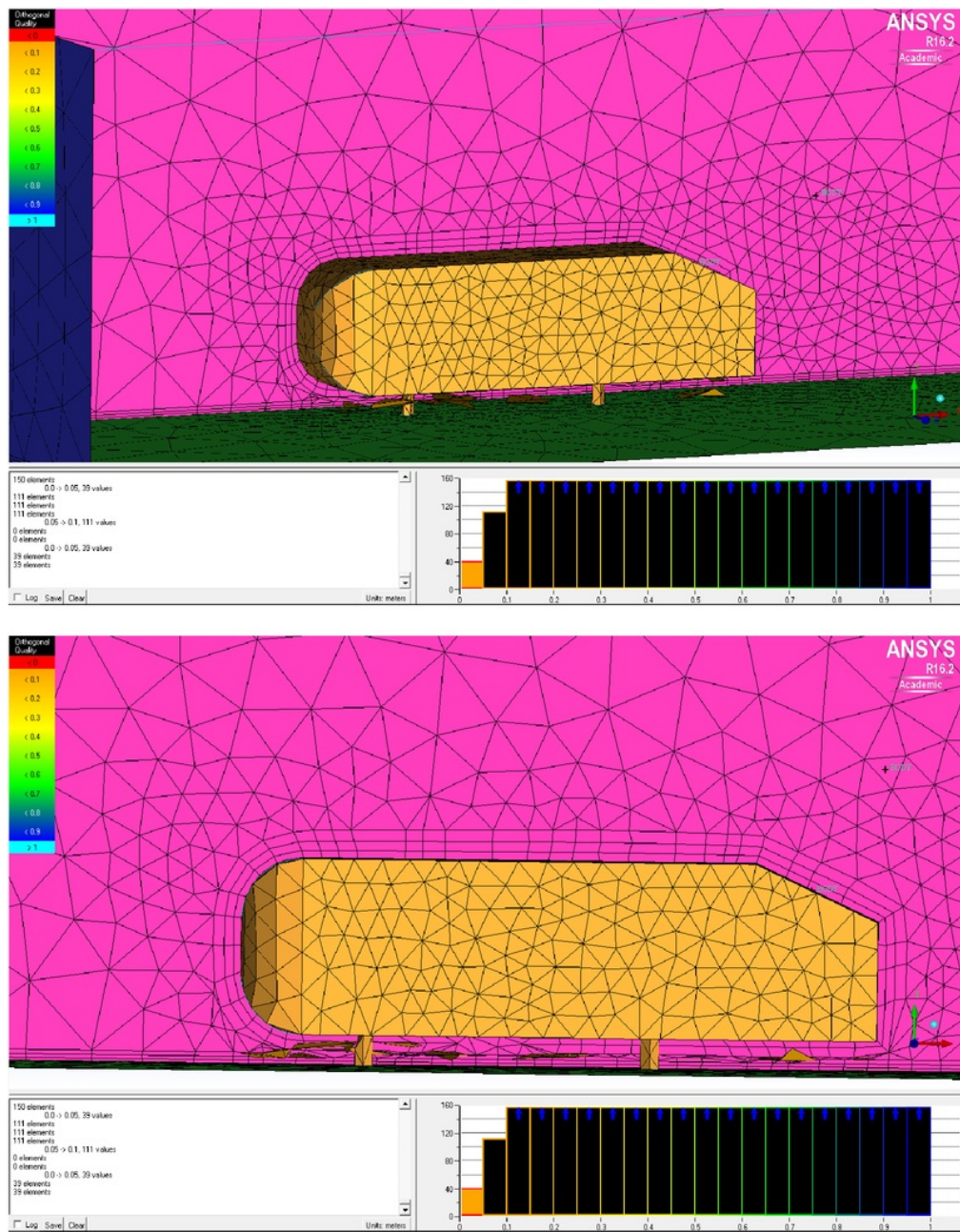


Figures 17: Simulation 2 - Pressure distribution.

Notably the pressure contours from the Ahmed back to outlet had normalised and at this stage it was not decided to increase the domain from the inlet to the front of the Ahmed. Instead the domain box was increased again from $x=8\text{m}$ to 10m for simulation 3; an additional 2 meters in length at the back of the Ahmed body. Simulation 3 was carried out with the solution converging on the 106th iteration producing lift and drag of 0.282 and 0.456; the drag slightly improved in this simulation whereas the lift deteriorated. No changes in results were noticeable in simulation 3.

4.2 Prism Layers

Another parameter investigated was the introduction of prism layers as these help simulate boundary layer effects i.e. the Ahmed and floor (ANSYS 2015). Simulations 1-3 showed poor boundary layer effects in figure 8 with abrupt changes in velocity from the Ahmed and floor. Another mesh was created with a 3-prism layer boundary on the Ahmed and floor. At this point it was found the orthogonal quality and skew were poor after re-meshing as show in figures 18 and 19 below:

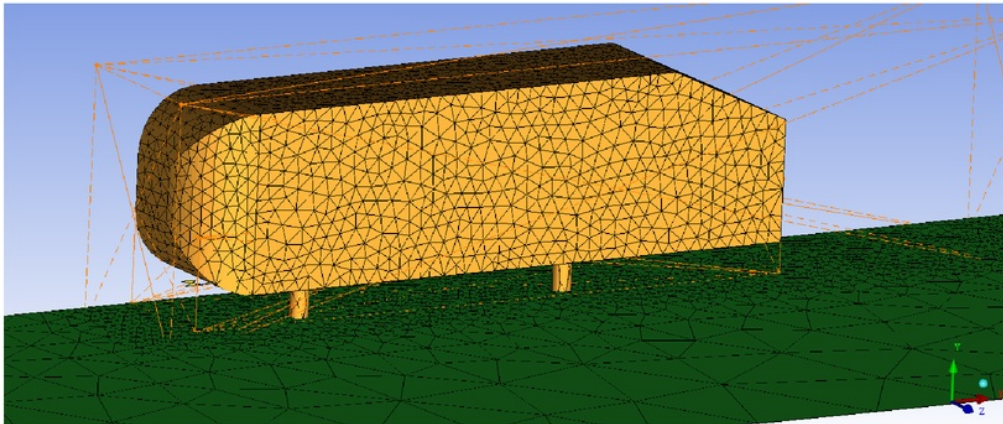


Figures 18 & 19: Simulation 4 – orthogonal and skew quality.

It was important to check orthogonal and skew quality to ensure there were no difficulties in the solver convergence and that mesh was valid for boundary layers; anything < 0.1 for orthogonal or skew were bad. Referring to figures 18 and 19 it was noted most poor-quality elements were underneath the Ahmed and to adjust this poor quality the 'density box' element size was decreased

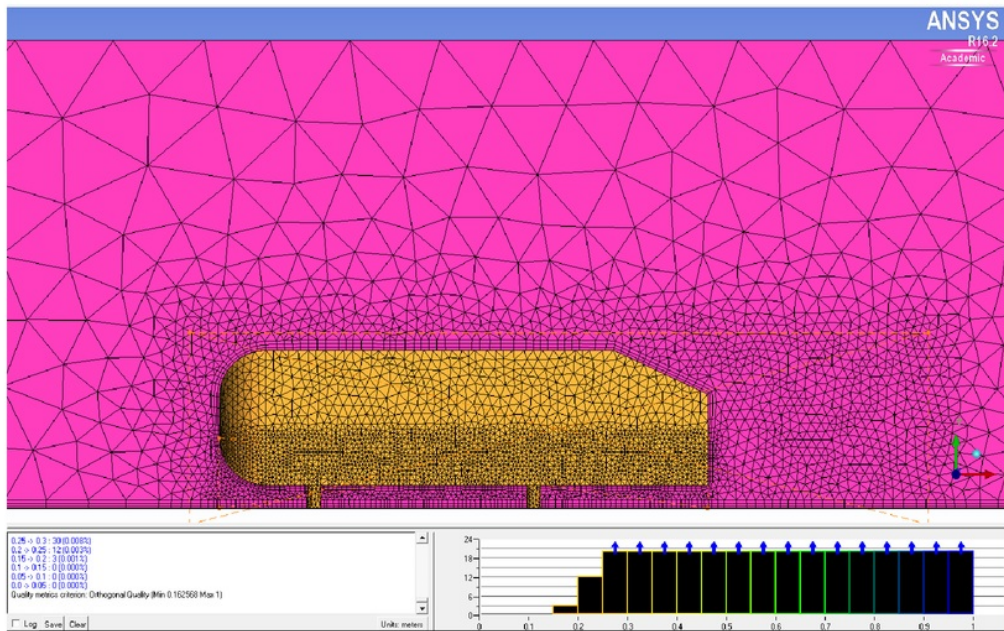
from 0.05 to 0.025. The orthogonal quality and skew were analysed again however were still found to be poor within these regions.

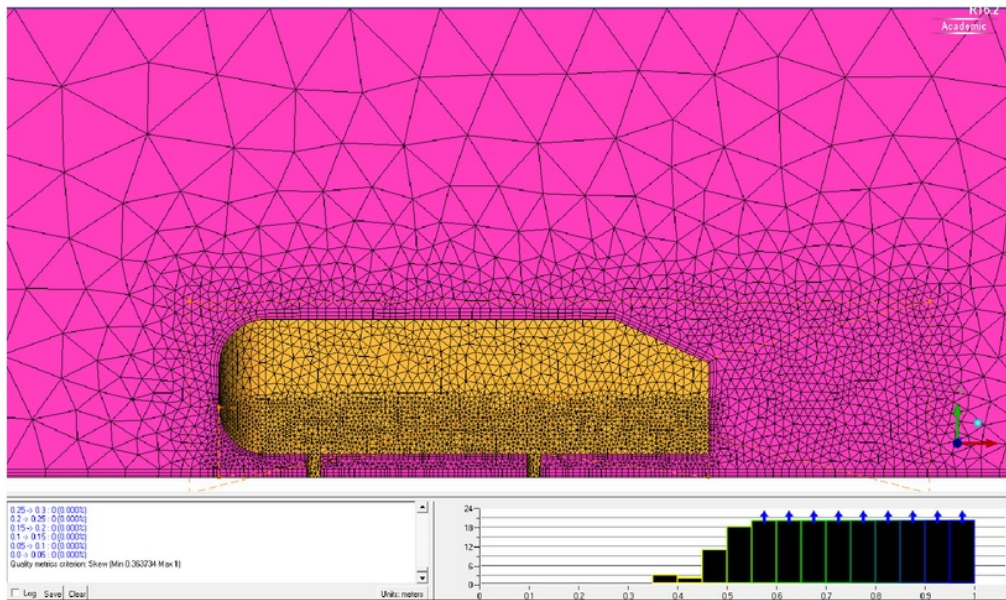
A new density box with element size 0.01m was introduced to refine these elements as shown in figure 20:



Figures 20: Simulation 4 – new floor density box.

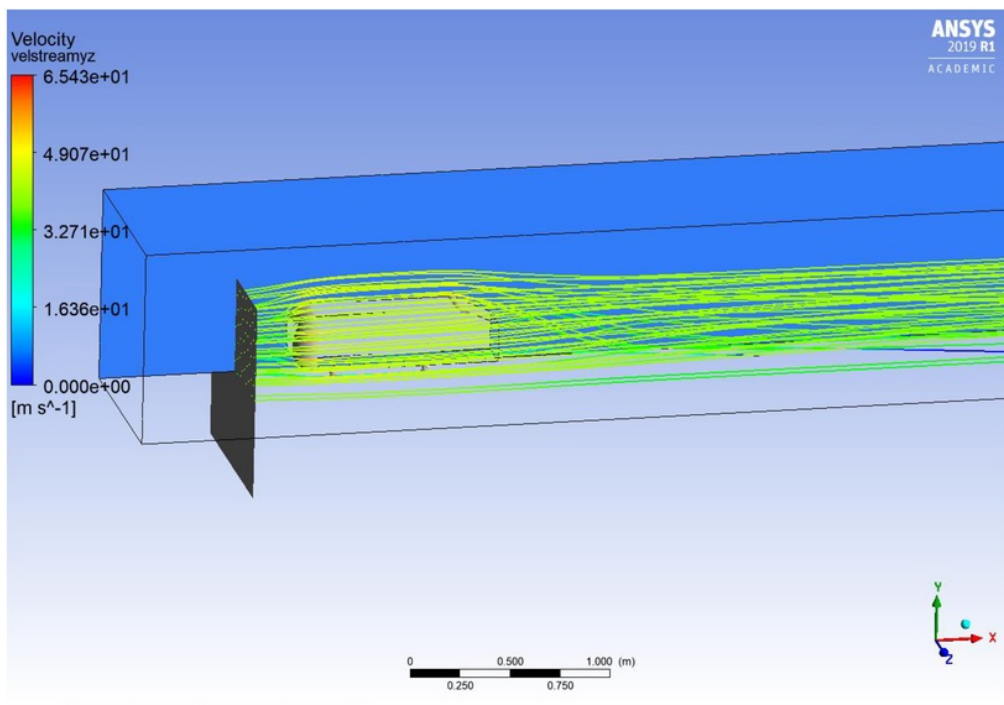
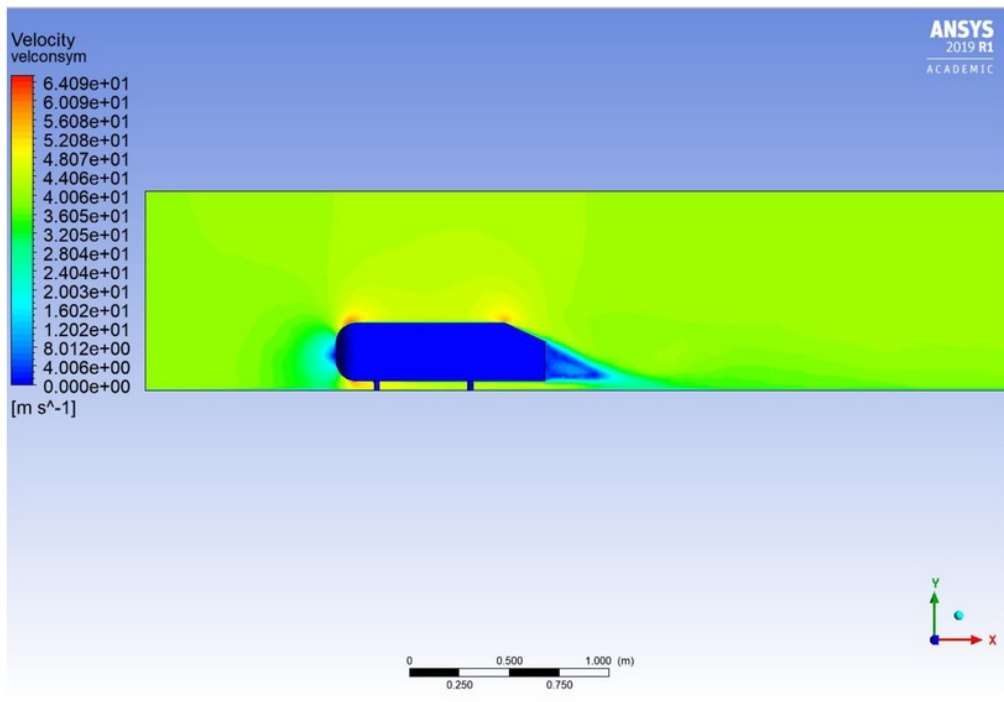
After re-meshing the orthogonal quality and skew were analysed again:

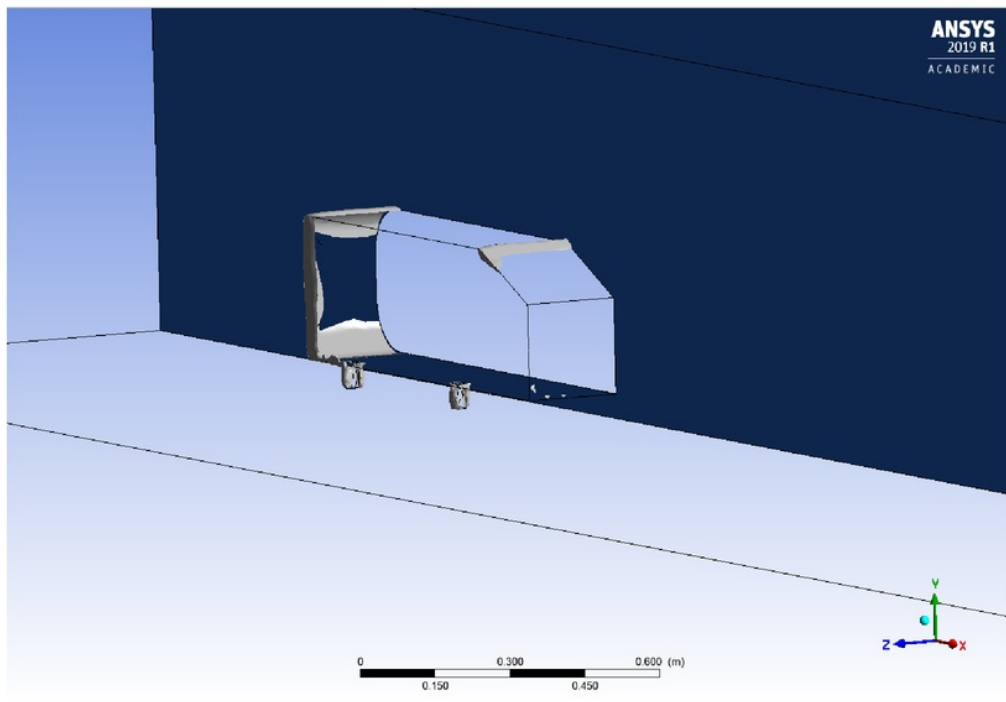




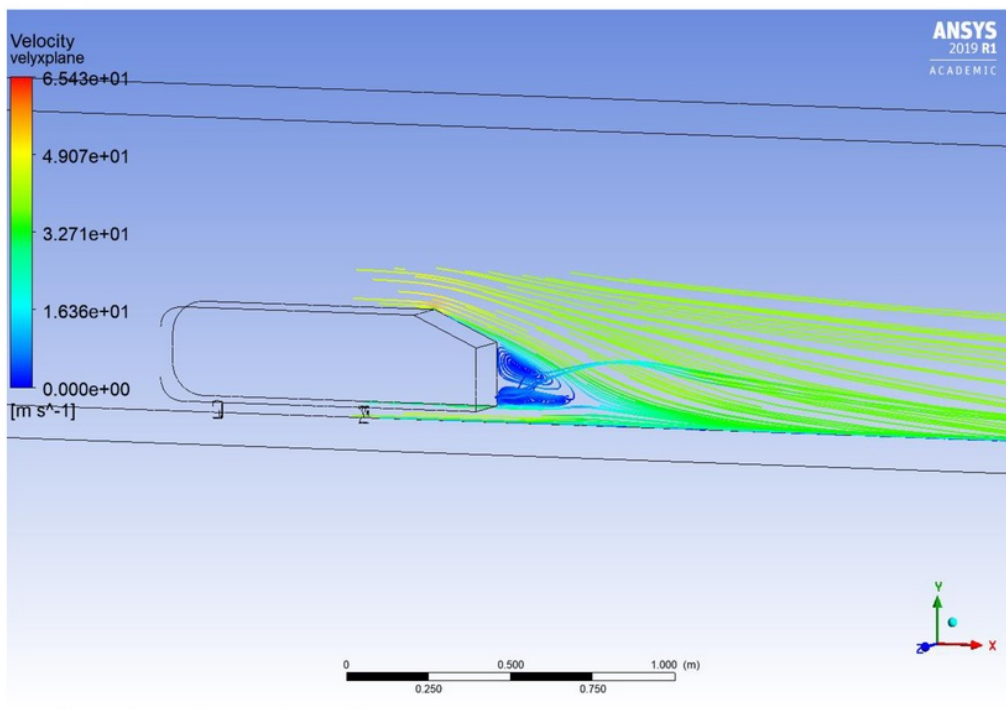
Figures 21 & 22: *Simulation 4 – orthogonal and skew quality with new floor density box.*

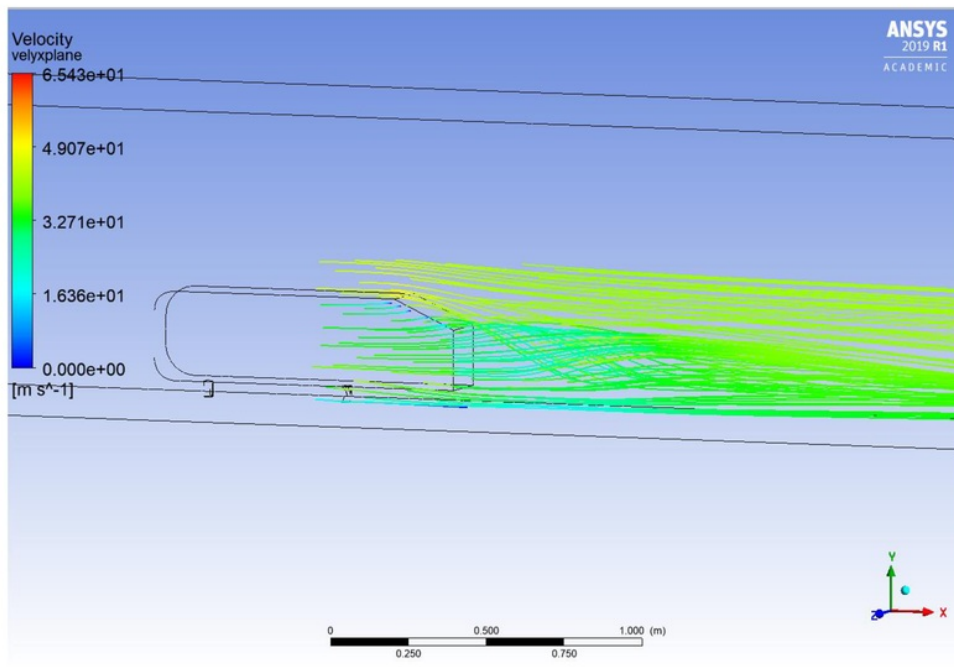
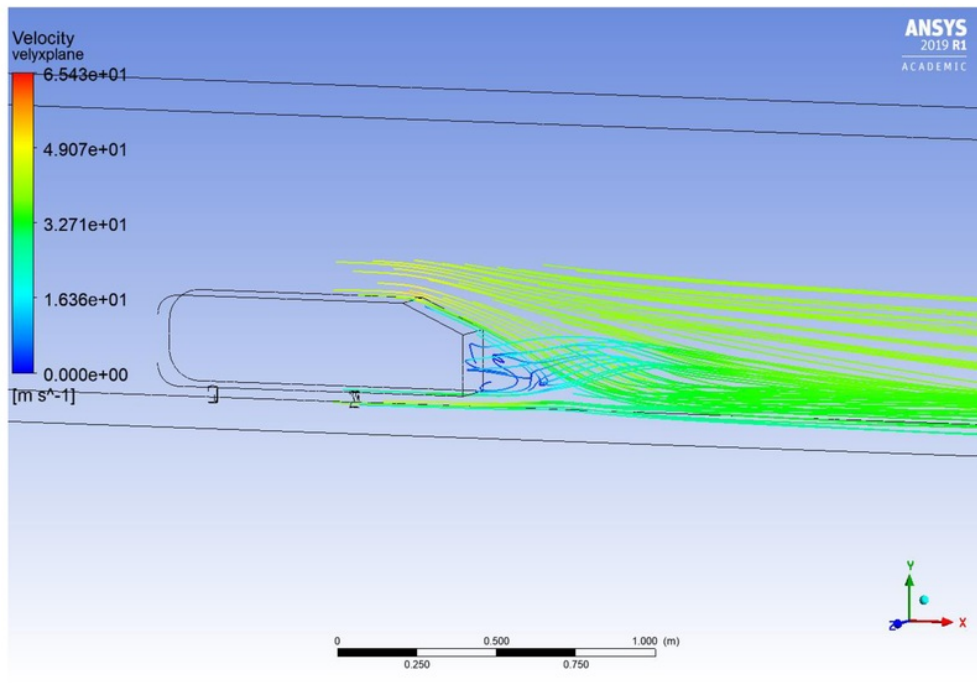
Both orthogonal and skew were greatly improved and simulation 4 was then carried out. The simulation converged on its 162th iteration with lift and drag values 0.375 and 0.370; both were improved when compared to simulation 3 but still inaccurate. Figures 8-16 were re-produced below for simulation 4 below:



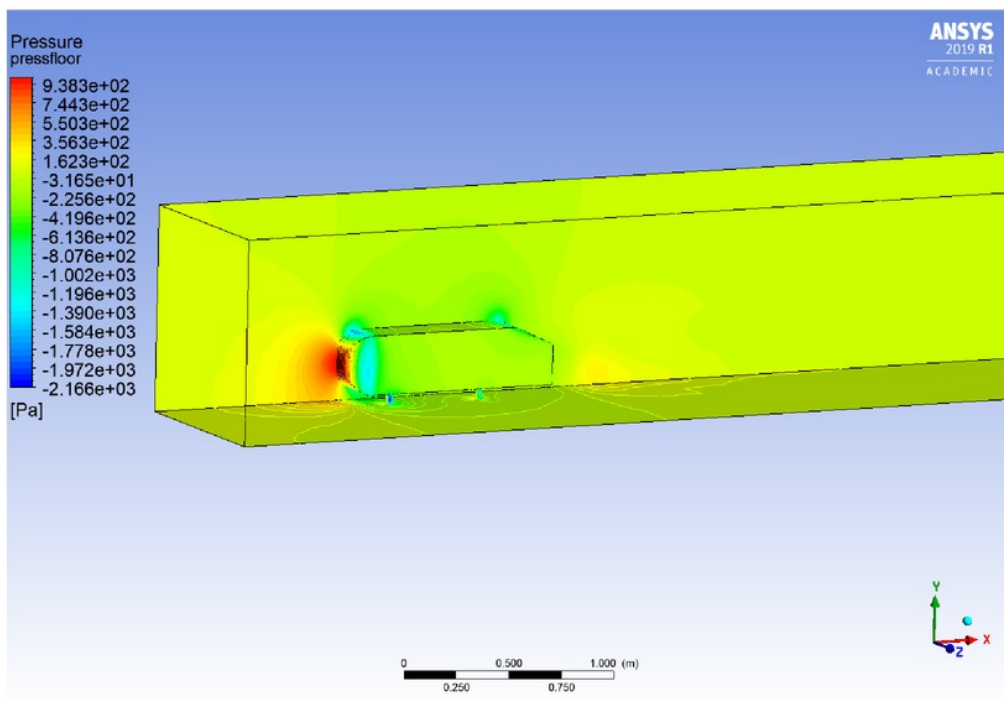
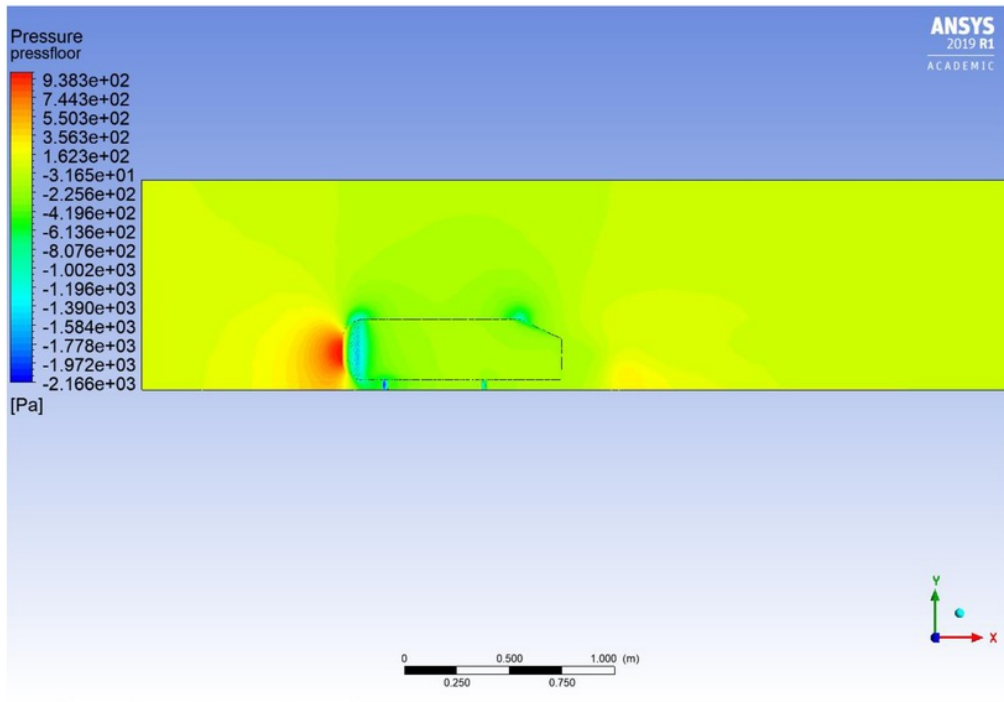


Figures 23, 24 & 25: Simulation 4 – Velocity contours and streamlines from the y-z plane with turbulent flow strength ≥ 0.2 .





Figures 26, 27 & 28: Simulation 4 – Velocity streamlines from the x-y plane with $z=0\text{m}$, 0.1m and 0.2m .



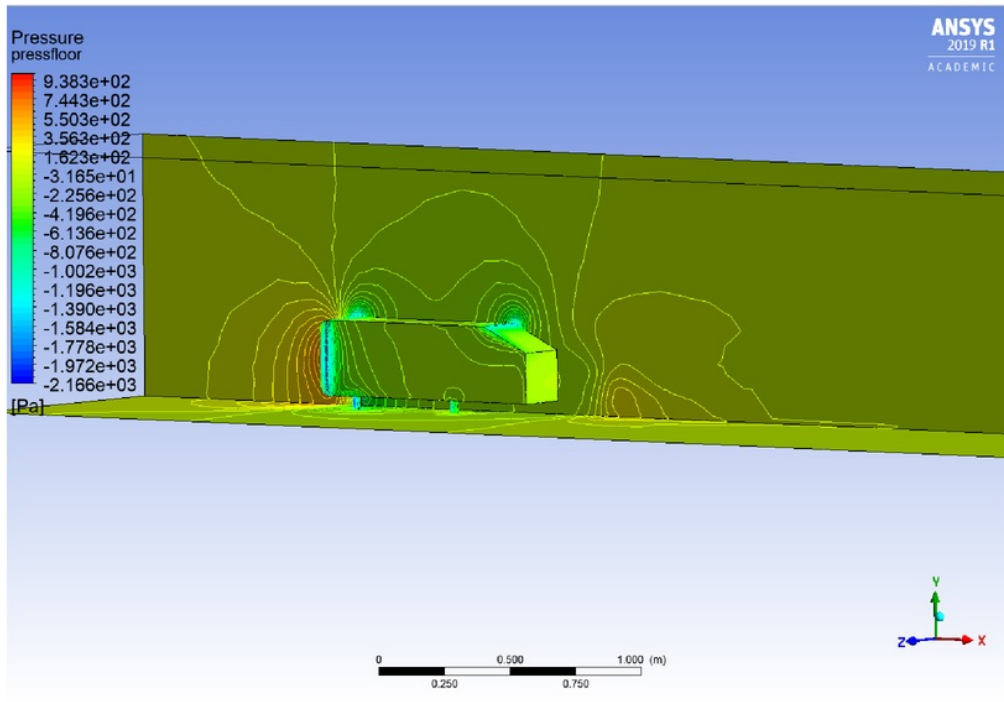


Figure 29, 30 & 31: Simulation 4 – Pressure distribution..

Notable changes were seen with the addition of a 3-prism layer and a floor density box; figure 8 and 23 have different velocity distributions and wake volume; figure 24 illustrates a realistic C-pillar generating and dissipating; figure 26 demonstrates the two recirculation regions in figure 3 in contrast to figure 11; and pressure distributions are different.

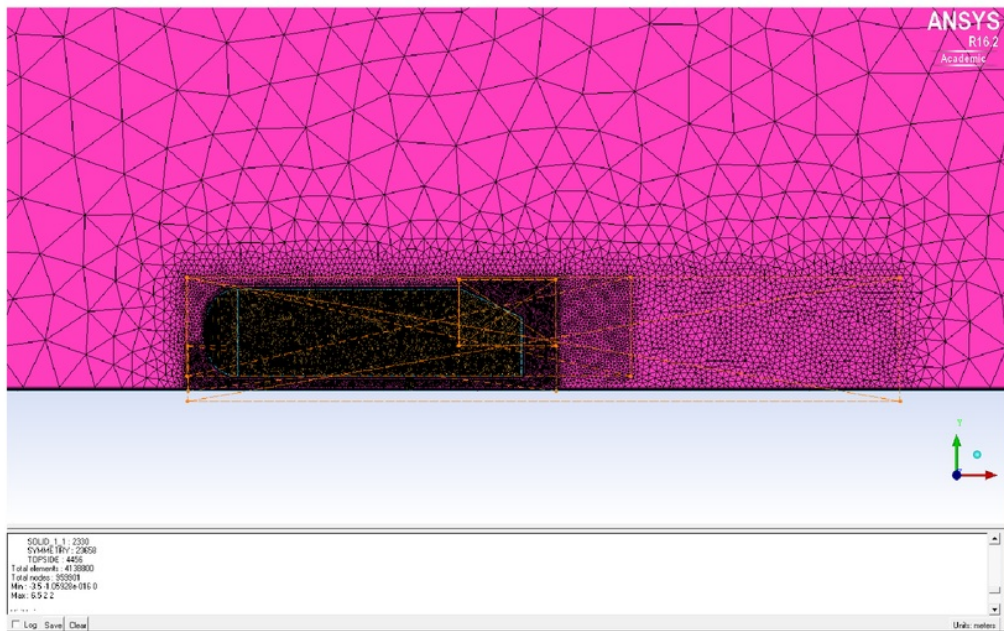
Simulation 5 was solved with 5-prism layers with a maximum height of 0.001m (to improve quality) converging on 235 producing lift and drag values 0.437 and 0.343 respectively. Lastly, simulation 6 was solved with 7-prism layers with a maximum height of 0.001m converging on 215 producing lift and drag values 0.406 and 0.367 respectively. Results showed the drag coefficient was improving however the lift wasn't. Both results were analysed further and no real change from 3-prism layer were notable.

4.3 Mesh Strategies

Simulation 7 investigated meshing by increasing the number of elements in specified areas. To address the inlet to Ahmed back pressure distribution irregularity the domain was increased to $x =$

13.5 (-3.5m front, and 10m back), $y=2\text{m}$, and $z = 2\text{m}$. Moreover, the surface element sizes of the Ahmed and Ahmed back were defined as 0.005m as recommended by Lanfrit 2015.

Additionally, the size of the density 'box' created in simulation 1 was increased to cover further volume ahead of Ahmed back. Furthermore, density boxes were added to Ahmed back and slope. The slope had an element size of 0.005m and covered the Ahmed wake region. The Ahmed back box covered the top of the Ahmed body (excluding the legs) and a meter ahead of the Ahmed back with finer detail of element size 0.01m. The smaller floor density box created in simulation 4 had its' element size decreased to 0.005m from 0.01m also increasing its' volume at the front and back of Ahmed. All these areas were refined as previous velocity and pressure distributions showed these key areas that required improvement. Figures 32- 33 show the final mesh created which contained over 4 million elements:



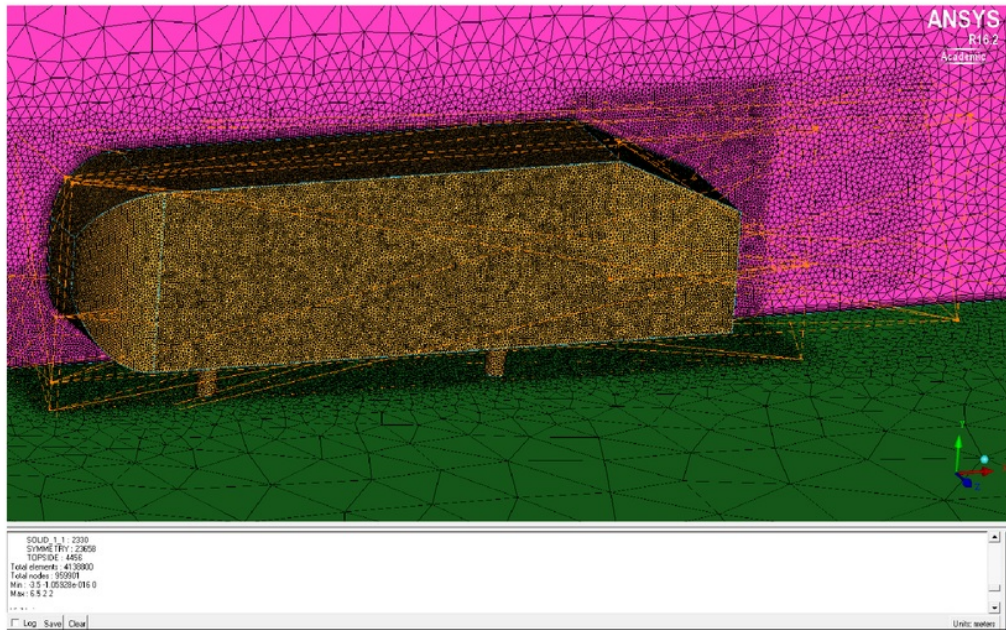
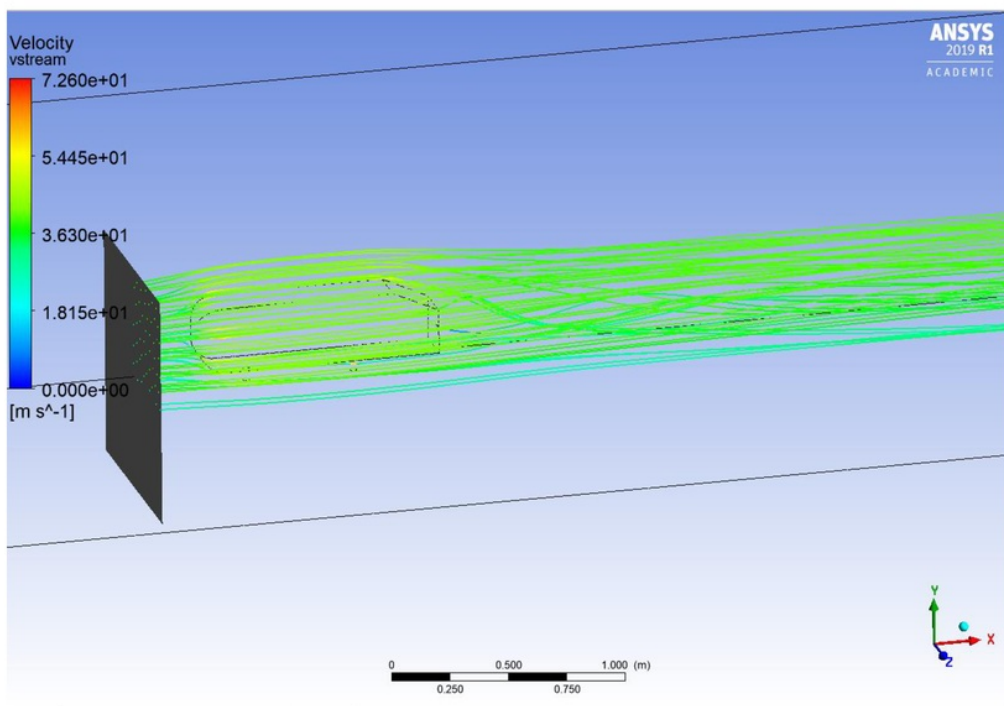
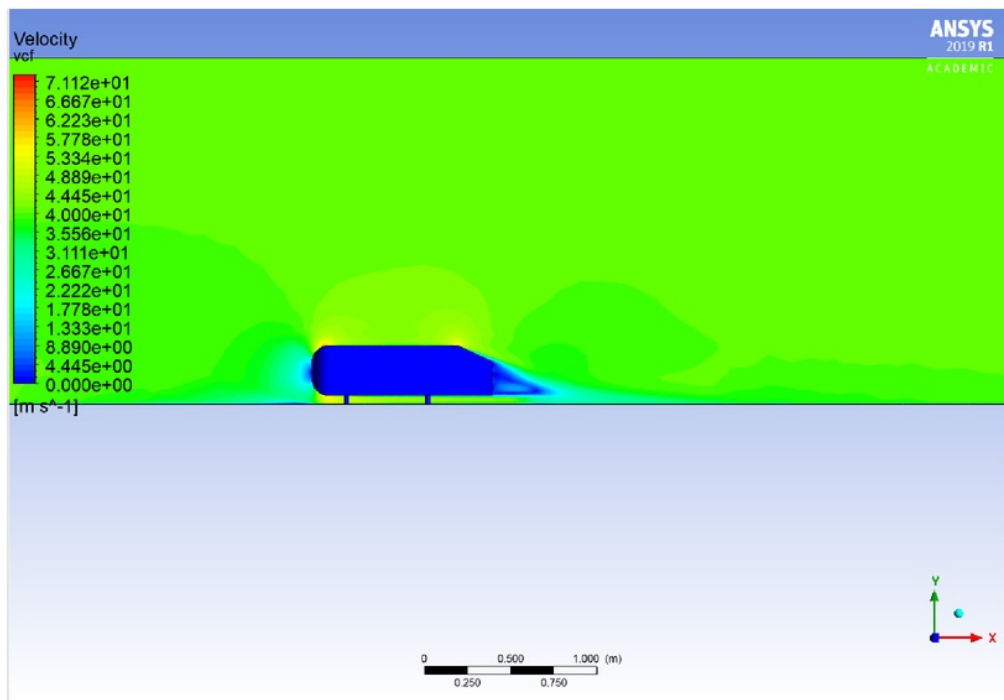
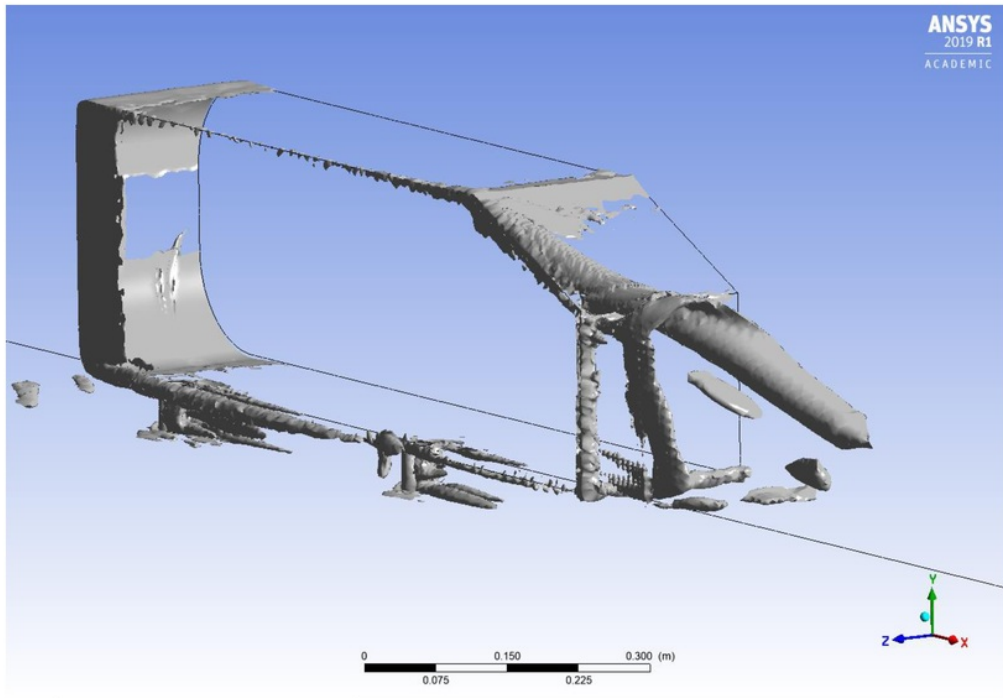


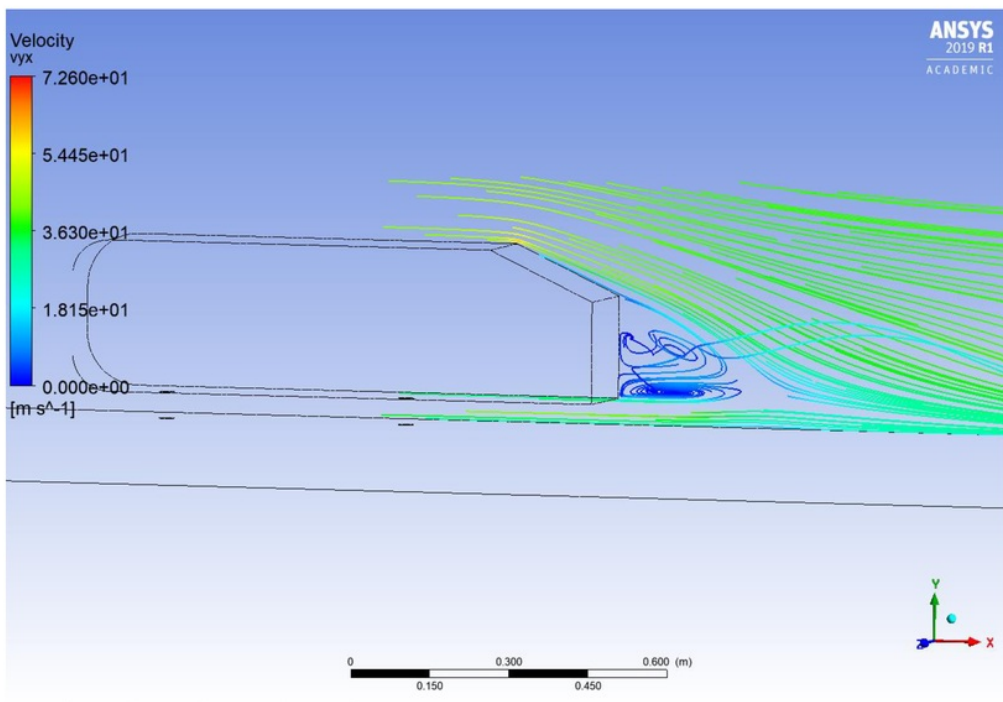
Figure 32 & 33: Final mesh created in ICEM.

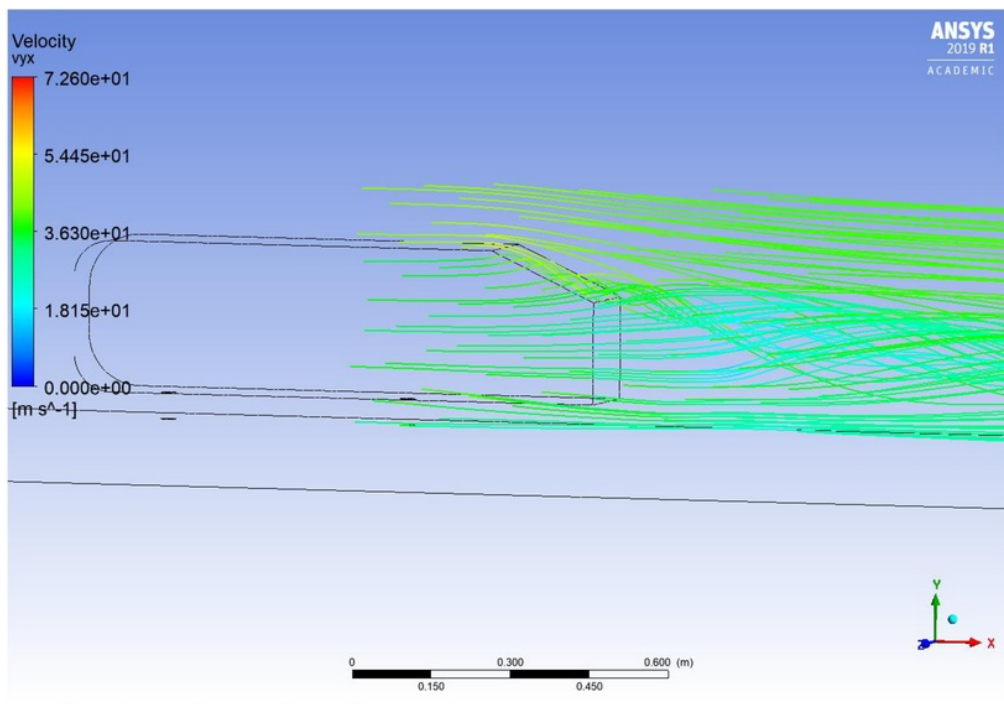
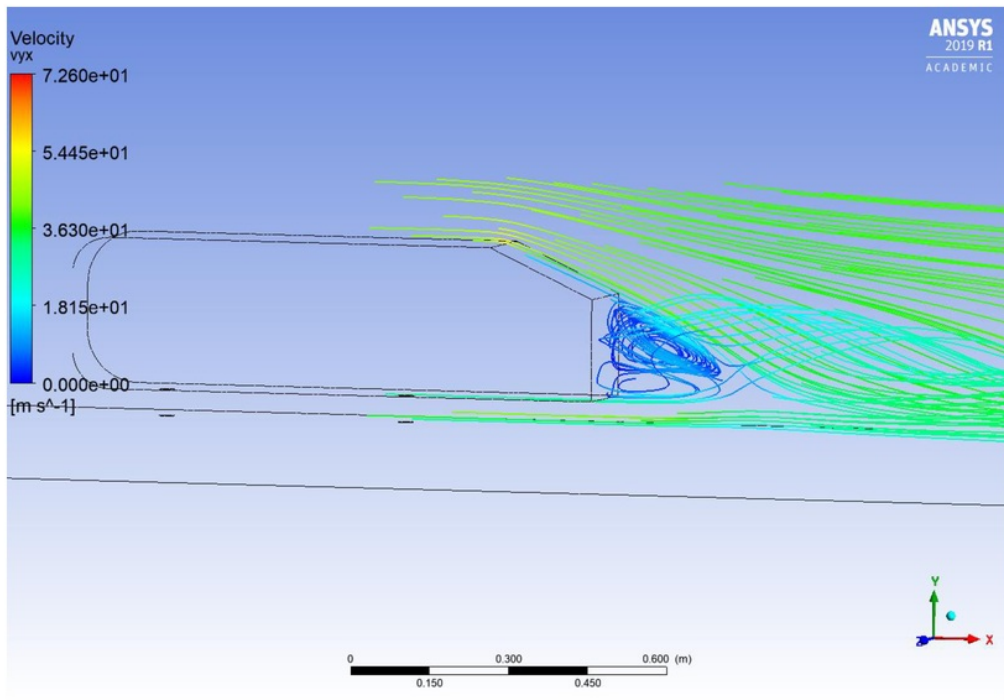
Simulation 7 was carried out converging on 241 and producing lift and drag values 0.342 and 0.288; these results were accurate with lift having 0.87% error and drag 3.68% error when compared against experimental results. Simulation 7 was analysed as below:



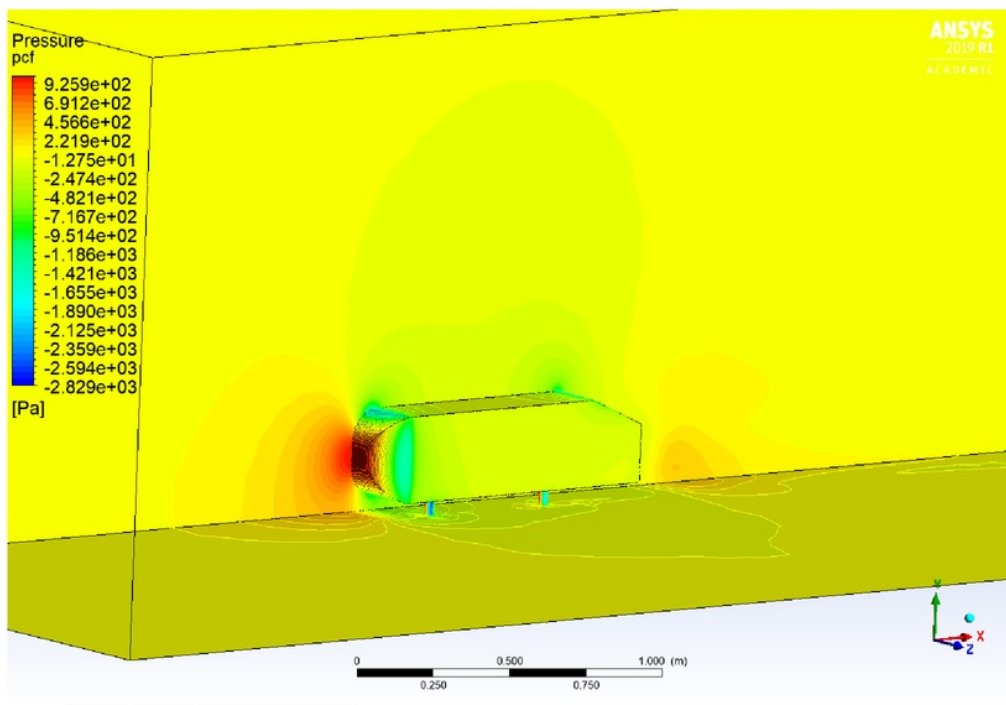
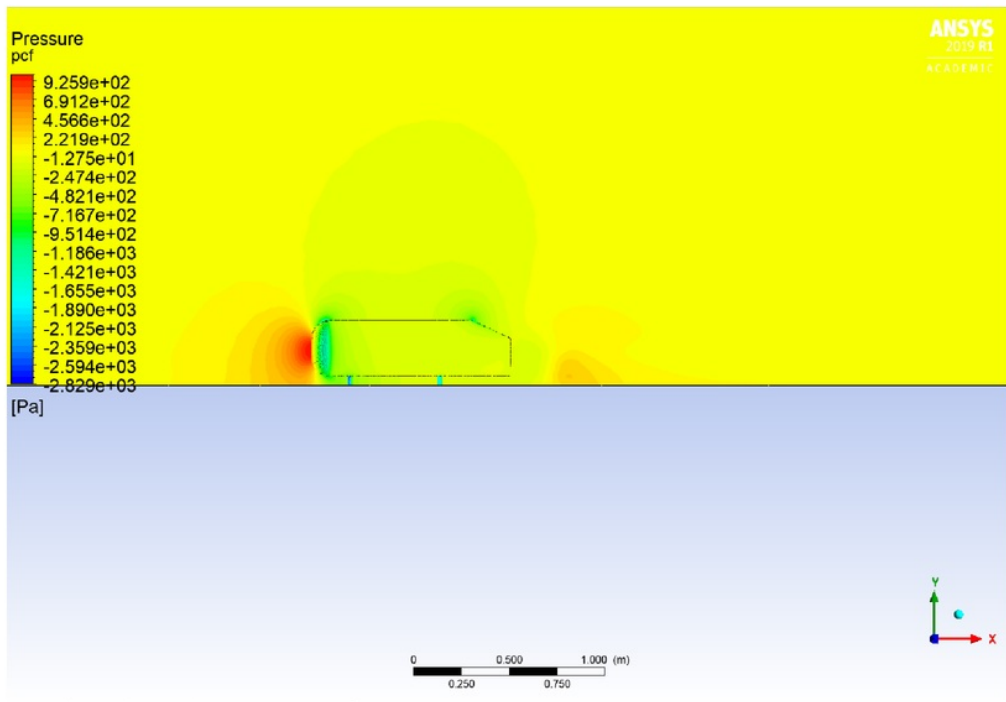


Figures 34, 35 & 36: Simulation 7 - Velocity contours and streamlines from the y-z plane with turbulent flow strength ≥ 0.02 .





Figures 37, 38 & 39: Simulation 7 – Velocity streamlines from the x-y plane with $z=0\text{m}$, 0.1m and 0.2m .



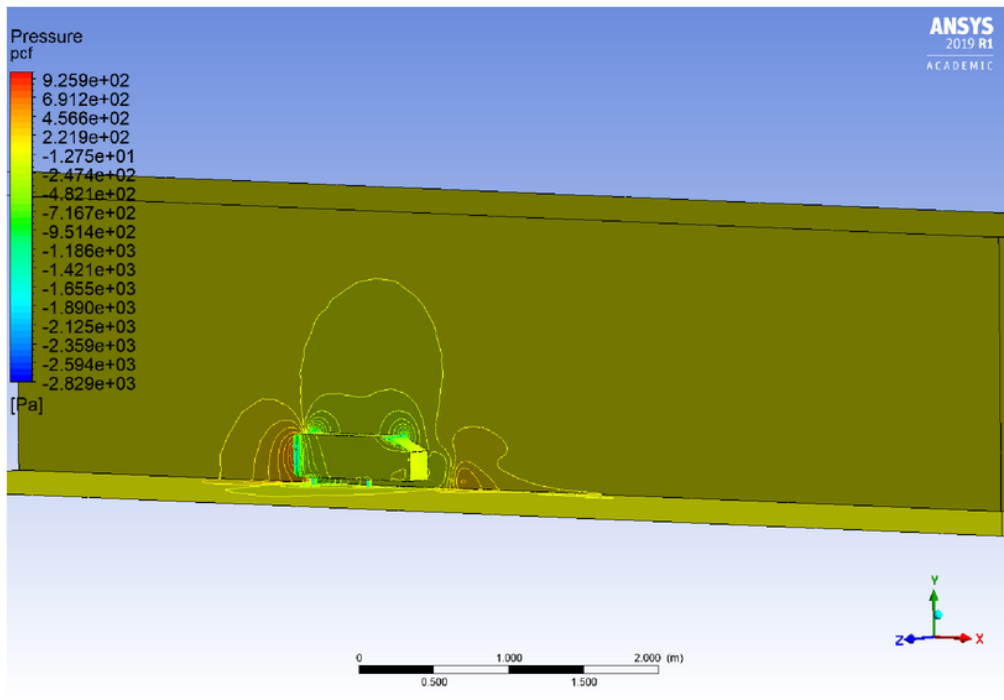
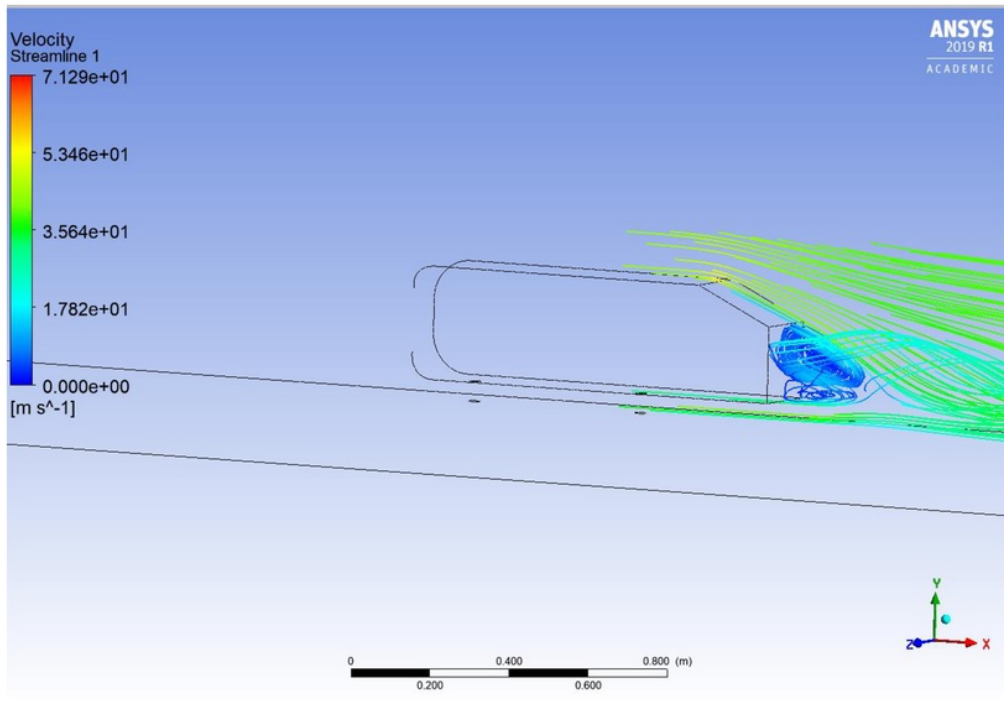


Figure 40, 41 & 42: Simulation 7 – Pressure distributions.

Figure 36 shows the C-pillar very prominently and after applying a finer element detail on the Ahmed floor to the body the pressure distribution around the legs are notable. Finally figure 42 shows that the pressure is normalised from the inlet, topside and outlet in contrast to figure 16.

4.4 Turbulence Models

Turbulence models were also investigated to note any changes. The same domain and mesh were used from simulation 7 except for the k-epsilon standard viscous model being used instead of realizable. The solver converged on 198 with lift and drag of 0.346 and 0.3 with errors 0.29% and 0.33% respectively. The only notable change in results were the two upper and lower recirculation regions being more prominent as shown in figure 43 below:



Figures 43: Simulation 8 – Velocity streamlines from the x-y plane with $z = 0.1m$.

Simulations 9 and 10 were carried out with viscous models as k-omega standard and omega SST however the solver did not reach a converged solution and so results were not analysed for these two models.

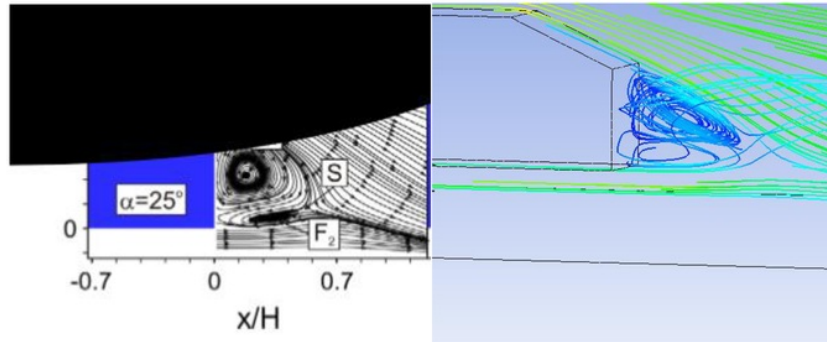
4.5 Overall Results

Table 2 below summarises all simulation results:

Simulation Number	Lift Coefficient	Drag Coefficient	Converged Number	Prism Layer
1	0.466	0.479	86	0
2	0.287	0.485	90	0
3	0.282	0.456	106	0
4	0.375	0.370	162	3
5	0.437	0.343	235	5
6	0.406	0.367	215	7
7	0.342	0.288	241	7
8	0.346	0.300	198	7
9	N/A	N/A	did not converge	7
10	N/A	N/A	did not converge	7

Table 2: ANSYS simulation results.

Table 2 reveals as the domain, mesh and solver were improved the lift and drag coefficient got more accurate; introducing prism layers and further density boxes were vital in obtaining accurate results. Furthermore, as the domain and mesh increased in size and element numbers the longer the Fluent Solver took to converge a solution. There is high confidence in the visual results produced from simulations 7 & 8 when compared to other research as shown below:



Figures 44: – 25° slant Ahmed wake region – left figure by Tunay, Sahin and Ozbolat 2014 and right image carried out in simulation 7.

Figure 44 demonstrates the upper and lower recirculation regions by Tunay, Sahin and Ozbolat 2014 that are attached to the Ahmed body. These results are exactly replicated in simulations 7 & 8 and are further supported by Lienhart, Stoots and Becker (2019) and Meile et al (2011). However, other research showed that using k-epsilon realizable non-equilibrium wall function for near wall treatment provided the most accurate results and this was not used.

5 Conclusions

A 3D simulation for a 25° slant Ahmed body travelling straight ahead at 40ms^{-1} using ANSYS and Fluent was successfully completed. Through investigating the domain size, number of prism layers, mesh strategy, solver settings and turbulent models an optimum visualisation of the Ahmed external aerodynamics was obtained along with drag and lift coefficients of 0.346 and 0.3 within errors 0.29% and 0.33% respectively using k-epsilon standard. It is recommended for future simulations that; domain/mesh used for simulation 7 be solved under a k-epsilon realizable non-equilibrium wall function as recommended by Lanfrit 2015; and to re-evaluate simulations 9 and 10 to obtain converged solutions to analyse the k-omega standard and omega SST viscous models.

6 References

- Ahmed S, Ramm G and Faltin G (1984). Some Salient Features of the Time Averaged Ground Vehicle Wake. SAE Technical Paper Series.
- ANSYS (2015). [Presentation] Lecture 2: Introduction to the CFD Methodology 16.0 Release.
- Halliday D, Resnick R and Walker J (2010). Principles of physics extended. 9th ed. Hoboken, N.J.: Wiley, pp.371-375.
- Lanfrit M (2015). Best practice guidelines for handling Automotive External Aerodynamics with FLUENT. Darmstadt Germany, pp.1-14.
- Lienhart H, Stoots C and Becker S (2019). Flow and Turbulence Structures in the Wake of a Simplified Car Model (Ahmed Model). [online] Cfd.mace.manchester.ac.uk. Available at: http://cfm.mace.manchester.ac.uk/ercotac/database/cases/case82/Case_ref/STAB2000_NNFM.pdf [Accessed 22 Apr. 2019].
- Linfield K and Mudry R (2008). Pros and Cons of CFD and Physical Flow Modeling. [online] Airflowsciences.com. Available at: <http://www.airflowsciences.com/sites/default/files/docs/Pros-and-Cons-of-CFD-and-Physical-Flow-Modeling.pdf> [Accessed 26 Apr. 2019].
- Meile W, Brenn G, Reppenhagen A, Lechner B and Fuchs A (2011). Experiments and numerical simulations on the aerodynamics of the Ahmed body. CFD Letters, 3(1), pp.32-39.
- Process of CFD (2019). [image] Available at: Cfdninja.com. (2019). CFD Beginners – Process of CFD – CFDNINJA. [online] Available at: <https://cfdninja.com/beginners/process-of-cfd/> [Accessed 18 Apr. 2019]. [Accessed 18 Apr. 2019].
- Rao A, Minelli G, Basara B and Krajnović S (2018). On the two flow states in the wake of a hatchback Ahmed body. Journal of Wind Engineering and Industrial Aerodynamics, 173, pp.262-278.
- Thabet S and Thabit T (2018). CFD Simulation of the Air Flow around a Car Model (Ahmed Body). International Journal of Scientific and Research Publications (IJSRP), 8(7), pp.517-525.
- Toet W (2019). Aerodynamics of F1. [online] Formula1-dictionary. Available at: http://www.formula1-dictionary.net/aerodynamics_of_f1.html [Accessed 22 Apr. 2019].
- Tunay T, Sahin B and Ozbolat V. (2014). Effects of rear slant angles on the flow characteristics of Ahmed body. Experimental Thermal and Fluid Science, 57, pp.165-176.
- Wu B (2011). CFD investigation of turbulence models for mechanical agitation of non-Newtonian fluids in anaerobic digesters. Water Research, 45(5), pp.2082-2094.