ANSYS Mesh Refinement Study

EGH423 – Fluid Dynamics

CFD Assignment Part – 1

Due 10/09/2021

Group members	Student Number	Contribution	Signature
Finbar Delbridge	**N9717854**	**50%**	
Jacob Garcia-Pavy	**N10012478**	**50%**	

1.0 Introduction

This report aims to simulate laminar flow around an assigned bluff body in ANSYS utilising meshing methods and refinement strategies to analyse converged and accurate data. The use of CFD software such as ANSYS helps to conduct a mesh refinement study to produce results that will justify the selection of the most appropriate grid size for mesh refinement. The specified variables for this simulation include an inlet velocity of 0.001m/s, default pressure outlet and no slip stationary walls. The laminar flow consists of water-liquid with a density of 998.2kg/m^3 and a viscosity of 0.001003kg/m-s. The defined simulation parameters include a residual target of 1*10^-6 and an iteration limit of 5000. The presented problem is shown in figure [1] below with the assigned h and L values respectively 10mm and 40mm.
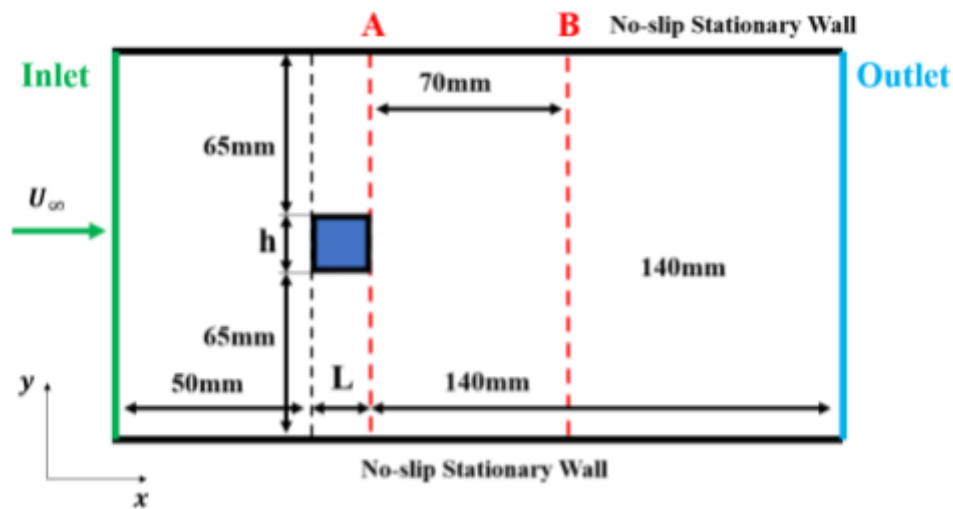

Figure 1, Assigned Problem

2.0 Geometry created in Design Modeler

Before a mesh refinement study happens first a geometry needs to be created in design modeler. Here, everything is dimensioned, and the parameters set with assigned values.
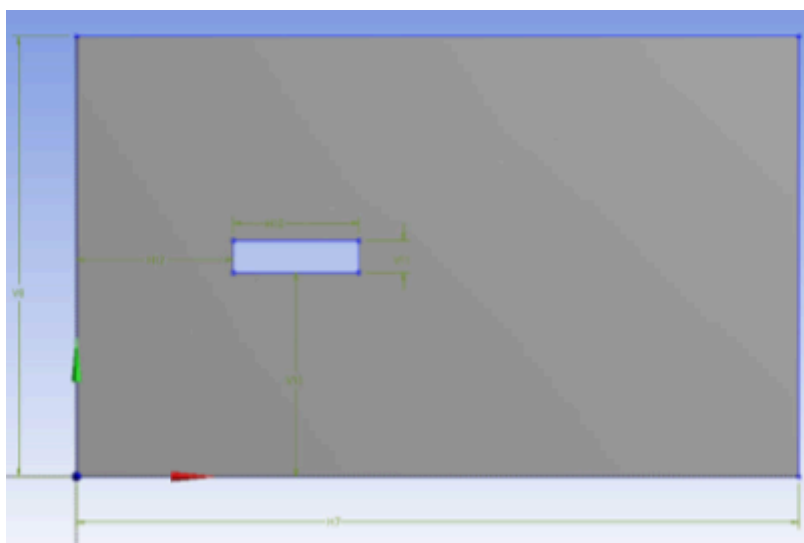

Figure 2, Geometry

Figure 3, Assigned Dimensions

3.0 Mesh generation type and refinement strategy:

All ANSYS simulations require a quality converged mesh to ensure accuracy in as little computational time possible. To do this the body could be triangular meshed which has no restrictions in shapes or pattern making it easy to mesh a quality 2D surface. Alternatively, the body could be split into separate faces defining the wake and key regions of the flow allowing consistent quadrilateral meshing and better representation of significant data. Moreover, quadrilaterals create less elements and greater uniformity, overall producing the better mesh. However, due to the nature of this simulation being laminar flow, the triangular method can calculate all the significant values surrounding the body with enough bias towards the body. This method was optimised by adjusting the edge sizing along the body by increasing the number of divisions each time. While the face was consistently refined by making the element size smaller. To ensure convergence, the refinement strategy consisted of scaling the number of divisions and element size by a factor of 2 seen in table 1 and demonstrated in figures 4-10.

***Note: the mesh arrangement seen across figure 4-10 is carried over in the following figures 11, 12 and 15.

Mesh Refinement

| Mesh	No. Divisions	Element Size	No. Elements	CPU Time (S)	Drag Coefficient	Drag Difference	Formation Length (M)
1	25	40	2442	0.694	5.77E+00	0	0.093243
2	50	20	3914	0.835	5.72E+00	-4.22E-02	0.093474
3	100	10	6834	1.529	5.80E+00	7.81E-02	0.093704
4	200	5	14036	1.551	5.81E+00	9.80E-03	0.093704
5	400	2.5	32247	6.017	5.81E+00	2.30E-03	0.093704
6	800	1.25	83089	1906.332	5.82E+00	8.60E-03	0.093704

Table 1: Mesh refinement over 6 meshes.
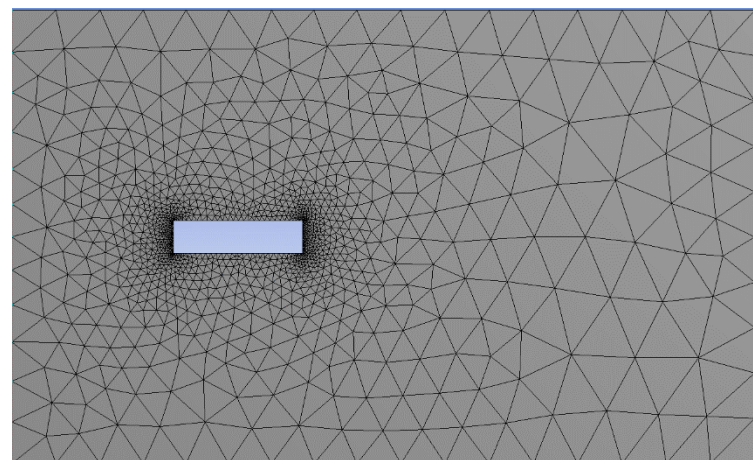
Figure 4: Mesh 1
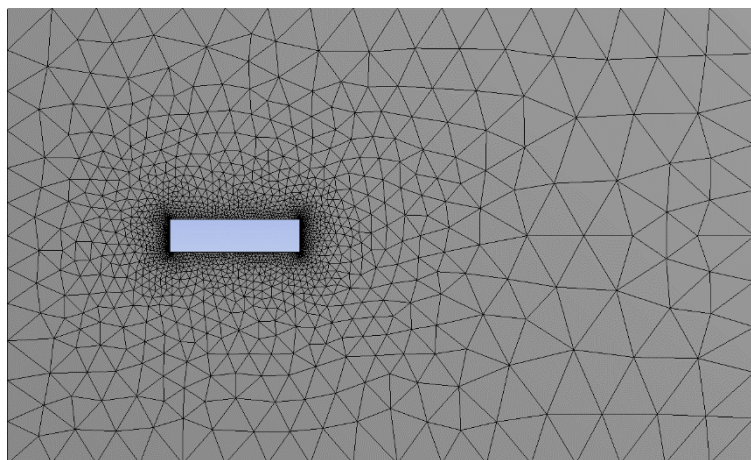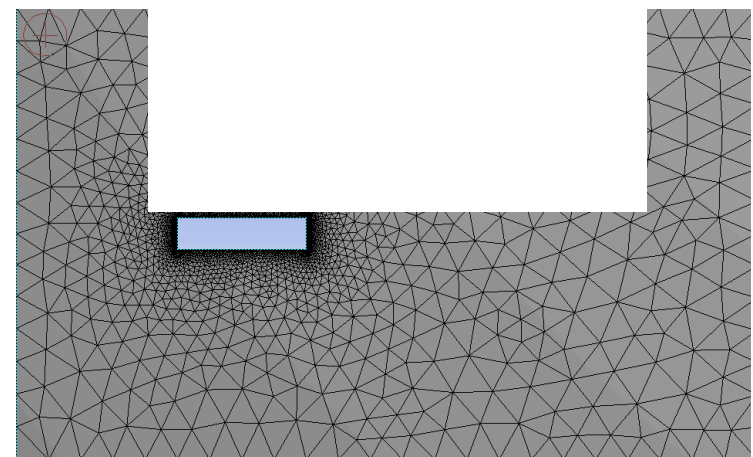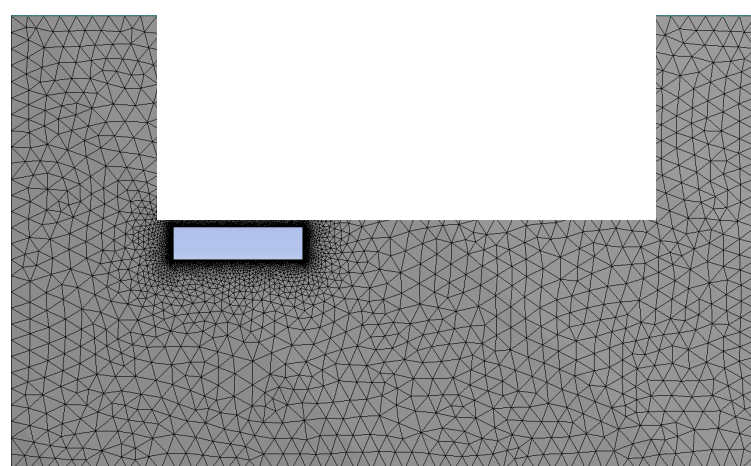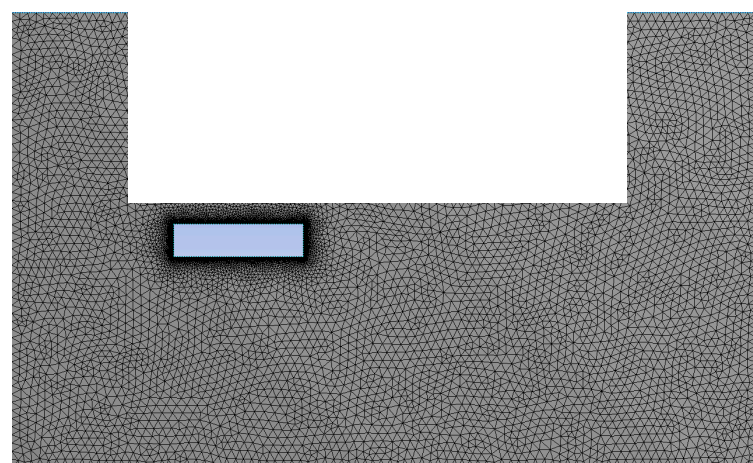

Figure 5: Mesh 2
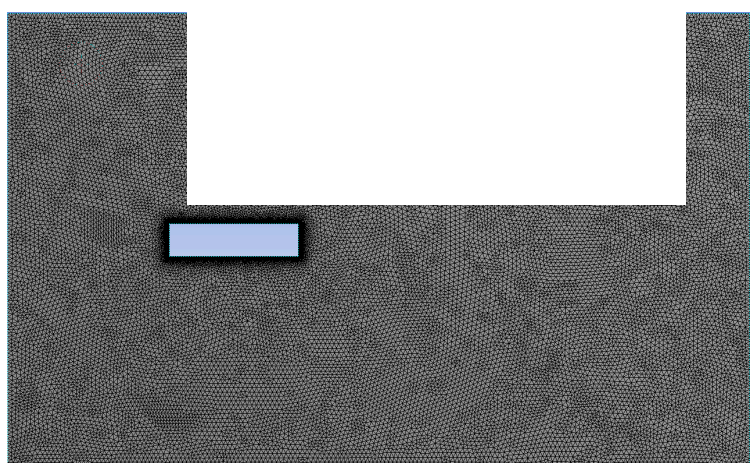

Figure 6: Mesh 3

Figure 7: Mesh 4

Figure 8: Mesh 5

Figure 9: Mesh 6

4.0 Mesh Refinement Study Results

    4.1 Contour plots for the velocity field for each grid size:

The figures below show the fluid velocity with the different meshes and how the bluff body affects this fluid flow. The red colour shows that the fluid has a high velocity while the dark blue shows basically no velocity.
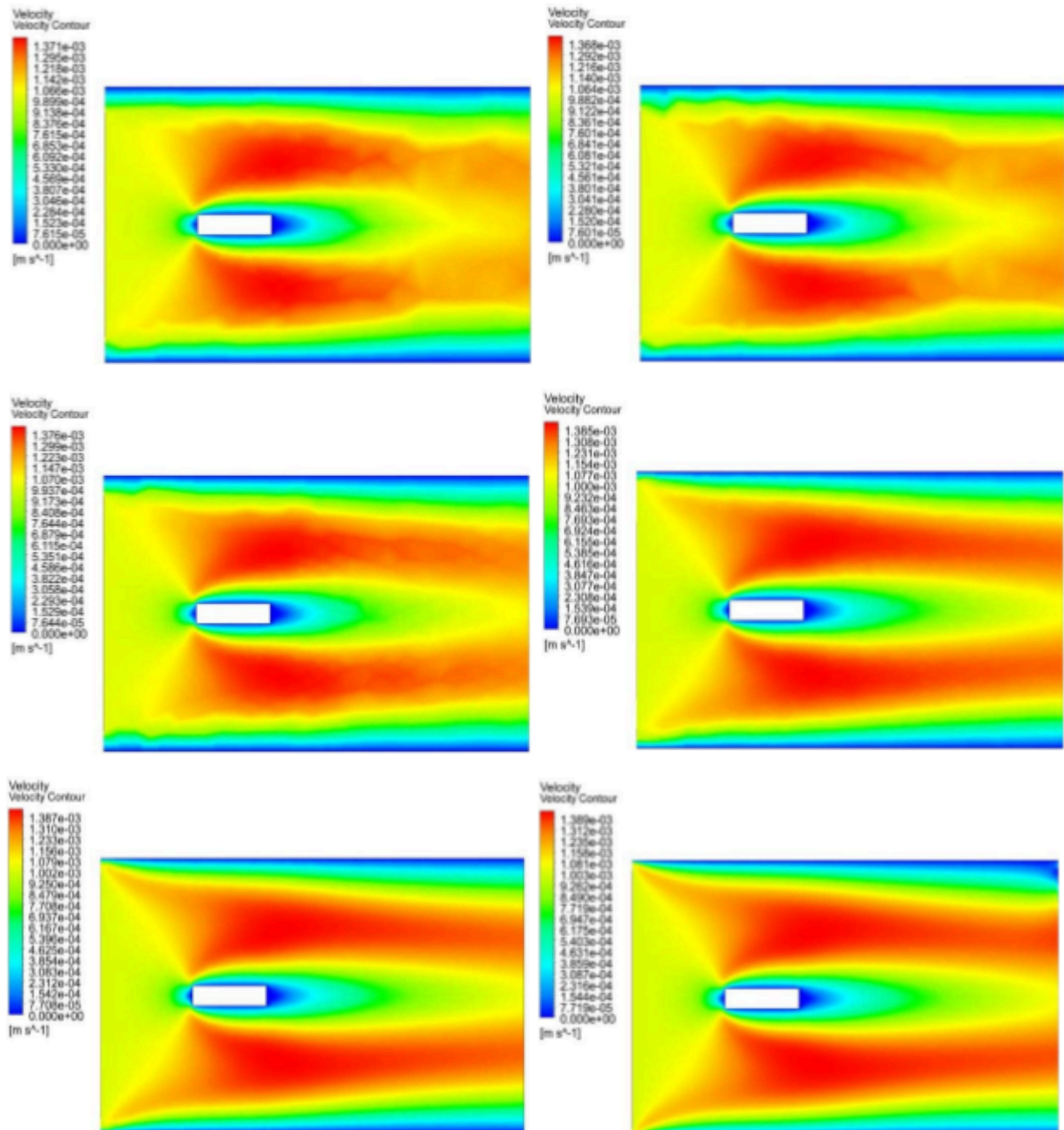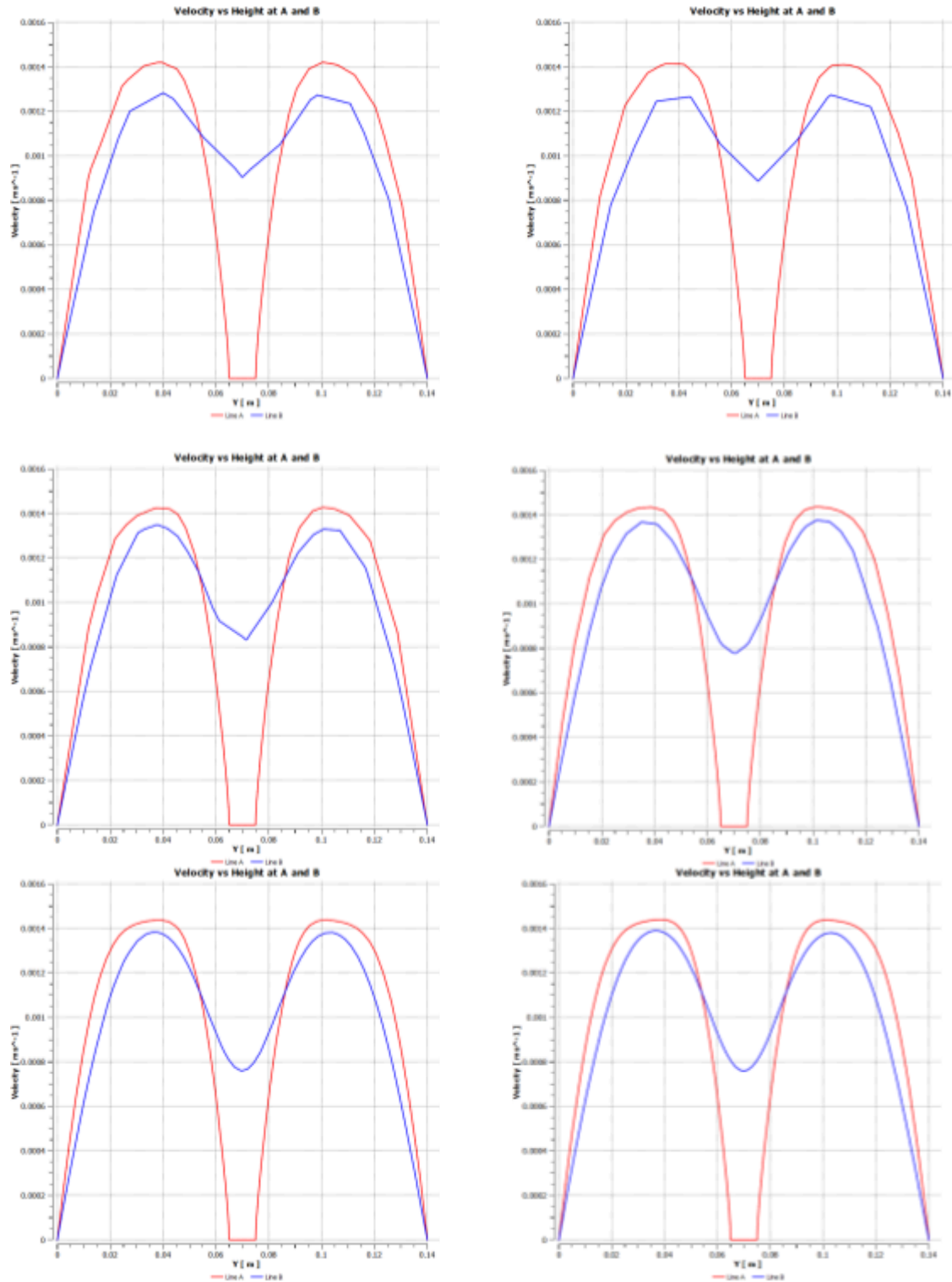

Figure 10: Contour plots for velocity field

4.2 Plot of velocity profiles at line A and B:

These graphs show velocity profiles taken at two different vertical points behind the bluff body. The red line represents line A and the blue line represents line B.

drag   *Figure 11: Pot of velocity profiles of line A and B*   **4.3 Tabulated coefficient and plot**

against the number of grid elements/nodes:

Here the drag coefficient and drag different are plotted against the number of elements. The dots on the line graph represent the mesh type with the greatest number of elements being the highest refined mesh.
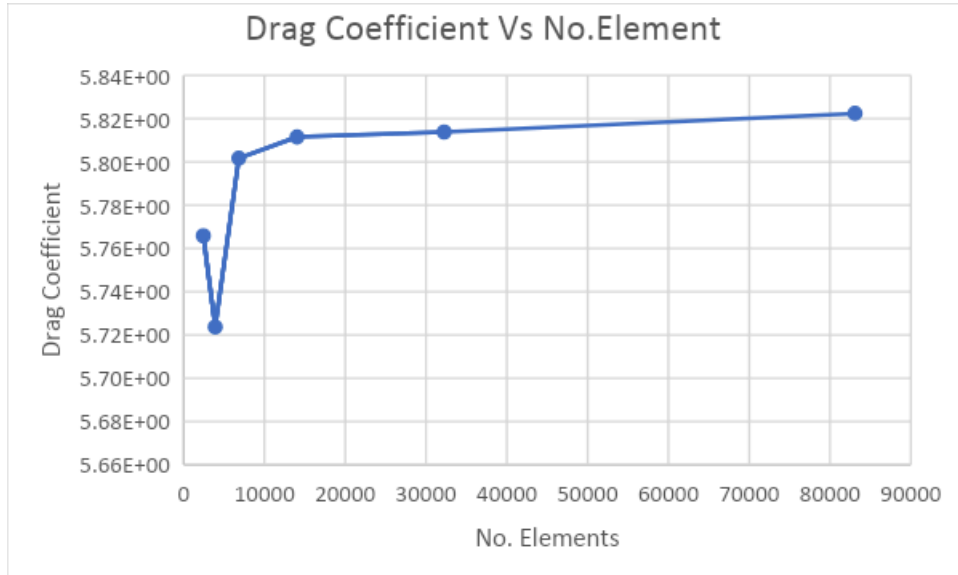

Figure 12: Drag coefficient Vs. no. Elements
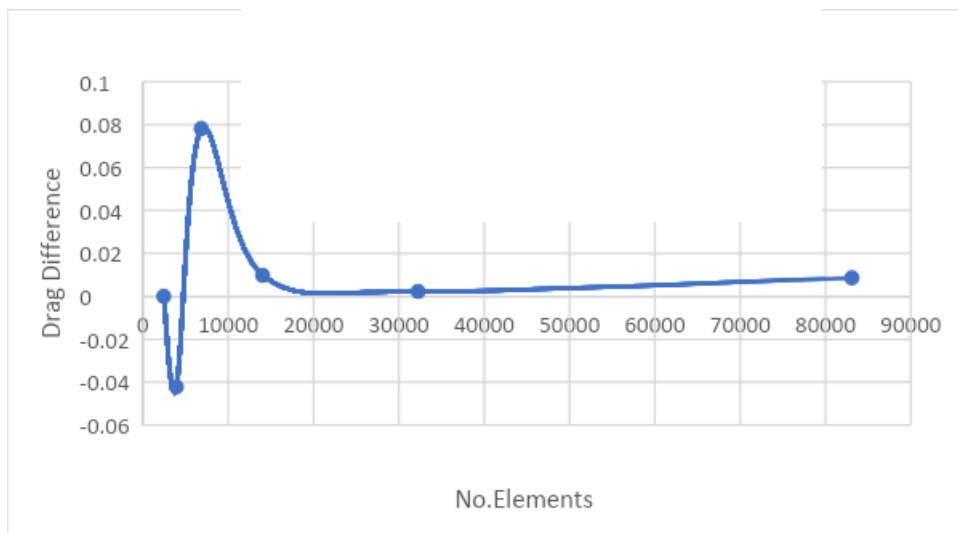
Figure 13: Drag difference Vs. no. of Elements

4.4 Formation length and plot the profile results

The formation length is the length from the stagnation point at the rear of the bluff body to the start of the wake. The velocity Vs domain length graphs represent the velocity along the entire length of the physical domain through the centre of the bluff body.
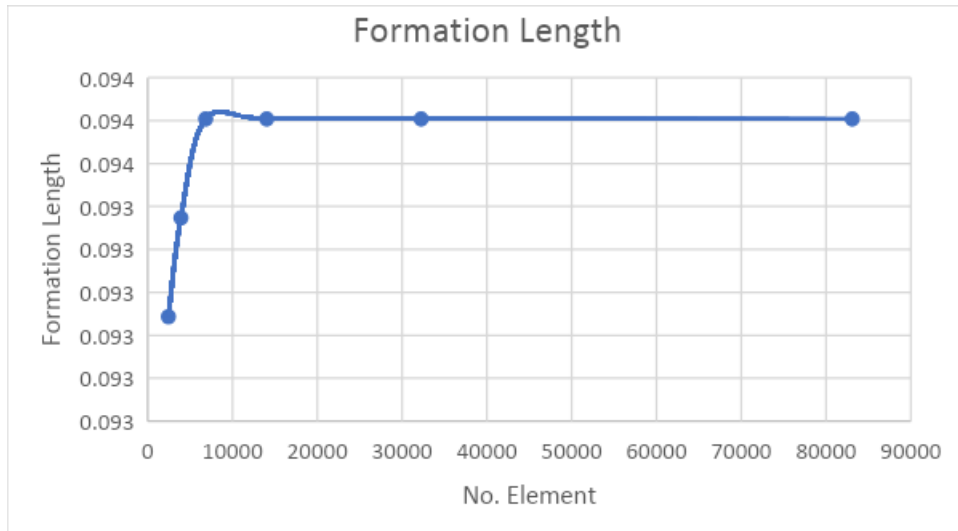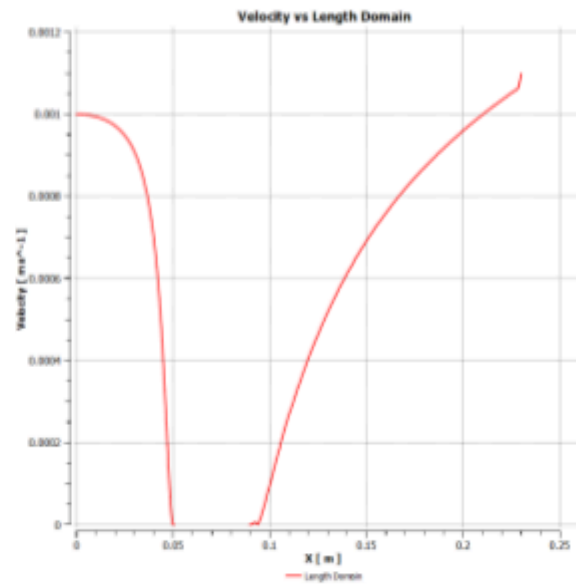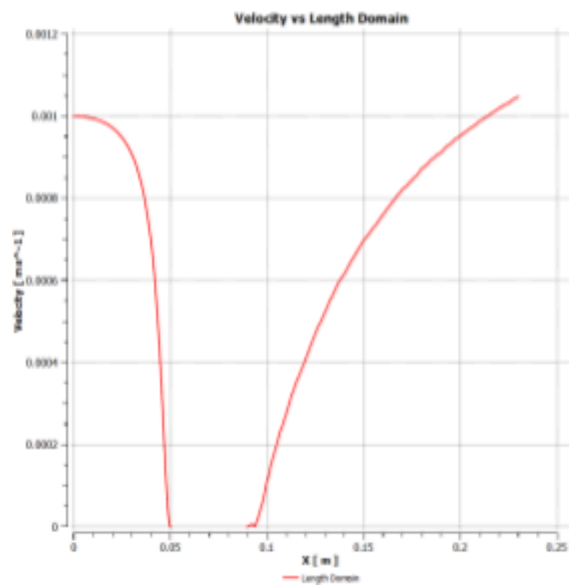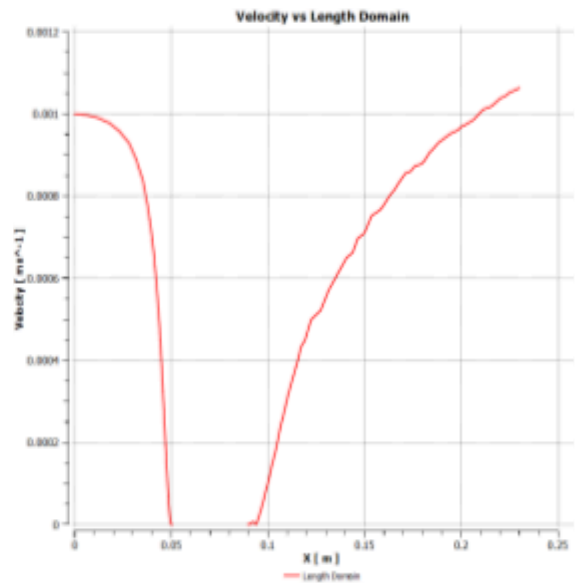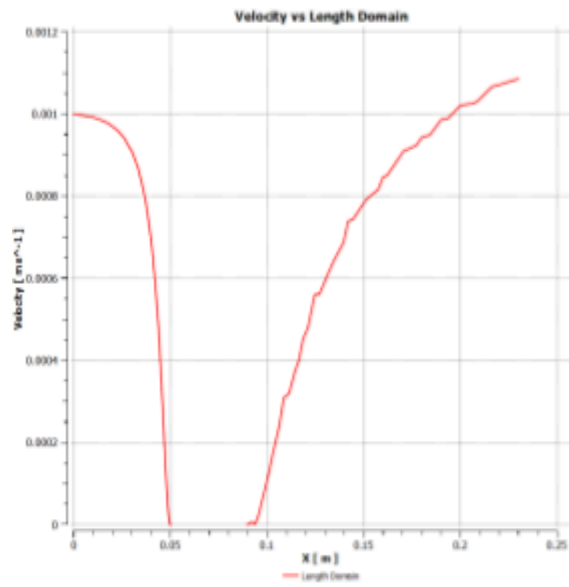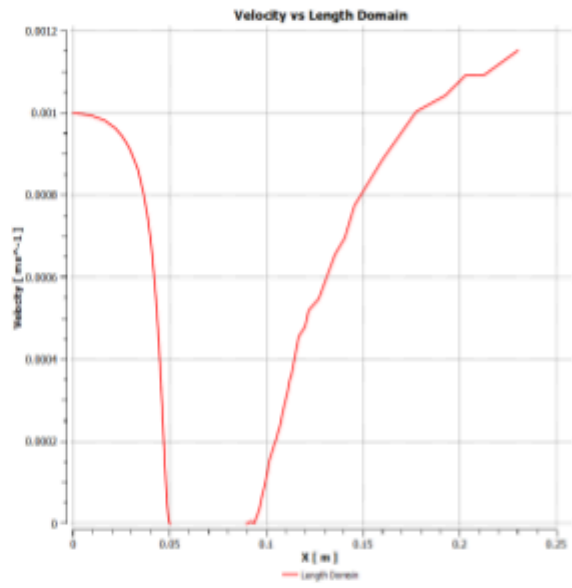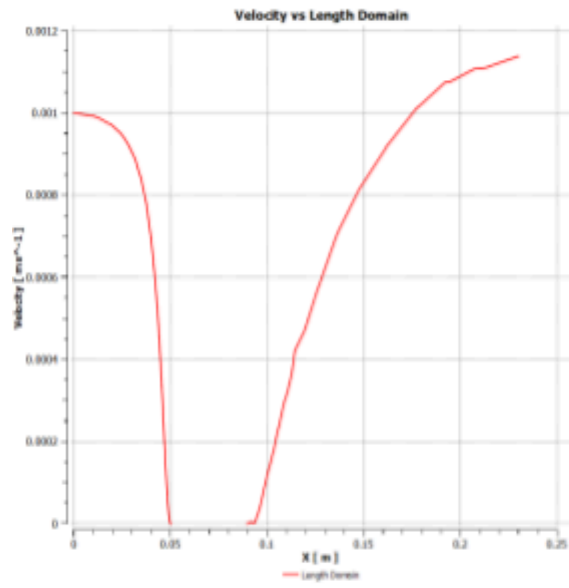

Figure 14: Formation length plot

Figure 15: Velocity Vs. domain length graphs

5.0 Discussion

Discuss contour plots for the velocity field for each grid size:

As can be seen in the contour plots for the velocity field for each different grid size, the more refined the mesh is the more defined and smoother the boundaries are between the different velocities shown. It shows less disturbance to the fluid flow around the body with a longer formation length following the body. Similarly, as the mesh becomes more refined the max velocity increases. Another observation to make is that the highest velocity occurs in the area that is not interrupted by the bluff body between the inlet and the outlet.

Discuss velocity profiles at line A and B:

As can be seen in figure 1 line A is the length from top to bottom starting directly after the outlet side of the body while line B is a top to bottom line 70mm away from line A. These velocity profile plots show the velocity along the vertical plane for line A and B. Similarly, to the contour plots, these graphs produce a smoother and more detailed data curve when there are more elements in the mesh. This is because the more refined the mesh the smaller the elements and the smaller the elements the more data points to plot. Another observation is that line B significantly changes in comparison to line A, its stagnation point visually decreases while its peak velocity increases which is corroborated with the velocity contour and formation length plots.

Discuss drag coefficient and plot against number of grid elements/nodes:

Analysing the drag coefficient graph, it can be seen that the general trend is that the drag coefficient increases as the number of elements increases. However, there is an anomaly for the second mesh which has a smaller drag coefficient than mesh one, with a reading of 5.72 compared to 5.77 respectively. This is most likely due to nature of triangular meshing lacking consistent shapes and patterns which can significantly alter results when the difference in elements is small. Such as it is between mesh one (2442 elements) and mesh 2 (3914 elements) there is a higher possibility for anomalies and inaccuracies when ANSYS calculates drag coefficients.

Discuss formation length:

As can be seen from the formation length graph, the formation length increases significantly during the first 3 meshes as they refine. During the last three meshes the formation length stabilises and seemingly converges at mesh 5. The reason the formation length increases as the element size increase is because the larger the element the more likely that part of that element has fluid with velocity. So ANSYS assumes that all of that element has some sort of velocity. While, in reality, part of that element is stagnant making it part of the formation length when more elements make this differentiation. This is again proven by the velocity vs length domain graphs that show how the more elements makes for a more accurate graph.

Justify appropriate grid size selection based on mesh refinement investigation:

One of the most important factors to consider when justifying the appropriate grid size selection based off the mesh refinement investigation is what mesh creates the most accurate solutions. Without a doubt the greater the number of elements and the greater the number of divisions created the most accurate results. As seen for the contour plots, velocity field graphs and the velocity magnitude vs domain length graphs the more elements means the more accurate because the body is better sampled across its physical domain. Having more elements in a refinement study is similar to having more pixels in a TV screen. Where the more pixels there are the clearer the picture is, in

this case the clearer the results are. Another reason why greater elements are better is because there is less likely to be anomalies. This was seen when observing the results for the drag coefficient of mesh 2.

However, another factor to consider when justifying the appropriate grid size selection is how long the results took to compute. CPU time was computed for each mesh where ideally a fast CPU time is the most ideal. Considering it took the most refined mesh (mesh 6) 1906.332 seconds (31 minutes) to produce results compared to the rest of the meshes which took below 7 seconds. This is a massive difference and hence why mesh 6, although the most accurate, is not the ideal grid size selection.

Therefore, the ideal mesh of choice is mesh 5, with its 400 divisions and 32247 elements it is refined enough to create very accurate results within a suitable time frame. To corroborate this most results have converged by mesh 5 meaning there is a stabilisation of results at this point. This can be seen by comparing results from mesh 5 and 6. For example the drag coefficient difference is a minor $8.6*10^{-3}$, the formation length is the same at 0.093704m and the graphs and plots are visually identical.

6.0 Conclusion

In conclusion when using a triangular mesh generation type and a scaling factor of 2, the most ideal mesh to produce accurate results for a refinement study in a suitable time frame was mesh 5. Mesh 5 had 400 divisions, 32247 elements, an element size of 2.5mm and a CPU time of 6.017 seconds. The most obvious trend from the results is that the smaller the element size and the more refined the mesh the more accurate the results. On the other hand, larger elements with the triangle mesh shape can create anomalies in the results, this was seen with the drag coefficient. Also, the larger the mesh the larger the CPU time required.