



School of Engineering MSc Mechanical Engineering

Simulation and Modelling – M7X01380

Assignment 1 Computational Fluid Dynamics Ahmed Body Analysis

James Milling
1503259
March 2019

Abstract

The report details the development of a CFD model and solution for an Ahmed body with a rear slant angle of 25° with a flow rate of 40 m/s utilising ANSYS Workbench R16.2 Academic and fluent solver software. Refinement of the domain and mesh are discussed, and result considered against experimental data and literature review results.

Contents

Abstract	ii
List of Figures	iv
List of Tables	iv
1.0 Introduction	1
2.0 Background	2
2.1 Pre-Processing.....	2
2.2 Solving	2
2.3 Post-Processing	2
3.0 Methodology.....	3
3.1 Fluid Domain Geometry	3
3.2 Meshing Quality	4
3.3 Meshing Strategy	4
3.4 Boundary Conditions.....	4
3.5 Turbulence Model.....	5
3.6 Solver Settings.....	5
4.0 Results.....	6
4.1 Initial Mesh and Solution Run 1	6
4.2 Progressive Refinement of Domain and Mesh	8
4.3 Simulation Run 2 and 3	9
4.4 Simulation Run 4	9
4.5 Simulation Run 5	10
4.6 Simulation Run 6, 7 & 8 – 3 Prism layers	11
4.7 Simulation Run 9 – 5 Prism layers.....	14
4.8 Simulation Run 10, 11 & 12 – 7 Prism layers	14
4.9 Simulation Run 12 – Evaluation of Turbulence models	16
4.10 Postprocessing – Simulation Run 12	17
5.0 Conclusion	21
6.0 References	22

List of Figures

Figure 1. General arrangement of Ahmed body (UWTSD, 2019)	1
Figure 2. Ahmed body geometry imported into Design Modeler with domain box plan on X-Y plane.	3
Figure 3. Final domain form with subtracted Ahmed body profile highlighted green.	3
Figure 4. Initial (Run 1) domain mesh applied within the ICEM CFD meshing tool.	6
Figure 5. Initial (Run 1) solution convergence history	7
Figure 6. Initial (Run 1) Solution – Contours of Velocity Magnitude	8
Figure 7. Initial (Run 1) Solution – Contours of Turbulent Kinetic Energy	8
Figure 8. Run 3 Solution - Contours of Velocity Magnitude	9
Figure 9. Density box viewed from the front face of the Ahmed body.	10
Figure 10. Density box viewed from the X-Y face of the Ahmed body.	10
Figure 11. Estimation of body surface mesh length (Adapted from Best practice guidelines for handling Automotive External Aerodynamics with fluent. Lanfrit, 2015).	10
Figure 12. Detail of Run 7 cell quality report, with poor cells highlighted in support area.	11
Figure 13. Run 8, second density box (brown) added in addition to first density box (Orange).	12
Figure 14. Detail of Run 8 cell quality report.	12
Figure 15. Run 8 Solution convergence history.	13
Figure 16. Run 8 Solution - Contours of Velocity Magnitude – Ahmed body forward	13
Figure 17. Run 8 Solution - Contours of Velocity Magnitude – Ahmed body rear	13
Figure 18. Density box 3 added as part of run 11, in the Ahmed body to floor area.	14
Figure 19. Run 10 Solution convergence history.	15
Figure 20. Run 11 Solution convergence history.	15
Figure 21. Density box 4 added as part of run 12, in the Ahmed body rear slant area.	15
Figure 22. Run 12 Solution convergence history.	16
Figure 23. Run 12 Solution - Contours of Static Pressure – Ahmed body rear	16
Figure 24. Simulation Run 12 – Postprocessing – Velocity streamlines in the near wake region	17
Figure 25. Simulation Run 12 – Postprocessing – Recirculatory flow regions and separation bubble.	17
Figure 26. Simulation Run 12 – Postprocessing – Formation of wake trailing vortex	18
Figure 27. Horseshoe vortex system in wake (Ahmed, et al., 1984).	18
Figure 28. Simulation Run 12 – Postprocessing – Vortex core region overlay, recirculatory flow regions	19
Figure 29. Simulation Run 12 – Postprocessing – Vortex core region overlay, trailing vortex.	19
Figure 30. Simulation Run 12 – Postprocessing – Pressure contour – Ahmed body rear slant.	20

List of Tables

Table 1. Experimental data (UWTSD, 2019).	1
Table 2. Results of simulation with progressive refinement	9

1.0 Introduction

The study investigates the development and validation of a computational Fluid Dynamic (CFD) model solution of the fluid flow characteristics over a simplified vehicle body and the resultant flow disruption in the near wake area. The simplified vehicle body specified for the study is the Ahmed bluff body (figure 1), which is a simplified geometric model of a passenger vehicle, having representative aerodynamic properties, and is frequently used as a computational benchmark.

The simulation utilises an Ahmed body with a rear slant angle of $\phi = 25^\circ$, where the incoming fluid flow velocity is taken as 40 m/s with the flow straight onto the body. The software package used for the simulation is ANSYS Workbench R16.2 Academic, with the Design Modeller, ICEM CFD mesh generator, Fluent solver and Results postprocessor utilised. The experimental data is supplied by UWTSD as part of the assignment specification, detailed in table 1.

Ahmed Geometry

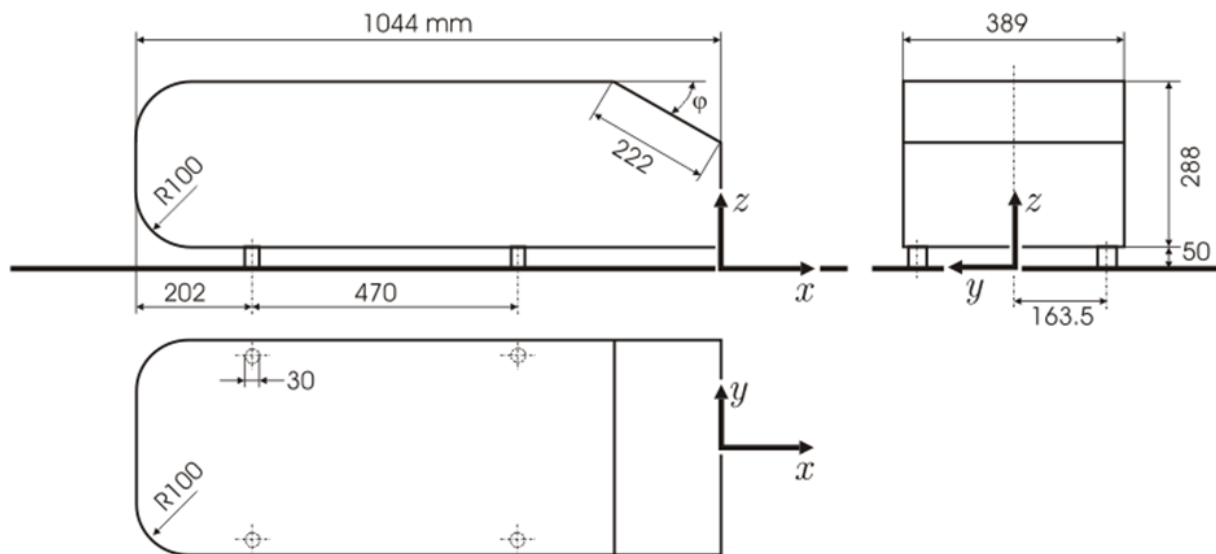


Figure 1. General arrangement of Ahmed body (UWTSD, 2019)

Experimental Data for Ahmed Geometry

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

Table 1. Experimental data (UWTSD, 2019)

A meshing strategy is implemented to refine successive meshes to establish an optimised solution which can be processed by the computer resource utilised in the study. The optimised solution is analysed in the post processor software where the pressure contours and velocity streamlines over the body rear slant and near wake area are considered.

2.0 Background

Computational fluid Dynamics (CFD) can be generally divided in to three separate processes, typically referred to as; Pre-Processing, Solving and Post-Processing.

2.1 Pre-Processing

Consists of domain and mesh type specification and generation. The design modeller and ICEM CFD mesh generator from within the ANSYS workbench complete the pre-processing function. The quality of the mesh being of critical importance if inaccurate results are to be avoided (Andersson, et al., 2012). Simplifying assumptions can be made to reduce the domain and mesh size, where the model is symmetrical only, one half would need to be considered, in turn reducing the computational capacity required.

2.2 Solving

Boundary conditions, fluid properties and turbulence model are specified at the solver stage. Fluent is utilised from ANSYS workbench to complete the solving process. The solving process utilises the finite volume method, where the Navier-Stokes general conservation equations for mass, momentum and energy are solved for each individual element (ANSYS, 2015). Where turbulent flows are solved the Navier-Stokes equations are not applied directly, but a turbulence model is utilised (Andersson, et al., 2012) The k-epsilon realisable model utilised extensively for turbulent flow problems. The overall solving process is determined by the convergence monitoring conditions, which specify the point where the solution is considered converged.

2.3 Post-Processing

The post-processing stage allows the solution to be analysed and quality of the result determined. The Results post processor being utilised. Pressure contours, streamlines and various other visual tools can be applied to the domain model to assess the solution results.

3.0 Methodology

3.1 Fluid Domain Geometry

The Ahmed body geometry is imported into the design modeler as a Parasolid model in the x_t. file format, the format allows manipulation of precision surface features found in computer generated simulation geometry.

The fluid domain is constructed within the design modeler by creating an extruded box around half of the Ahmed geometry running along the X-Y centre line plane, the box plan being defined as a sketch on the X-Y plane. As the flow downstream of the Ahmed body is expected to be turbulent in nature the sketch dimensions in the Y plane are constrained to the Y origin allowing independent reassignment of the dimensions upstream and downstream.

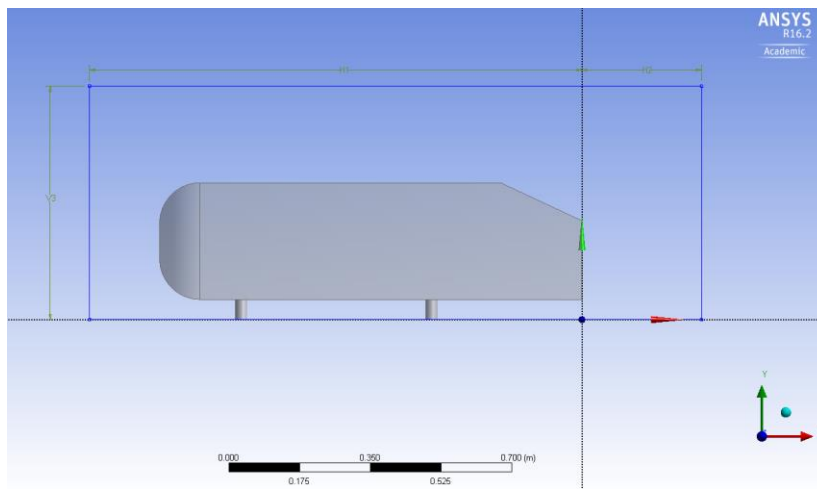


Figure 2. Ahmed body geometry imported into Design Modeler with domain box plan on X-Y plane.

The Ahmed body being subtracted from the box using a Boolean function to create the final fluid domain.

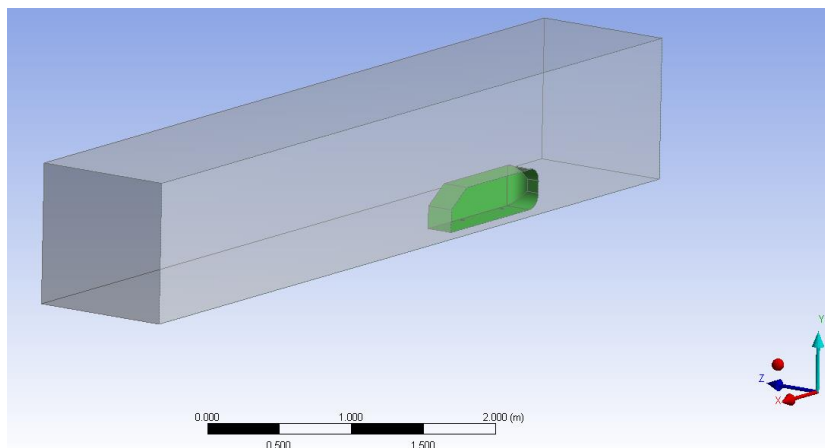


Figure 3. Final domain form with subtracted Ahmed body profile highlighted green.

The simulation conditions allowed a number of simplifying assumptions to be made with regards to the domain geometry which reflect the conditions under which the experimental validation data was recorded.

3.2 Meshing Quality

The quality of the generated mesh has a direct effect on the solver output, resulting in calculation convergence problems and poor interpretation of the physical problem. To ensure that the mesh generated is of an acceptable quality before uploading to the solver it is assessed for Skewness and Orthogonal Quality. Where the generated mesh is found outside of acceptable limits it is subject to refinement in the identified problem areas and re-assessed before uploading to the solver. Acceptable limits for maximum Skewness being < 0.95 , and minimum Orthogonal Quality > 0.1 .

3.3 Meshing Strategy

To ensure the optimum use of computing resources the meshing strategy will focus on refinement of the mesh in areas where high turbulence and pressure changes are expected, these are considered the near wake area to the rear slant and back face of the Ahmed body, the underbody space between the Ahmed body and the floor, and the immediate front face of the Ahmed body. Refinement of the mesh is managed through the introduction of density boxes with defined mesh size in identified areas.

The flow rate of 40 m/s over the Ahmed body equates to a turbulent Reynolds number of 2.8×10^6 , therefore the boundary layer thickness and corresponding mesh resolution in the boundary layer is considered of critical importance in establishing realistic simulation environments. The meshing strategy uses tetrahedral elements to fill the domain and considers the introduction of prism layers into the boundary layer as a refinement method.

3.4 Boundary Conditions

As the Ahmed body is simulated in a straight-ahead direction with no cross forces it is assumed that the flow around the Ahmed body is symmetrical about the X-Y plane, this allows one half of the Ahmed body and subsequent domain to be omitted from the simulation. By optimising the model in this way allows a significant amount of computational time and resources to be saved due to the reduction in the number of elements present within the domain. Lanfrit (2005) states that the modelling approach will be determined by the computer resources available. This is considered an important point in the study due to the simulations being undertaken on a desktop computer with the following specification: Processor – Intel® Core™ i7-7700 CPU 3.6 Ghz (4 available cores) RAM: 32.0 GB. The X-Y plane is assigned as a symmetry boundary condition.

The Ahmed body and floor are both considered as fixed solids within the domain and are assigned wall boundary conditions respectively. The domain side and top faces are assigned symmetry boundary conditions. As the study considers the simulation parameters the dimensions for the described faces with exception of the Ahmed body are reassigned as part of the study and are not specified in accordance with experimental wind tunnel parameters.

The inlet face boundary of the domain is assigned as a velocity inlet, with a velocity of 40 m/s in the X plane. The outlet face boundary of the domain is assigned as a pressure outlet, with a gauge pressure of 0 pa. A pressure outlet being defined over a velocity outlet for the domain outlet as better rates of convergence are achieved (Andersson, et al., 2012).

3.5 Turbulence Model

The turbulence model utilised for the initial solution and subsequent solutions during the domain and mesh refinement is *Realisable k-epsilon model* with *Standard Wall Functions*, being recommended for applications where separation and vortex shedding is present in the study of bluff bodies (ANSYS, 2015)

Further turbulence models are applied to the final refined mesh for evaluation, these being; RNG k-epsilon, Standard and SST k-Omega.

3.6 Solver Settings

The default solver settings are applied for simulations runs, being pressure based as the simulation is sub-sonic. An exception being the pressure-velocity coupling scheme being set to *Coupled* as this is preferred over *Simple* where convergency problems are encountered (ANSYS, 2015). The use of the coupled pressure-velocity coupling also being considered appropriate due to the computer resource available. Due to the time taken to complete each solution the maximum number of iterations considered is 400, as solutions for high domain element counts take in the region of an hour to complete.

4.0 Results

The affect of simulation parameters is investigated by running simulations with varying parameters and comparing results to experimental data. The first simulation run consists of a relatively coarse mesh and domain dimensions based on the assumption that the downstream flow is disrupted and turbulent due to the Ahmed body, the intention of the initial solution being to establish flow structure so that further refinement is applied in the correct areas of the domain in accordance with meshing strategy discussed.

4.1 Initial Mesh and Solution Run 1

An initial run is configured with minimum refinement of the domain and mesh to establish the mesh strategy.

The initial solution domain having overall dimensions of

- Length – 7 m
- Height – 2 m
- Depth – 2m

The initial maximum domain mesh being specified as – 0.2m

Initial number of elements within domain - 114553

A density box is applied about the Ahmed body (from the centroid) with dimensions increased by the following factors X - 2, Y - 1.5, Z – 1.5. The density box being offset in the X plane by 0.8m to include the near wake area of the Ahmed body. The maximum mesh size applied to the density box being – 0.05m

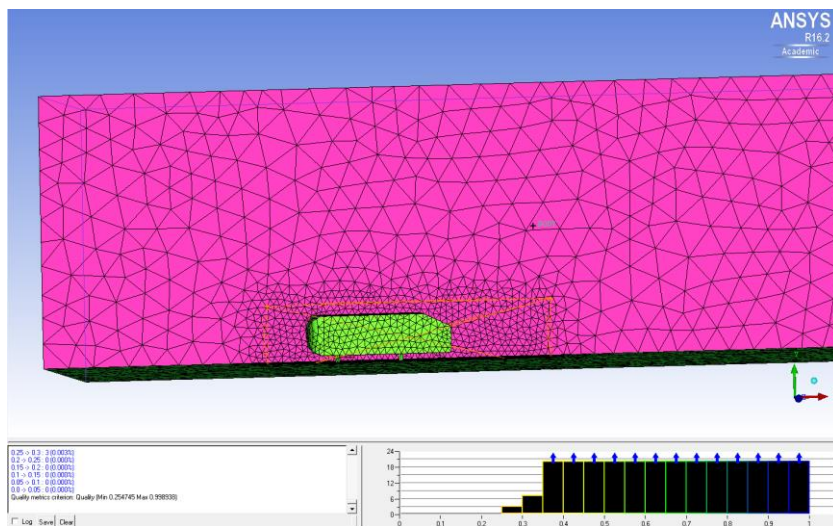


Figure 4. Initial (Run 1) domain mesh applied within the ICFM CFD meshing tool.

The initial solution converged after 87 iterations, with lift and coefficients of 0.338 and 0.419 recorded respectively.

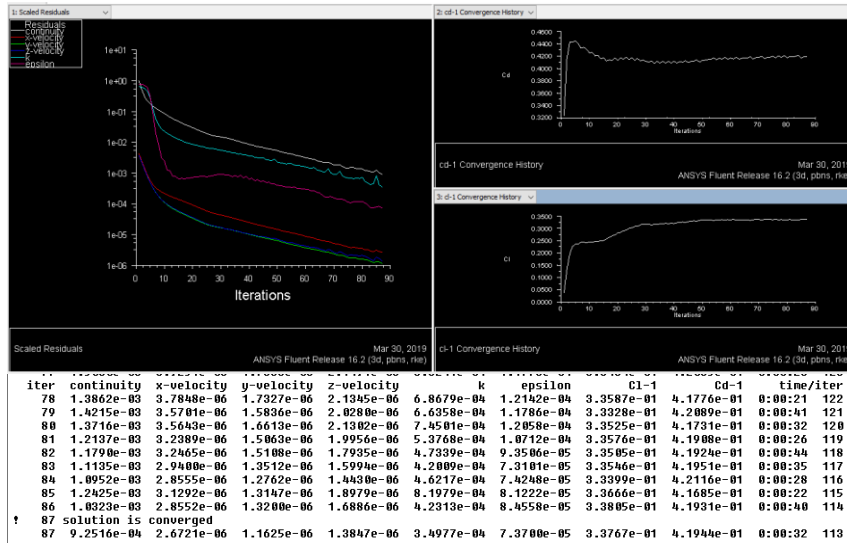


Figure 5. Initial (Run 1) solution convergence history

The coefficients for lift and drag on convergence for the initial solution were considerably outside of the figures stated from the experimental data (lift – 0.345, drag – 0.299), this being expected with a low-resolution domain mesh. However, a valuable graphical interpretation of the flow structure around the Ahmed body and in the near wake region can be utilised from the Fluent results graphics postprocessor for Pressure, Velocity Magnitude and Turbulent Kinetic Energy Distributions. Which will aid in determining the refinement meshing strategy for the following solution calculation runs.

The initial solution contour graphics for contours of velocity magnitude (figure 6) and contours of turbulent kinetic energy (Figure 7) describe the areas where the greatest disturbances are present as expected in the near wake region and underbody floor region of the Ahmed body, despite the fact that the domain mesh is of a low resolution. These areas therefore will be the focus of subsequent runs in the development of the domain mesh in establishing a simulation model which reflects the experimental data.

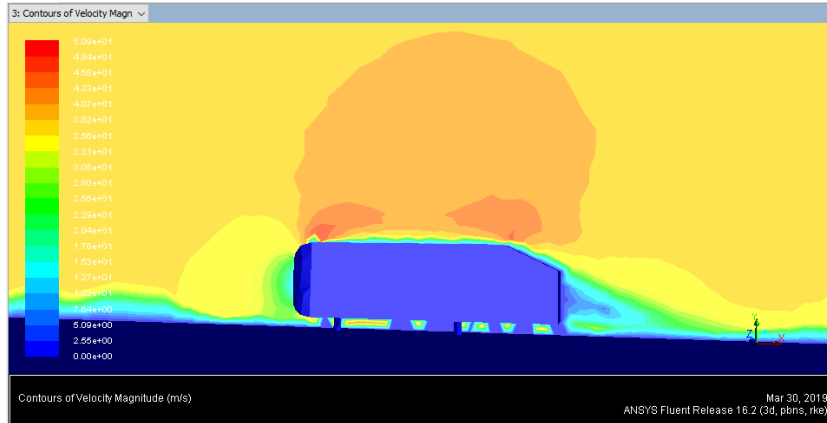


Figure 6. Initial (Run 1) Solution – Contours of Velocity Magnitude

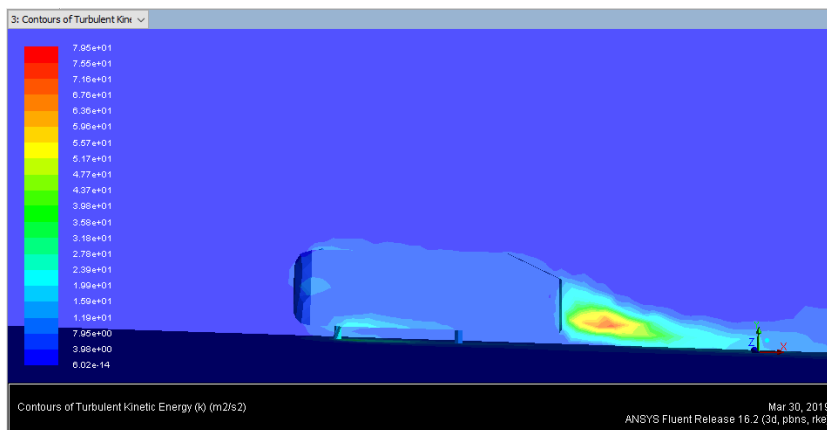


Figure 7. Initial (Run 1) Solution – Contours of Turbulent Kinetic Energy

4.2 Progressive Refinement of Domain and Mesh

For the following refinement simulation runs utilising the realisable k-epsilon turbulence model the domain size was set in accordance with the recommendation suggested by Lanfrit (2005), where the dimensions of the domain forward of the Ahmed body are set at 3 body lengths, and to the rear 5 body lengths. The displacement of the Ahmed body in the X face direction where the length and breadth of the domain was retained at 2m in each direction respectively was found to be 1.44%, within the range suggested by Lanfrit (2015) at 1-1.5%.

By retaining the domain dimensions as described, the study focuses on the effect of mesh strategy on the simulation results and those of coefficients of lift and drag. The mesh parameters subject to variation being density in areas of disturbance, and the introduction of prism layers. The summary of the progressive refinement simulation runs is described in table 2. The following paragraphs discuss the observations noted and refinements made.

Ansys - Ahmed 25 @ 40m/s				
Run	Lift	Drag	Converged	Prism layers
1	0.338	0.419	87	0
2	0.245	0.413	87	0
3	0.412	0.361	104	0
4	0.342	0.356	110	0
5	0.325	0.284	131	0
6	0.363	0.303	127	3
7	0.362	0.31	164	3
8	0.367	0.31	171	3
9	N/A	N/A	NC@400	5
10	0.354	0.292	233	7
11	0.357	0.287	239	7
12	0.341	0.287	255	7

Table 2. Results of simulation with progressive refinement

4.3 Simulation Run 2 and 3

Run 2 consisted of the same mesh sizes as run 1 however within the domain dimension suggested by Lanfrit, run 3 refines run 2 by the improvement of the resolution of the density box mesh to 0.03m. The mesh quality for both runs remaining within acceptable limits. An improvement in the structure of the turbulent flow was noted in the near wake area at the rear of the Ahmed body which is shown in figure 8, detailing contours of velocity magnitude during run 3, further supporting the increase of mesh resolution within the density box covering the near wake area, when compared to figure 6 for run 1.

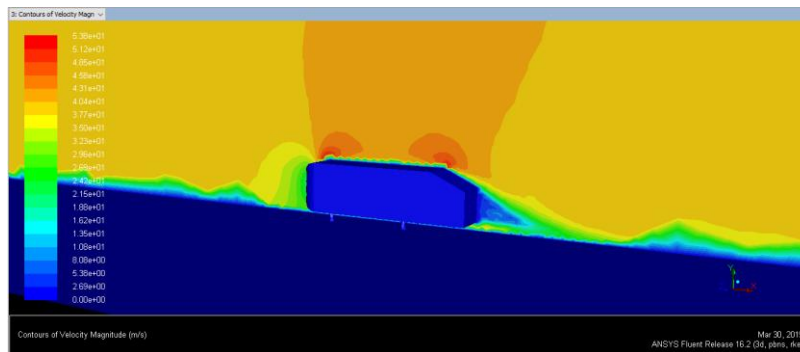


Figure 8. Run 3 Solution - Contours of Velocity Magnitude

4.4 Simulation Run 4

Mesh resolution for the domain and Ahmed body density box remain as run 3, however the dimensions of the density box are specified in line with recommendation made by Lanfrit (2015), where dimensions are in terms of the body, 0.5 body forward, 0.5 body top, 0.5 body side, 1 body forward. Figures 9 and 10 describe the extent of the density box applied to the Ahmed body.



Figure 9. Density box viewed from the front face of the Ahmed body.

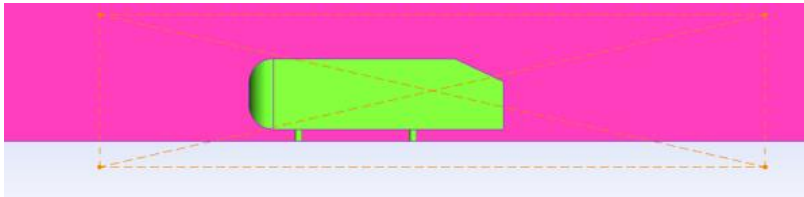


Figure 10. Density box viewed from the X-Y face of the Ahmed body.

The mesh quality remains consistent with earlier runs within acceptable limits. The increase in size of the density box allows the flow disruption to be observed as in previous runs but with no further improvement of detail about the Ahmed body surfaces, as the flow is turbulent further refinement is required in the boundary layer area of the near wall surfaces.

4.5 Simulation Run 5

The surface mesh element size is increased in line with recommendations made by Lanfrit (2015). Due to the computer resource available a medium mesh resolution is considered with an element count for the domain size in up to 5 million elements, this correlates to a y^+ value in the range of 150.

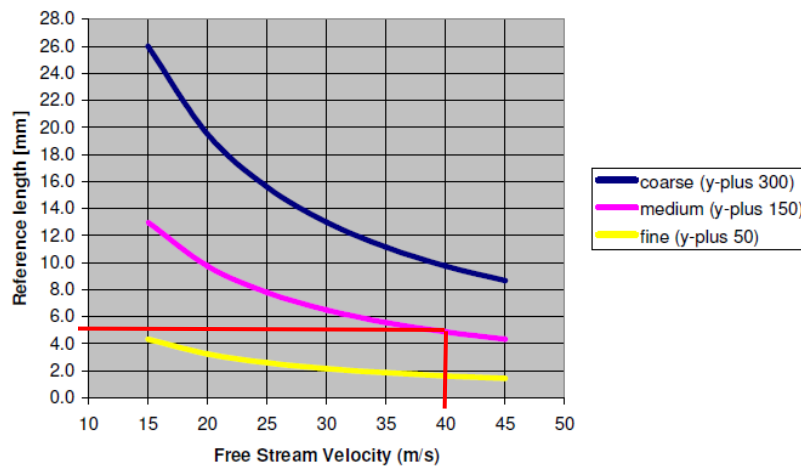


Figure 11. Estimation of body surface mesh length (Adapted from Best practice guidelines for handling Automotive External Aerodynamics with fluent. Lanfrit, 2015).

The estimated surface mesh length obtained from Lanfrit's graph is approximately 0.005m for a medium mesh with a fluid velocity of 40 m/s.

The maximum surface mesh size of 0.005m is applied to the Ahmed body, the domain and density box remain as run 4. The element count within the domain increases to 1196390, mesh quality was found to be within acceptable limits.

4.6 Simulation Run 6, 7 & 8 – 3 Prism layers

As the flow over the Ahmed body is turbulent, the viscous sublayer in the near wall area is of significant importance in establishing realistic models that reflect experimental conditions. The area of the viscous sublayer is covered by prism layers from the body wall, through the viscous sublayer into the tetrahedral elements of the domain mesh.

The size of the first prism layer is important to ensure that the desired Y^+ value is near to the desired ($Y^+ = 150$)

From the equation that defines y^+ the approximate height of the first prism layer can be calculated

$$y^+ = \frac{\rho U_\tau y}{\mu} \rightarrow y = \frac{y^+ \mu}{U_\tau \rho}$$

Taking the velocity and Reynolds number specified for the simulation, standard sea level conditions for density and dynamic viscosity of air. The value of the first prism layer is approximately 0.0014m.

Run 6 was completed with no height control over the prism layers, however the cell quality was noted to deteriorate in the way of the support legs of the Ahmed body. Height control was introduced to the prism layers for Run 7, however there was no improvement noted in the cell quality in the area of the support legs. Figure 12 details the mesh quality report for Run 7.

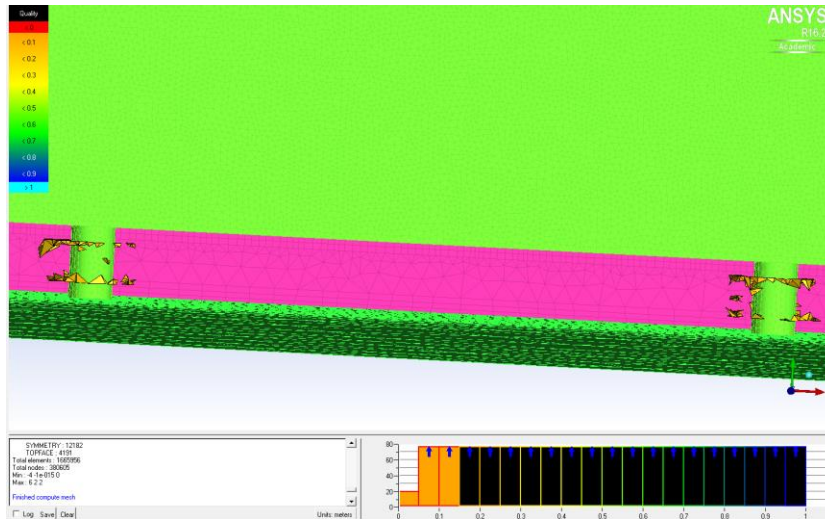


Figure 12. Detail of Run 7 cell quality report, with poor cells highlighted in support area.

The domain mesh was modified for Run 8 to include a second density box with a size factor of 1.2 of that of the Ahmed body, the maximum mesh size within the second density box was specified as 0.01m. The original density box maximum mesh size was reduced to 0.02m, the maximum mesh size to the Ahmed body faces remained at 0.005m. Figure 13 details the arrangement of density boxes for run 8.

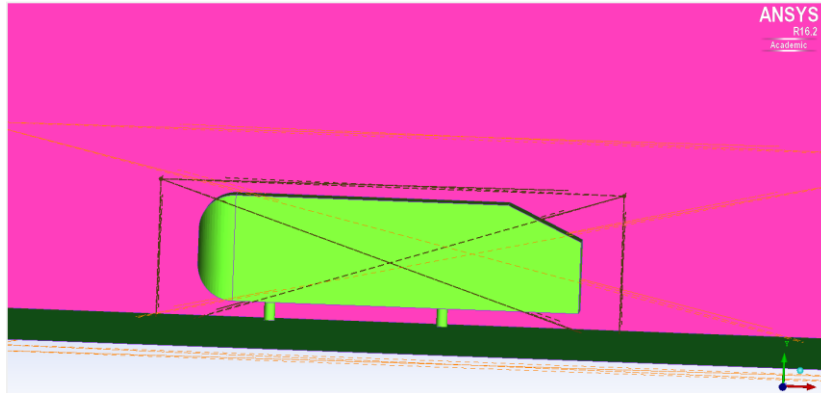


Figure 13. Run 8, second density box (brown) added in addition to first density box (Orange).

Considerable improvement in cell quality was found in the way of the support legs by increasing the resolution of the cells in the area between the lower of the Ahmed body and floor when compared to run 7. However, the total cell count within the domain had increased to 1783626.

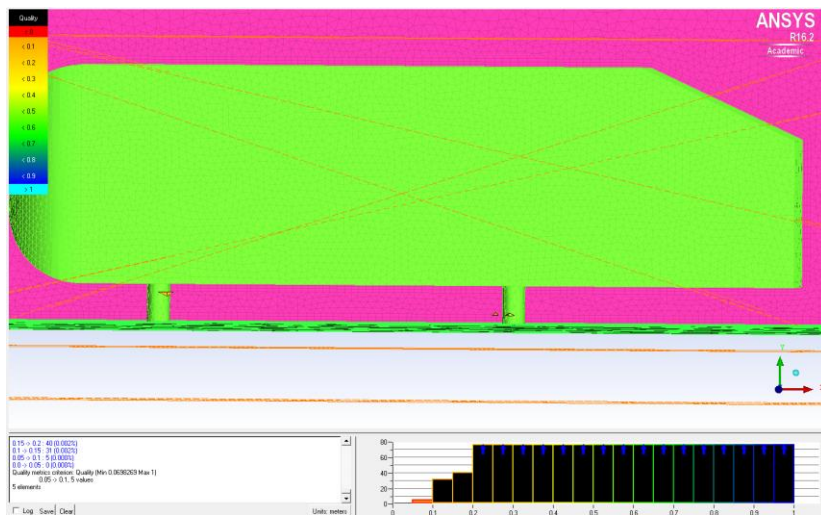


Figure 14. Detail of Run 8 cell quality report.

The solution for run 8 converged after 171 iterations, with coefficients of lift and drag reported as 0.367 and 0.310 respectively.

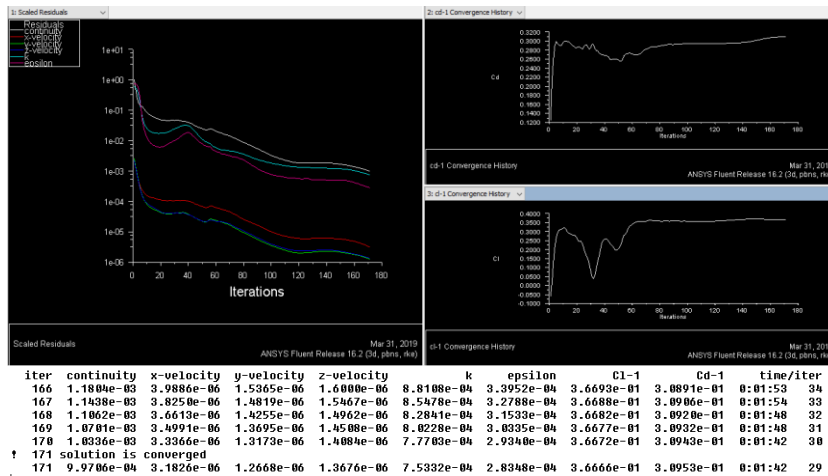


Figure 15. Run 8 Solution convergence history.

The increase in cell count within the domain and specification of maximum cell size to the Ahmed body and density boxes has resulted in significant improvement in the resolution of the contours of velocity magnitude plots. Areas of high velocity are clearly visible from the change in section on the front of the Ahmed body to the top and bottom and the rear where transferring to the slant. The near wake turbulence structure has clear definition at the rear of the body.

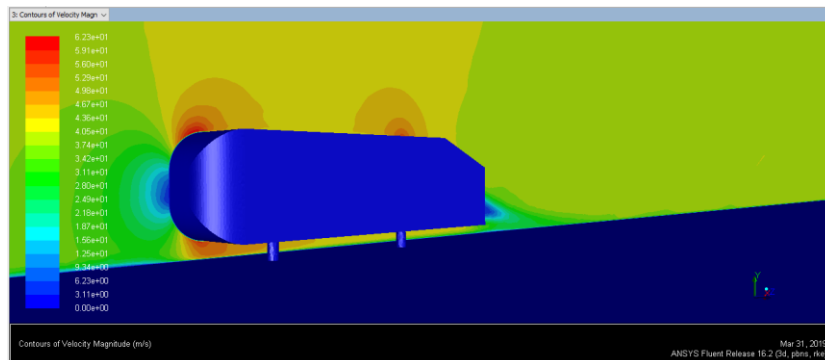


Figure 16. Run 8 Solution - Contours of Velocity Magnitude – Ahmed body forward

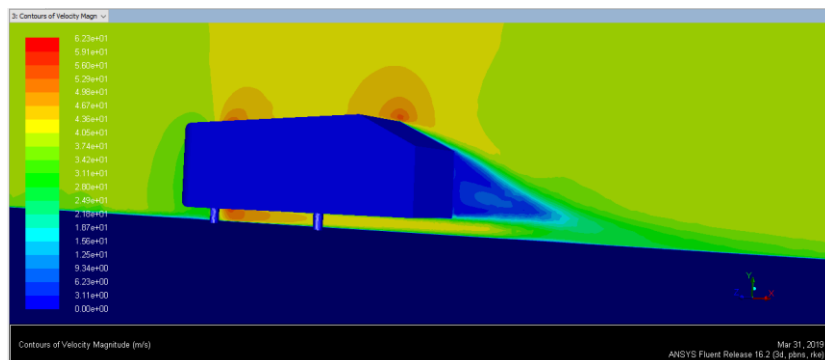


Figure 17. Run 8 Solution - Contours of Velocity Magnitude – Ahmed body rear

4.7 Simulation Run 9 – 5 Prism layers

The domain mesh of run 8 was refined for run 9 to include 5 prism layers, however it was found that the solution had not converged after 400 iterations. No further amendments were made to the prism layer height or mesh density. Further investigation of the effects on coefficients was abandoned in favour of progressing with investigation of 7 prism layers.

4.8 Simulation Run 10, 11 & 12 – 7 Prism layers

The domain mesh of run 8 was refined for run 10 to include 7 prism layers, the solution was found to converge and have coefficients that were generally in agreement with Run 8. However the mesh quality again was noted to be poor in the support leg area.

For run 11 a further density box, box 3 was added in the Ahmed body to floor area with a maximum mesh size of 0.005m applied which aligns with the maximum mesh size of the Ahmed body surfaces at 0.005m. Density box 2 was increased by a factor of 1.5 in the x direction due to increased resolution of the near wake turbulence noted during run 8.

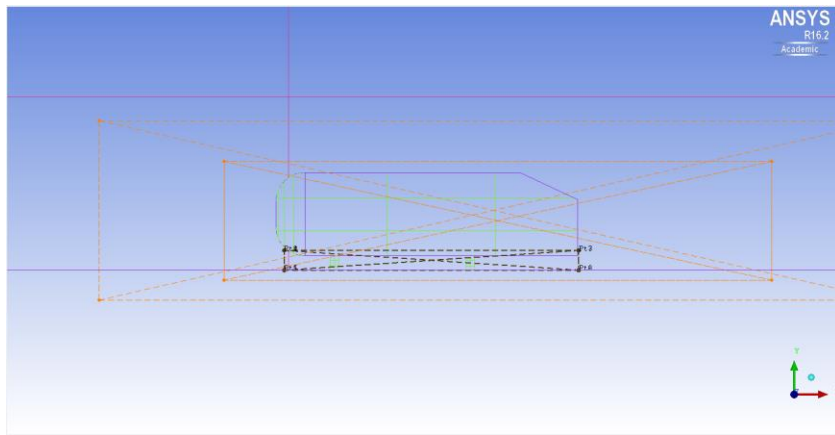


Figure 18. Density box 3 added as part of run 11, in the Ahmed body to floor area.

The total cell count in the domain increasing to 3977132. Run 11 convergence and reported coefficients were found to be in good agreement with run 10. However, the total cell count in the domain for run 10 was 2579909, which would suggest that the solution is becoming mesh independent. Figures 19 and 20 depict the solution convergence history for runs 10 and 11 respectively. Comparison of the respective convergence history tends to support mesh independence.

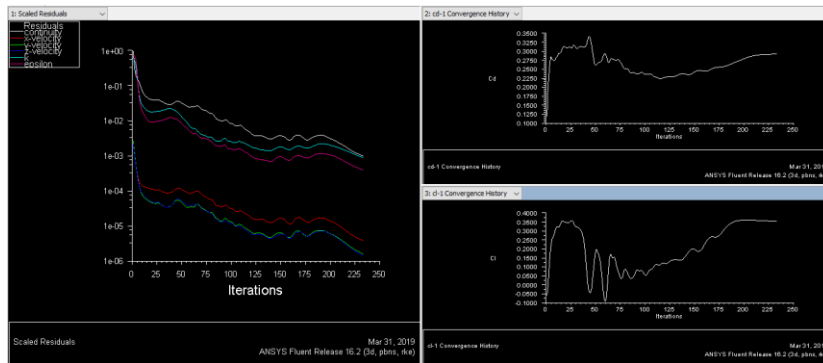


Figure 19. Run 10 Solution convergence history.

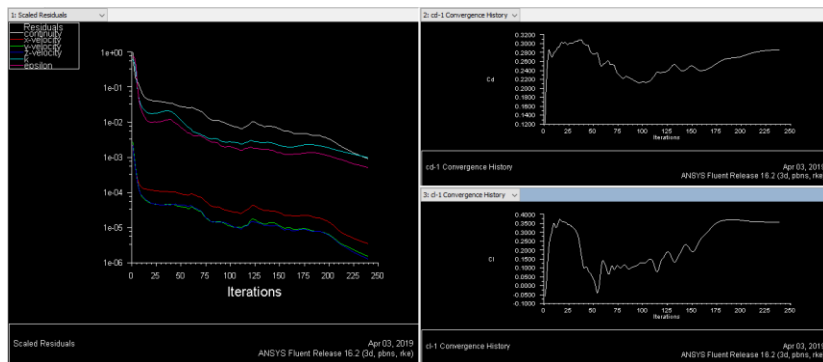


Figure 20. Run 11 Solution convergence history.

Review of the contours of static pressure and velocity magnitude for run 11 indicated that a pressure gradient and possible flow separation is present on the Ahmed body rear slope, therefore run 12 incorporates an additional density box in the way of the rear slope with a maximum mesh size of 0.005m. The total cell count in the domain for run 12 increasing to 4793496. Comparison of run 12 convergence history with that of run 10 and 11 show good agreement. The total cell count aligns with the medium resolution mesh (y^+ 150) discussed by Lanfrit, which the dimensions of the simulation domain are based.

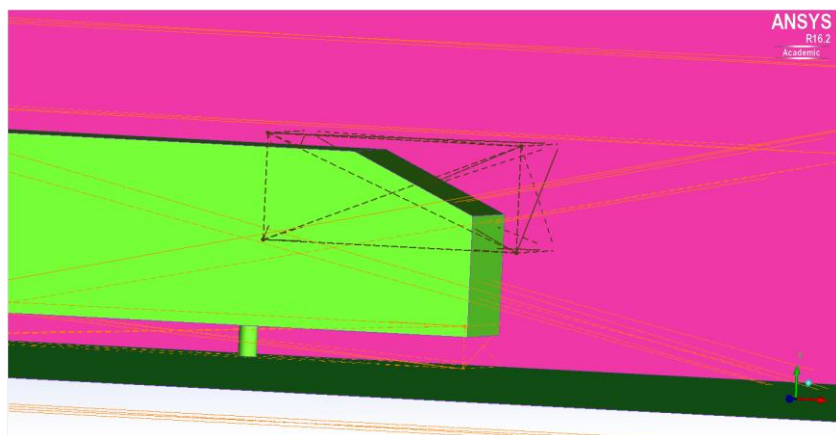


Figure 21. Density box 4 added as part of run 12, in the Ahmed body rear slant area.

Run 12 was found to converge after 255 iterations with recorded coefficients of lift and drag at 0.341 and 0.287 respectively. An improvement in the lift coefficient was noted over run 11 with the addition of the rear slant density box. Review of the contours of static pressure confirm that a pressure gradient is present on the slant surface.

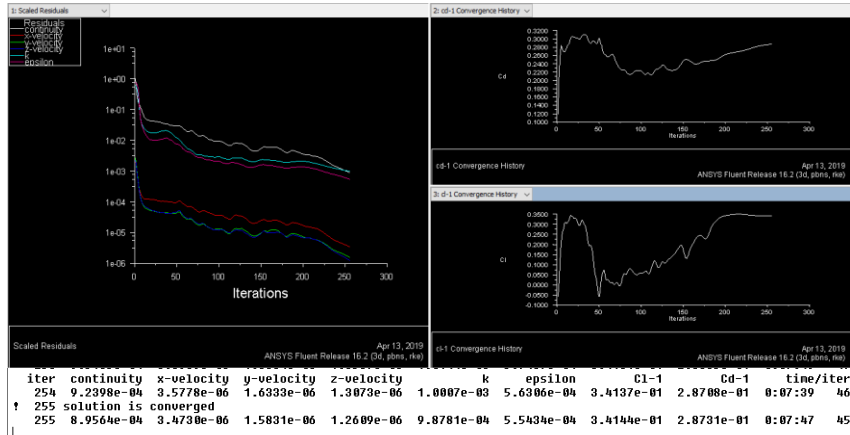


Figure 22. Run 12 Solution convergence history.

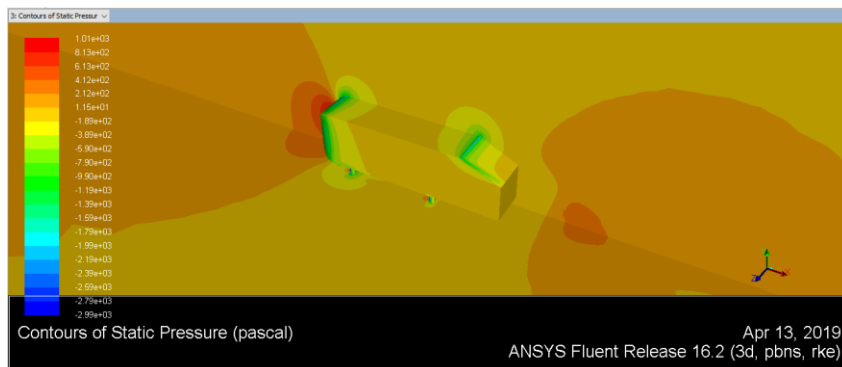


Figure 23. Run 12 Solution - Contours of Static Pressure – Ahmed body rear

4.9 Simulation Run 12 – Evaluation of Turbulence models

The solution of run 12 are considered in good agreement with the experimental results for coefficients of lift and drag, therefore run 12 is taken forward for further analysis using post processing techniques.

The solution of run 12 was also further modified to evaluate other turbulence models, RNG k-epsilon, Standard and SST k-Omega. In all cases no convergence of the solution was reached within 400 iterations, therefore further investigation was abandoned.

4.10 Postprocessing – Simulation Run 12

The solution data from fluent is passed through to the postprocessor module, Results.

By applying a velocity streamline plane to the symmetry plane two recirculatory flow regions in the near wake structure are visible, which show good agreement with the recirculatory flow described by Ahmed, et al. (1984) in the same location, (wake schematic diagram figure 27). The separation bubble also discussed is seen to develop in the region between the lower recirculatory region and floor.

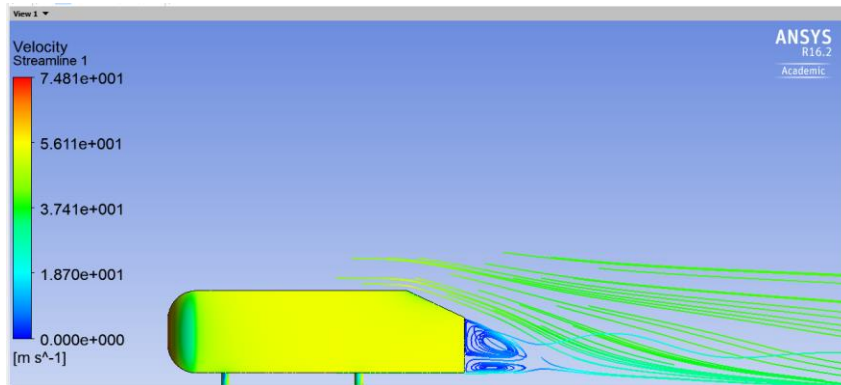


Figure 24. Simulation Run 12 – Postprocessing – Velocity streamlines in the near wake region

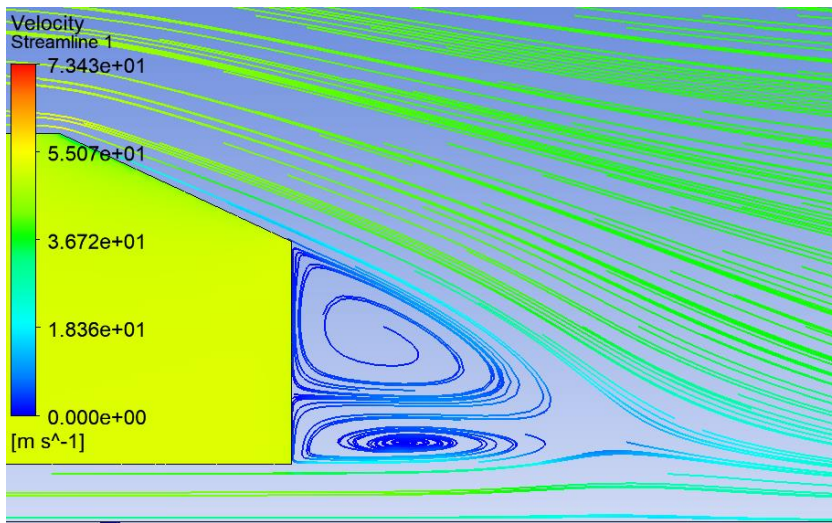


Figure 25. Simulation Run 12 – Postprocessing – Recirculatory flow regions and separation bubble

The streamline plane is moved in 0.1945m in the Z axis to align with the vertical edge of the Ahmed body, the development of the vortex described by Ahmed is seen to form from the slant edge.

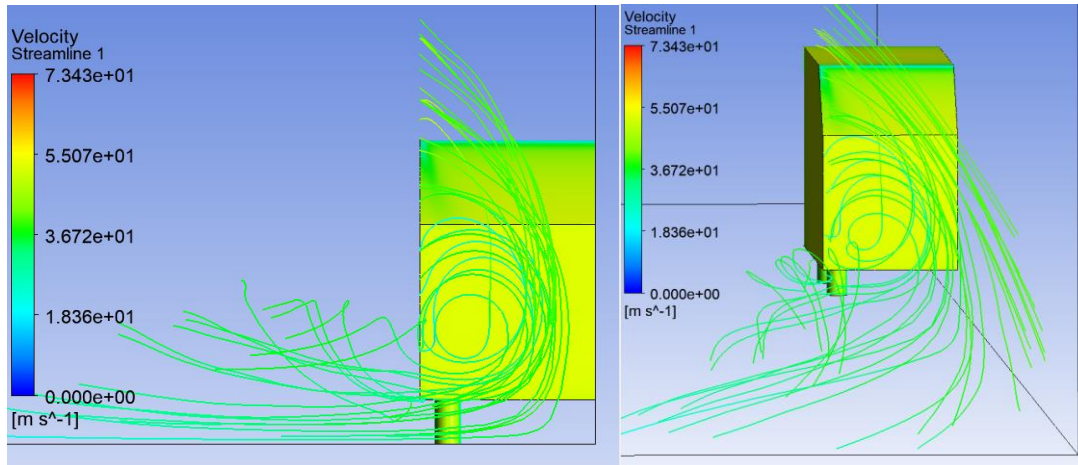


Figure 26. Simulation Run 12 – Postprocessing – Formation of wake trailing vortex

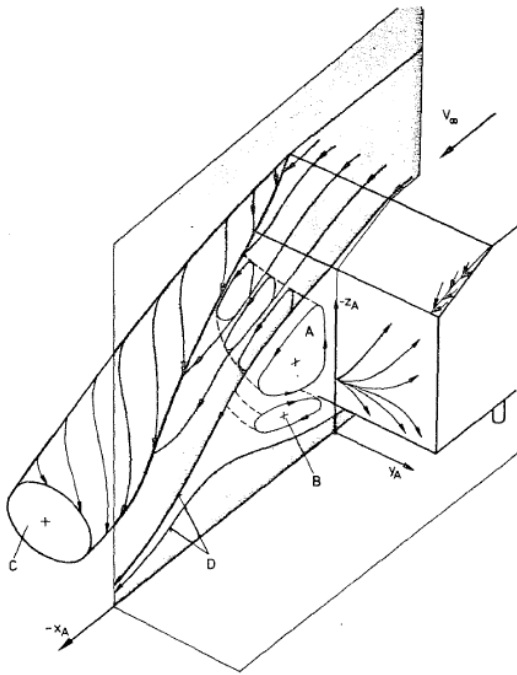


Figure 27. Horseshoe vortex system in wake (Ahmed, et al., 1984)

By applying a vortex core region overlay to the Ahmed body, the formation of the near wake recirculatory regions and trailing vortex are clearly seen. The separation of the vortex structure from the slant face showed similar characteristic to that presented in a study by Guilmineau (2018) where the computation solution was derived from a domain mesh containing 23.1×10^6 cells, suggesting further that the solutions obtained for run 12 are accurate for the medium resolution mesh and domain utilised in the study as recommended by Lanfrit (2015).

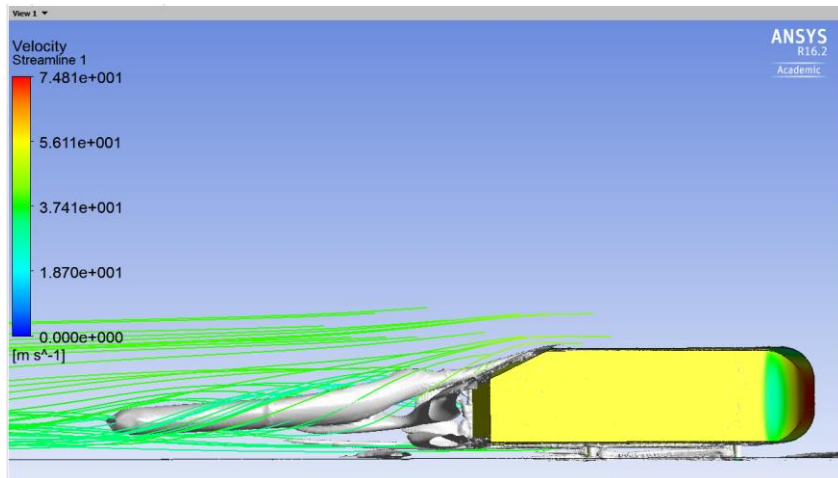


Figure 28. Simulation Run 12 – Postprocessing – Vortex core region overlay, recirculatory flow regions

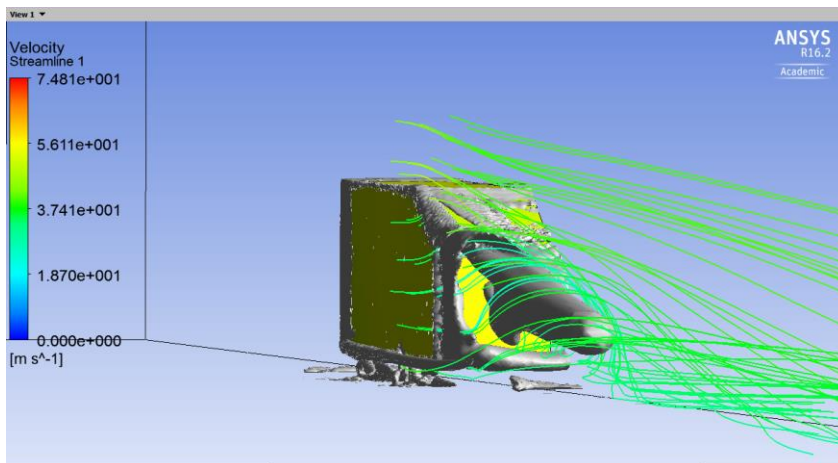


Figure 29. Simulation Run 12 – Postprocessing – Vortex core region overlay, trailing vortex

Pressure distribution contours of the slant face (figure 30) identify lower pressure areas where flow separation and the trailing vortex is noted to form and start to recirculate as shown in the streamline plane, the lower pressure areas increasing drag respectively. The distribution was found to be similar to those presented by Lienhart and Becker (2003) and also Keating, et al. (2008).

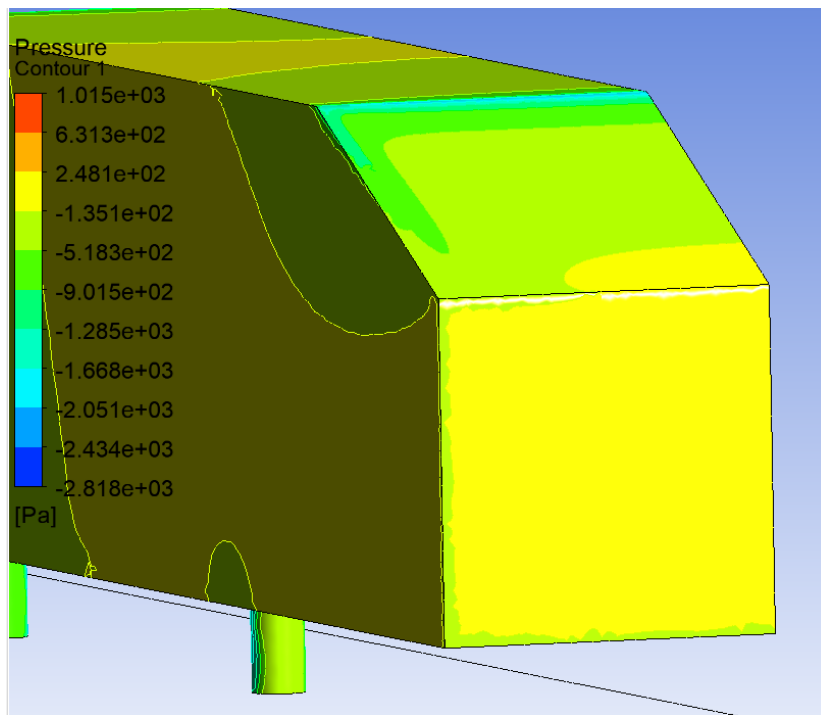


Figure 30. Simulation Run 12 – Postprocessing – Pressure contour – Ahmed body rear slant

5.0 Conclusion

The simulated results obtained for the Ahmed body utilising the ANSYS Workbench R16.2 Academic software were in good agreement with the experimental result. Where the calculated solution coefficients for lift and drag are 0.341 and 0.287 respectively, compared to experimental results of 0.345 and 0.299. The pressure contours and streamlines applied during postprocessing were comparable with results obtain by literature review.

Computing resource should be considered at an early stage in the design of a fluid domain and mesh, as the number of elements in a domain can quickly exceed what can be viably processed, the resolution and location of fine mesh density boxes should be considered carefully and supplemented or replaced with prism layers where possible. The use of prism layers in the viscous sub-layer provided considerable refinement of the solution. Mesh quality was found to be affected where prism layers were introduced around small features such as the body supporting legs, further refinement of the mesh is required in these areas to address the quality.

The study has confirmed that refinement of the domain and mesh by the application of a sound strategy is critical in obtaining reliable results, application without due consideration is likely to result in much uncertainty.

6.0 References

Ahmed, S. R., Ramm, G. & Faltin, G., 1984. *Some Salient Features Of The time-Averaged Ground Vehicle Wake*, Detroit: Society of Automotive Engineers, Inc.

Andersson, B., Andersson, R., Hakansson, L., Mortensen, M., Sudiyo, R. & van Wachem, B., 2012. *Computational Fluid Dynamics for Engineers*. In: 1st ed. Cambridge, UK: Cambridge University Press.

ANSYS, 2015. *Lecture 2: Introduction to the CFD Methodology*, s.l.: ANSYS.

Guilmineau, E., 2018. *Effects of Rear Slant Angles on the Flow Characteristics of the Ahmed Body by IDDES Simulation*, Detroit: Society of Automotive Engineers, Inc.

Keating, A., Shock, R. & Chen, H., 2008. *Lattice Boltzmann Simulations of the Unsteady Flow Behind the Ahmed Body*, Detroit: Society of Automotive Engineers, Inc.

Lanfrit, M., 2005. *Best practice guidelines for handling Automotive External Aerodynamics with FLUENT*. 1.2 ed. Darmstadt, Germany: Fluent Deutschland GmbH.

Lienhart, H. & Becker, S., 2003. *Flow and Turbulence Structure in the Wake of a Simplified Car model*, Detroit: Society of Automotive Engineers, Inc.