

Lab Report #6

Jake Fernhout

July 30, 2024

Drag Coefficient of the Letter A

Professor Lange

Nomenclature

U_0 – Free stream velocity [m/s]

Re – Reynolds number [dimensionless]

ρ – Density [kg/m³]

μ - Dynamic viscosity [kg/(m s)]

ν – Kinematic viscosity [m²/s]

Introduction

Throughout the course of MEC E 539, each tutorial and associated lab has reinforced various CFD methods and best practices such as the application of boundary conditions, meshing, and appropriate turbulence models. This lab will take all the lessons learned and apply them to a geometry of my own design, which will be flow over the letter A. The only geometry restrictions given by the assignment were that it must fit within the given flow set up as seen in Figure 1 and that it must be relatively blunt. The aim of this assignment is to find the drag coefficient associated with the letter “A” and compare it to known drag coefficients of similar shapes to demonstrate the validity of changes to meshing and turbulence models. Two simulations will be performed, the first a coarse mesh with an upstream advection scheme and then a finer mesh using a CDS advection scheme.

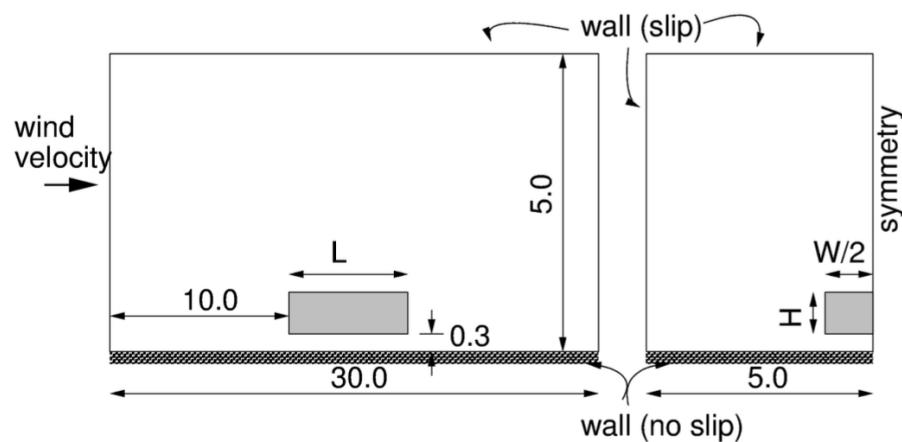


Figure 1: Schematic of Setup (all dimensions given in meters).

The given geometry must fit within the gray box with the dimensions from Table 1 and touch every wall of the box at (i.e., the designed geometry must have maximum and minimum dimensions L, H and W).

Table 1: Dimensions of Box Size

L [m]	6.5
H [m]	2.5
W [m]	3.0

I designed the blunt object using the CAD software Onshape and imported it into CFX. The sizing of the object can be seen below in Figure 2.

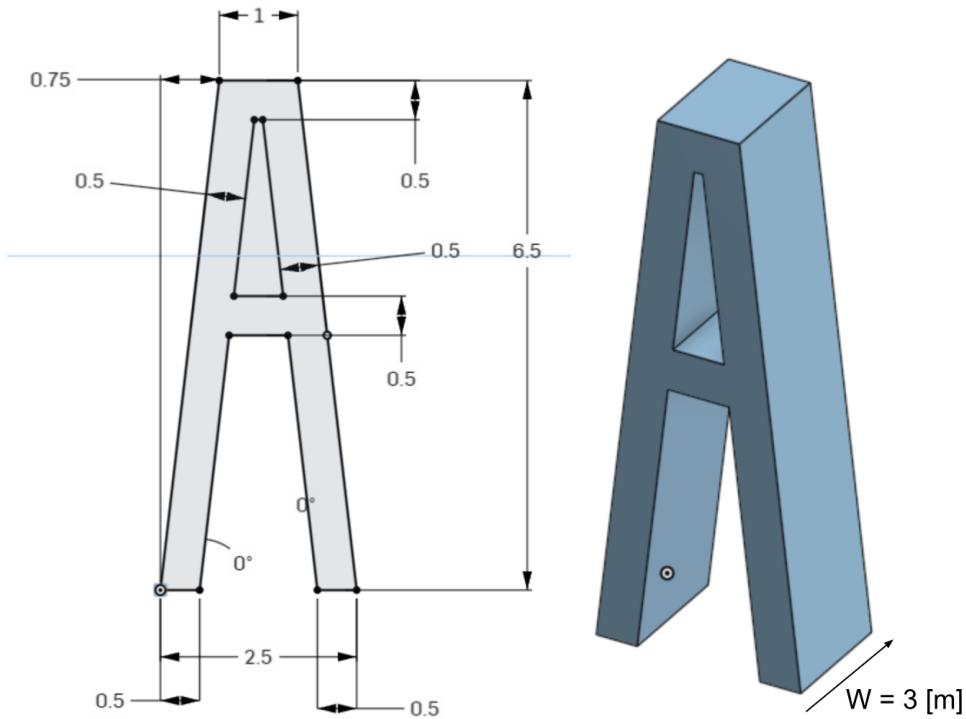


Figure 2: Dimensions of Blunt Object "A" (all dimensions in meters)

For simplicity and memory requirements, the object will be split in half symmetrically such that $W = 1.5$ [m] for the new object. A symmetry plane will be placed along this split as laid out in Figure 1. Merging the object into the geometry form Figure 1 can be seen in the following figures:

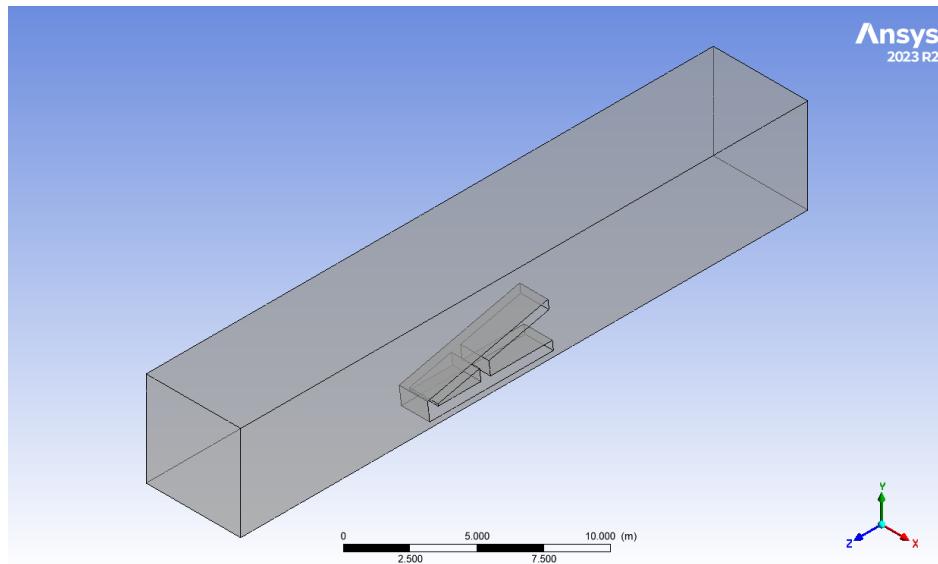


Figure 3: Schematic of Flow Problem

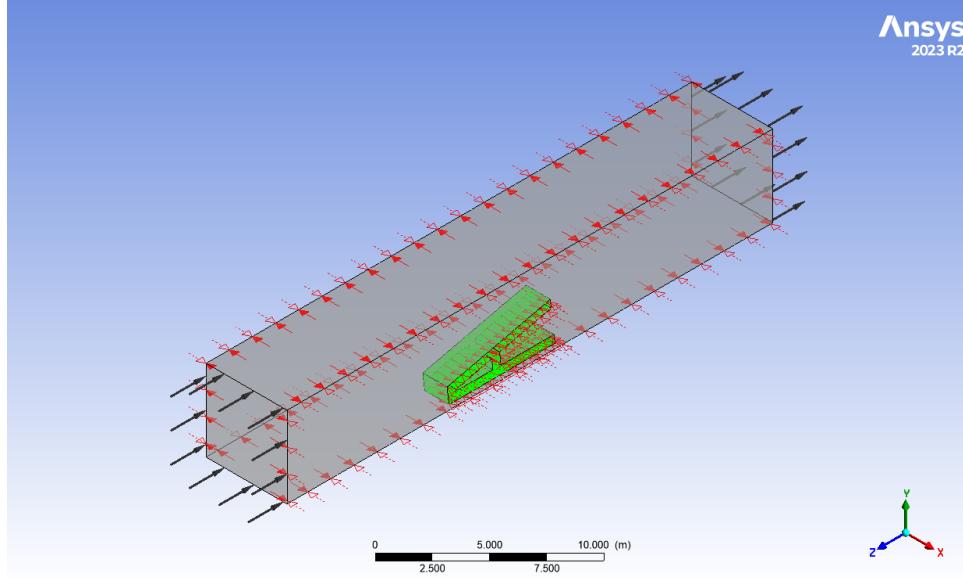


Figure 4: Schematic of Flow Problem Showing Boundary Conditions

Figure 4 shows the flow direction as well as boundary conditions for the faces of the problem. The black arrows show the direction of flow, the red ones indicate various slip conditions. The air flow parameters and slip conditions can be found in the tables below

Table 2: Air Fluid Properties

Fluid Entrance Speed	15.6 [m/s]
Temperature	25 [$^{\circ}$ C]
Pressure	100 [kPa]
Density	1.185 [kg/m ³]
Dynamic Viscosity	1.831E-05 [kg/(m s)]
Air is not an ideal gas	

Table 3: Slip Conditions at Features

Inlet	Subsonic Flow
Outlet	Subsonic Flow, Relative Pressure is 0 kPa
Symmetry	Symmetry
"A"	No Slip
Floor	No Slip
Top	Free Slip
Walls	Free Slip

To determine if turbulence models are needed, the Reynolds Number was calculated (see appendix) and found to be 2.75E6, which means the flow is turbulent. Turbulence settings for all simulations can be found in the table below.

Table 4: General Turbulence Settings

Turbulence Intensity	5%
Turbulence Length Scale	0.1 [m]

The first coarse mesh was generated and is to be used as a baseline in order to compare with the finer mesh. The settings used for the coarse mesh and the fine mesh can be found in Tables 5 and 6 respectively. An intermediate mesh was designed and simulated however it did not have the required number of nodes (300 000) and so it will not be discussed, however it can be found in the Appendix. To view the meshes, two planes were used as locations in CFX. One plane is right beside the symmetry plane, and the other “slice” is placed such that it cuts through the letter horizontally parallel with the x-z plane. For a view of this slice and its coordinates see the Appendix.

Table 5: Coarse Mesh Settings

Method	Tetrahedrons
Max Element Size	0.2 [m]
Element Size	0.2 [m]
Geometry	Entire Flow Area
Adaptive Meshing	No
Number of Nodes	176 242

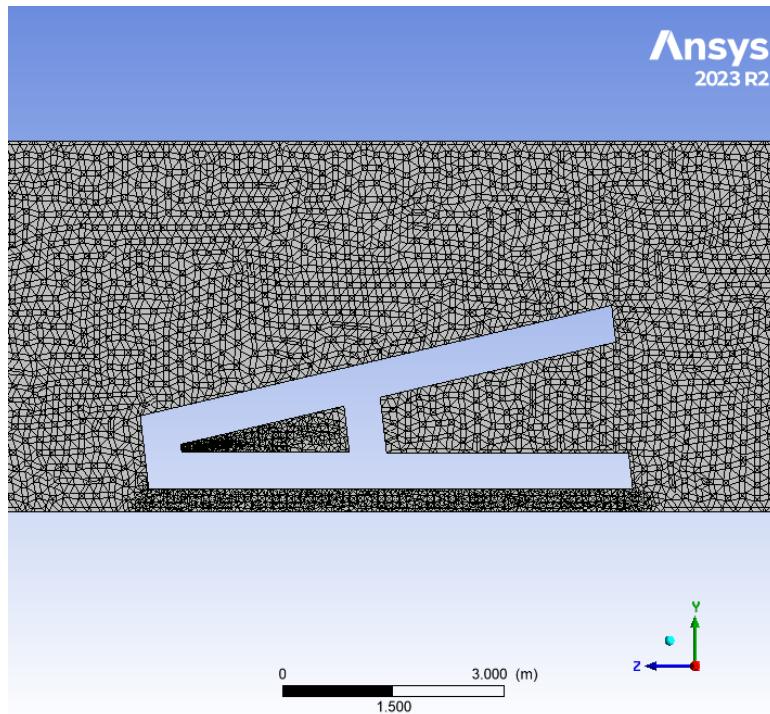


Figure 5: Coarse Mesh from X View on Symmetry Plane

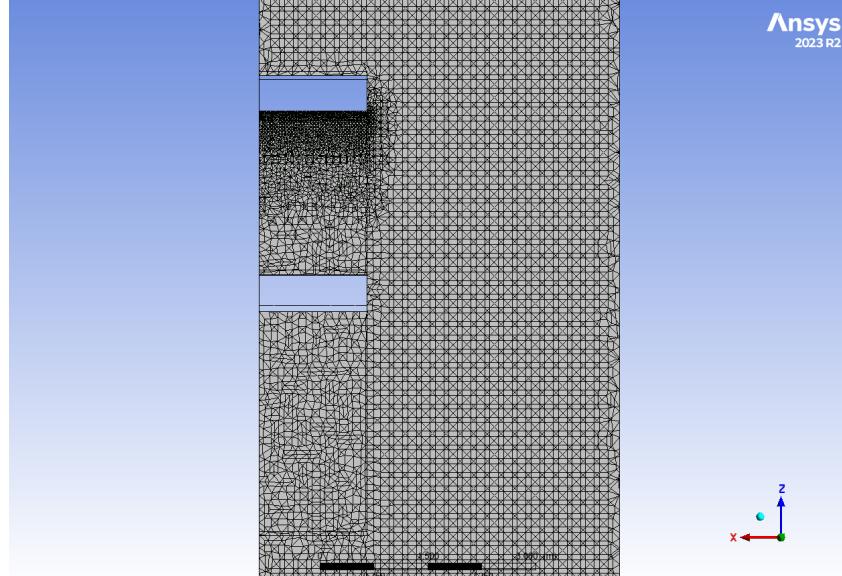


Figure 6: Coarse Mesh from Y View on Slice

Table 6: Fine Mesh Settings

Method 1	Tetrahedrons
Max Element Size	0.2 [m]
Element Size	0.2 [m]
Geometry	Entire Flow Area
Method 2	Body of Influence
Element Size	0.1 [m]
Method 3	Inflation Layer
Max Thickness	0.1 [m]
Number of Layers	5
Adaptive Meshing	No
Number of Nodes	290 044

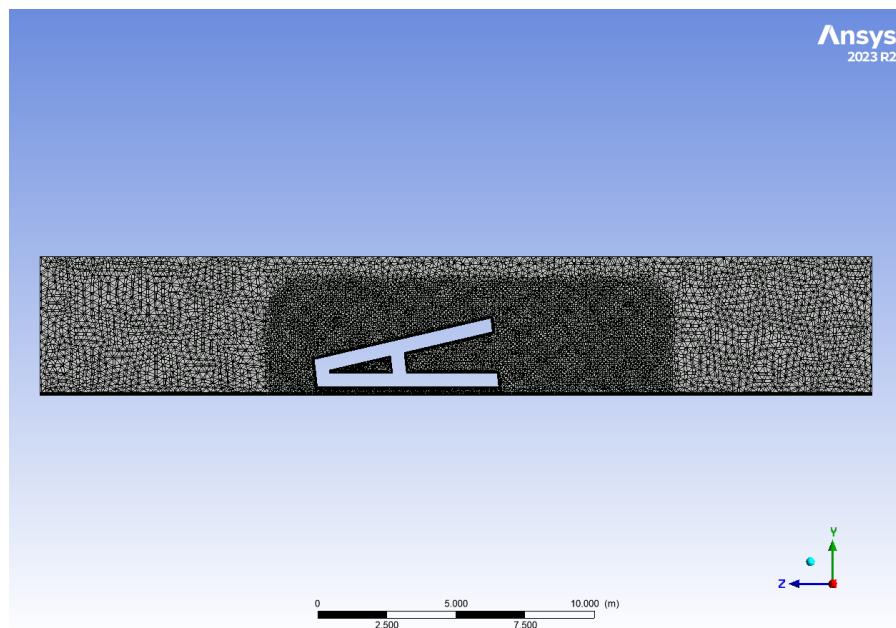


Figure 7: Fine Mesh from X View on Symmetry

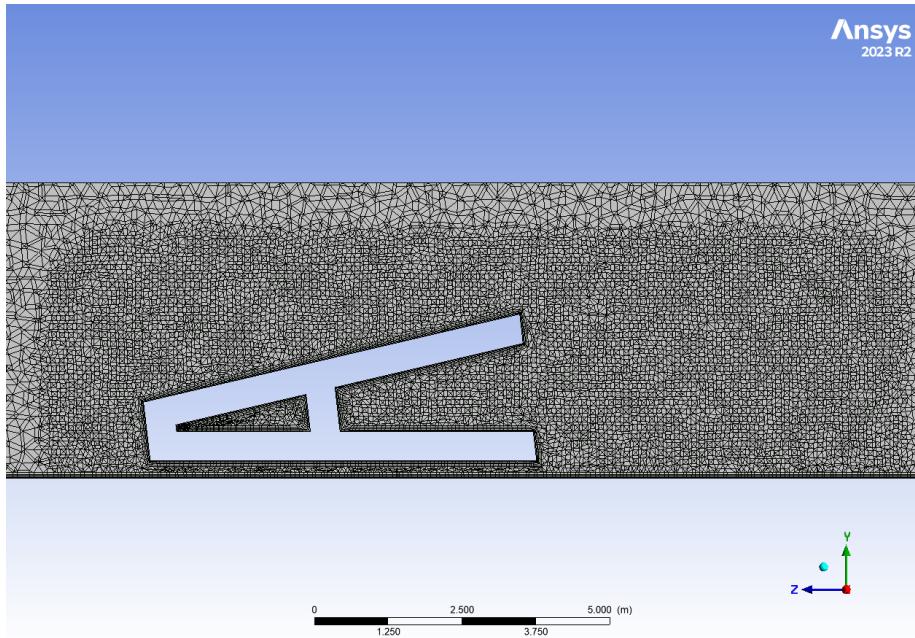


Figure 8: Fine Mesh from X View on Symmetry Plane Closer Look

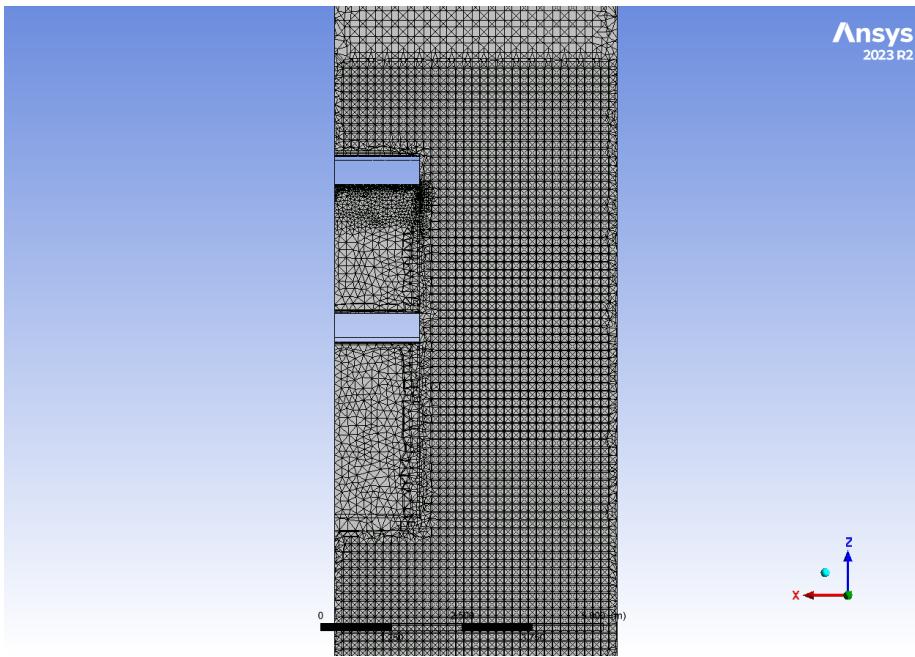


Figure 9: Fine Mesh from Y View on Slice Closer Look

For full views of meshes not enlarged around the geometry see the Appendix.

Monitoring points were added before simulations were performed in order to see velocity convergence for the different meshes. These points are shown in Figure 10 and their coordinates can be found in Table 7.

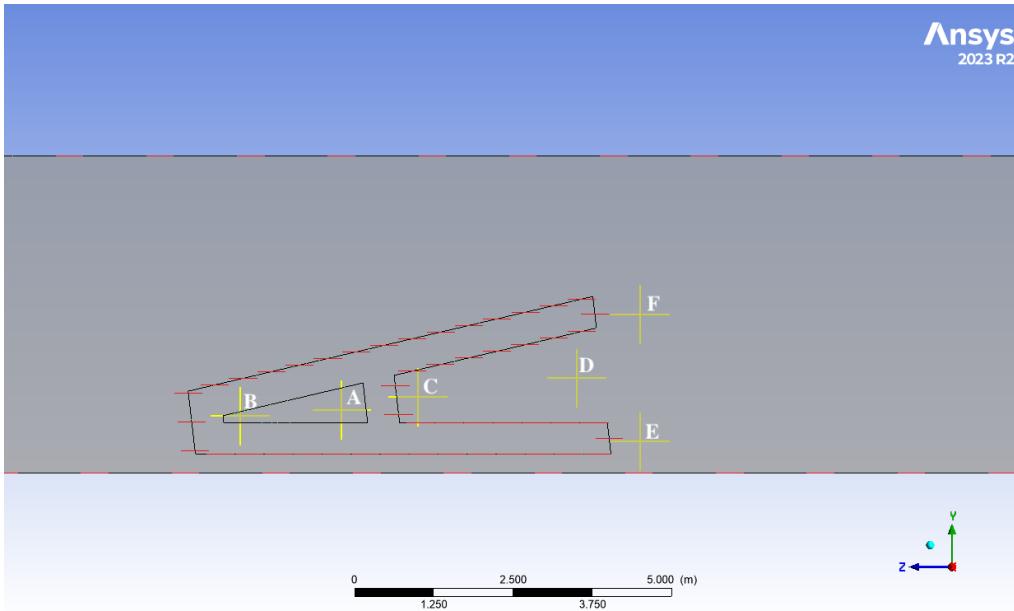


Figure 10: X View of Monitoring Points

Table 7: Monitoring Points Coordinates

Point	X [m]	Y [m]	Z [m]
A	4.95	1	17.7
B	4.95	0.9	19.3
C	4.95	1.2	16.5
D	4.95	1.5	14
E	4.95	0.5	13
F	4.95	2.5	13

Analysis

The coarse mesh simulations were performed with the parameters found in Table 8.

Table 8: Parameters for Coarse Mesh

Advection Scheme	UDS
Turbulence Model	k-epsilon
Time	Steady State
Fluid	Continuous and Non-Ideal
Turbulence Intensity	5%
RMS Convergence	1E-5

This simulation produced the following plots:

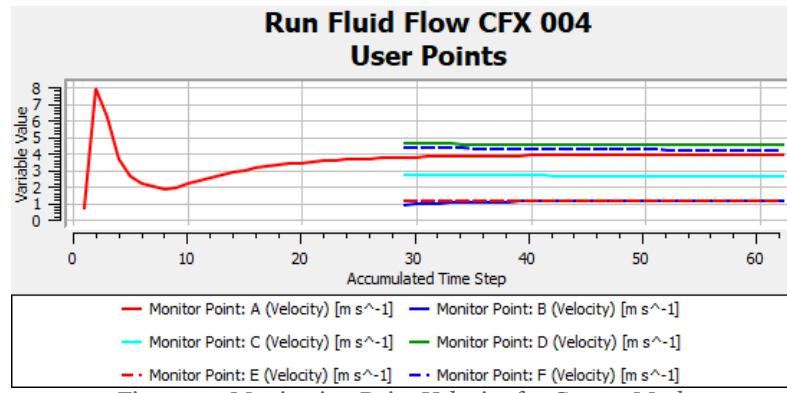


Figure 11: Monitoring Point Velocity for Coarse Mesh

It can be seen in the figure above that the monitoring points converged to a steady velocity, validating the mesh.

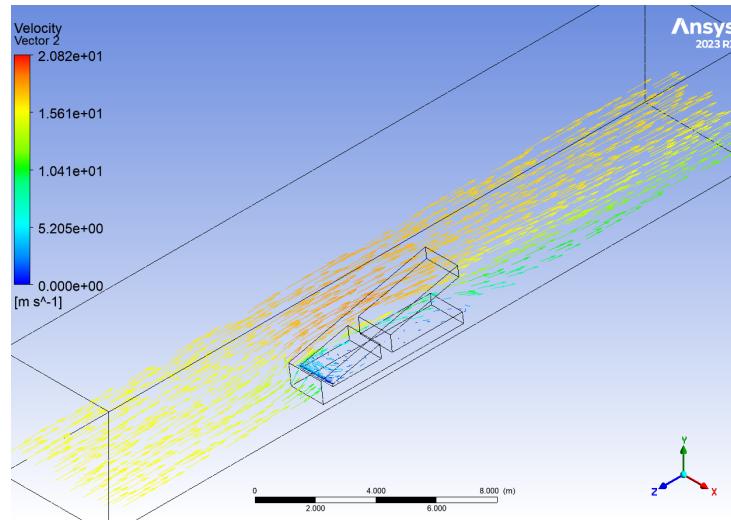


Figure 12: Vector Plot on Slice for Coarse Mesh

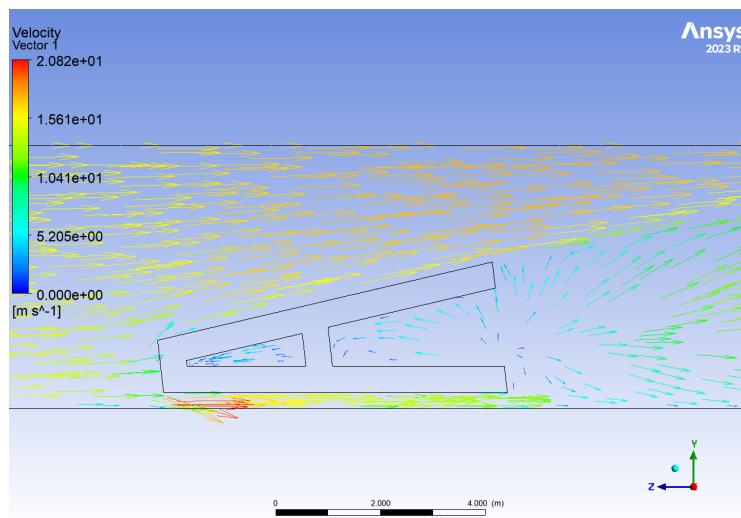


Figure 13: Vector Plot on Symmetry of Coarse Mesh

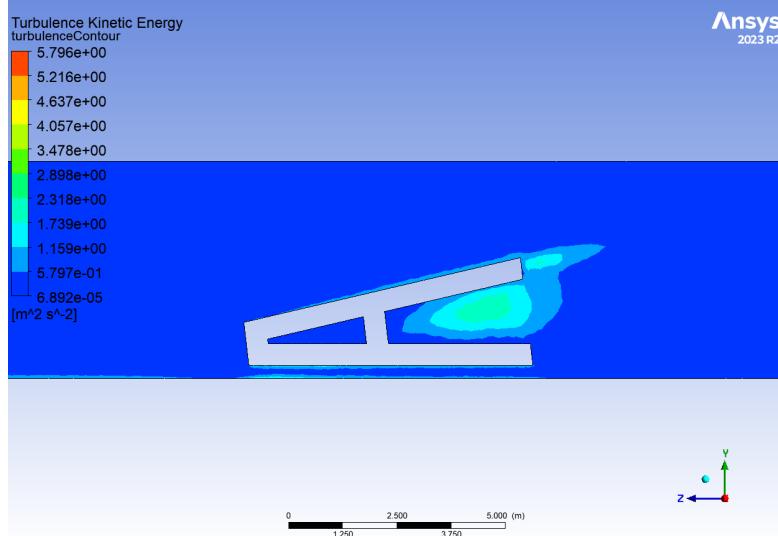


Figure 14: TKE Plot on Symmetry Plane Coarse Mesh

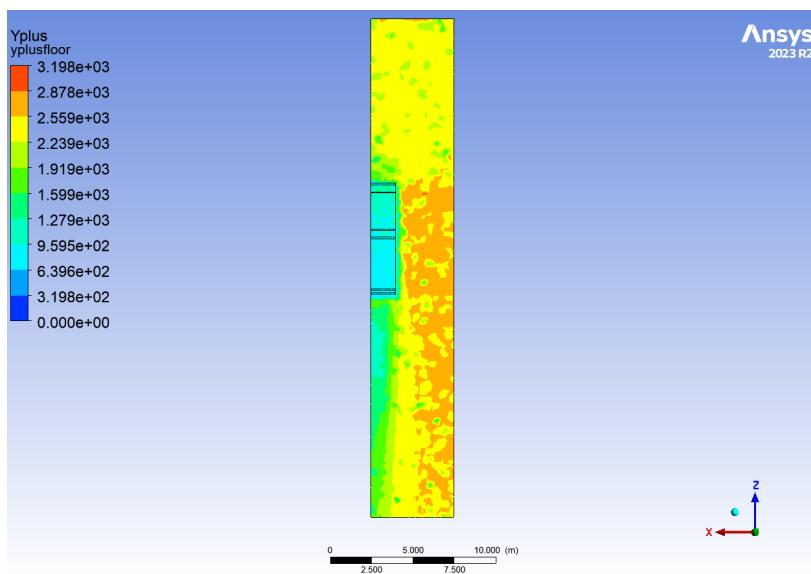


Figure 15: Coarse Mesh Y Plus Floor

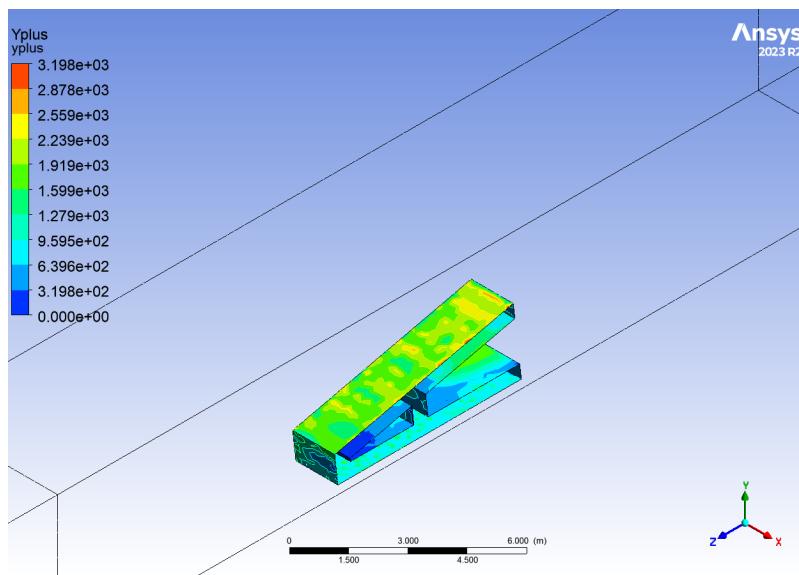


Figure 16: Coarse Mesh Body Y Plus

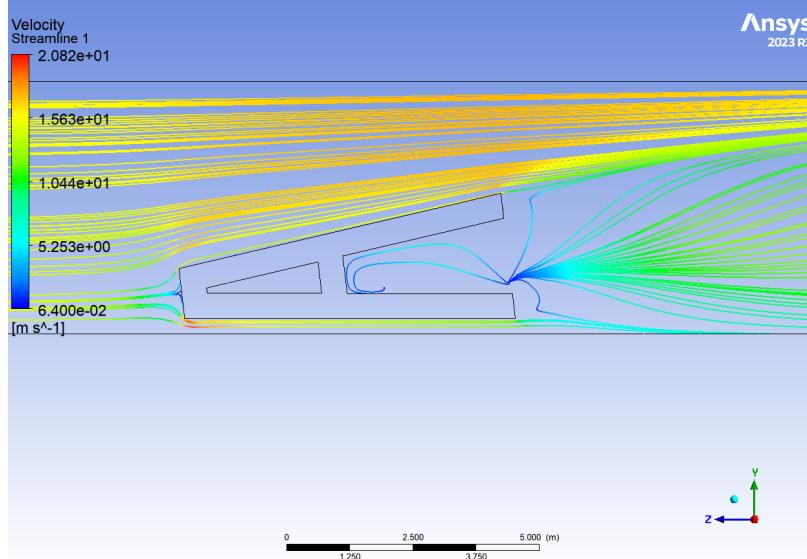


Figure 17: Streamlines on Symmetry for Coarse Mesh

More pictures of the coarse mesh flow can be found in the Appendix. The drag force was calculated in CFX Post using expressions and equations. First the drag force in the Z direction was found to be 493.6 [N]. The formula for drag coefficient:

$$C_D = \frac{2D}{\rho V^2 A} = \frac{2(493.6 \text{ [N]})}{1.185 \text{ [kg/m}^3\text{]} * 15.6^2 \text{[m/s]} * 3.75 \text{ [m]}} = 0.91$$

The fine mesh simulations were performed with the parameters found in Table 9.

Table 9: Parameters for Fine Mesh

Advection Scheme	CDS (Blend Factor 1)
Turbulence Model	SST
Time	Steady State
Fluid	Continuous and Non-Ideal
Turbulence Intensity	5%
RMS Convergence	1E-5

This simulation resulted in the following figures:

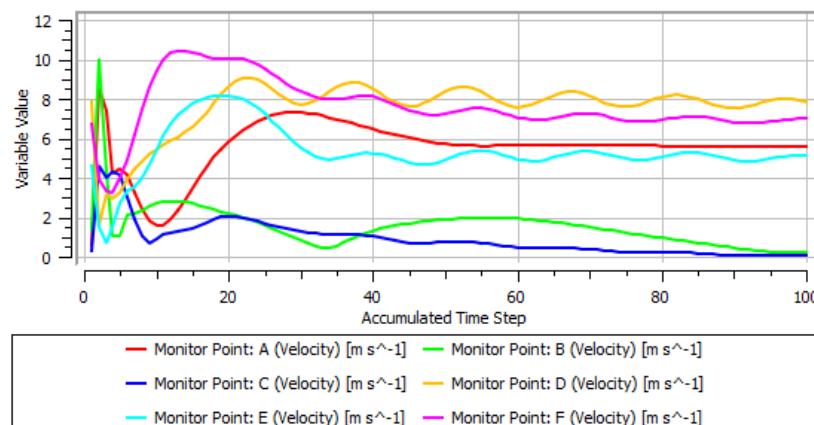


Figure 18: Flow Velocities at Monitoring Points for Fine Mesh

It can be seen in the figure above that the monitoring points converged to a steady velocity, validating the mesh.

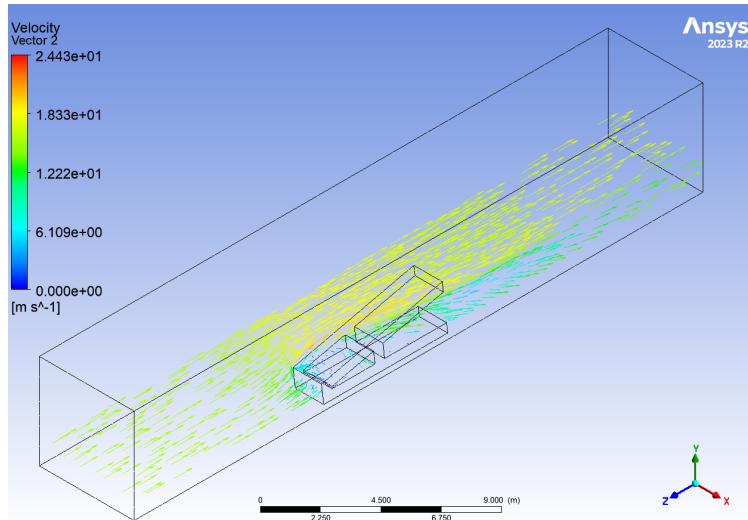


Figure 19: Vectors on Slice for Fine Mesh

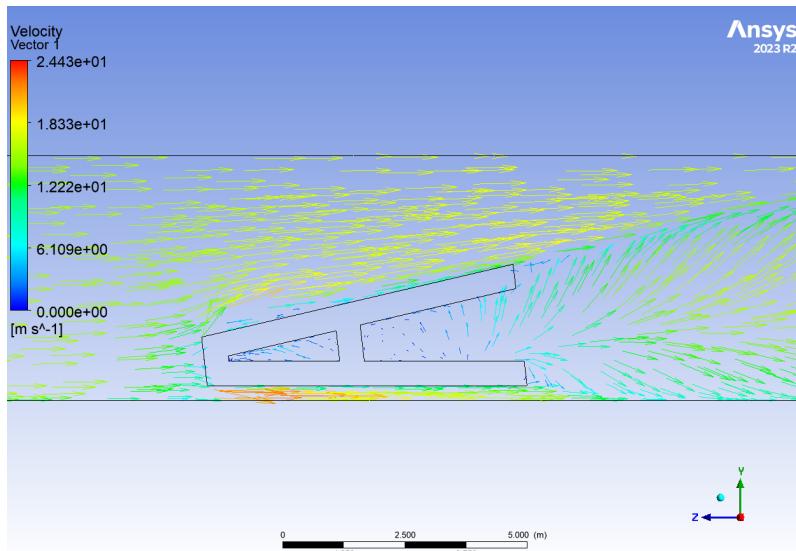


Figure 20: Vector Plot on Symmetry Plane for Fine Mesh

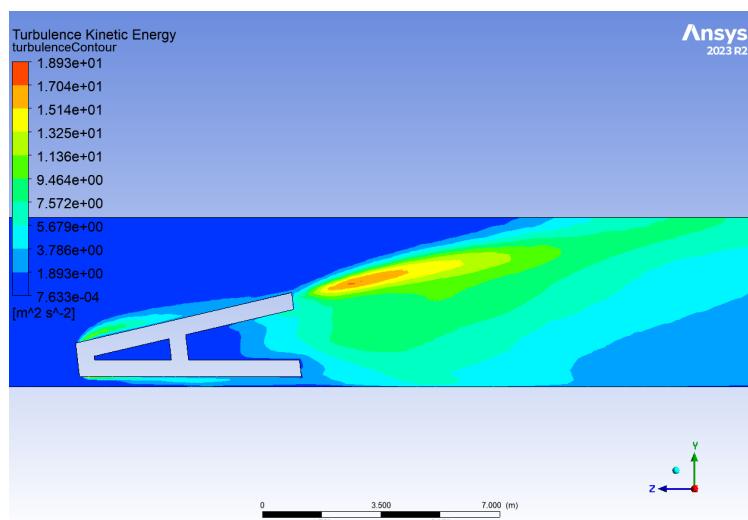


Figure 21: TKE Plot on Symmetry Plane for Fine Mesh

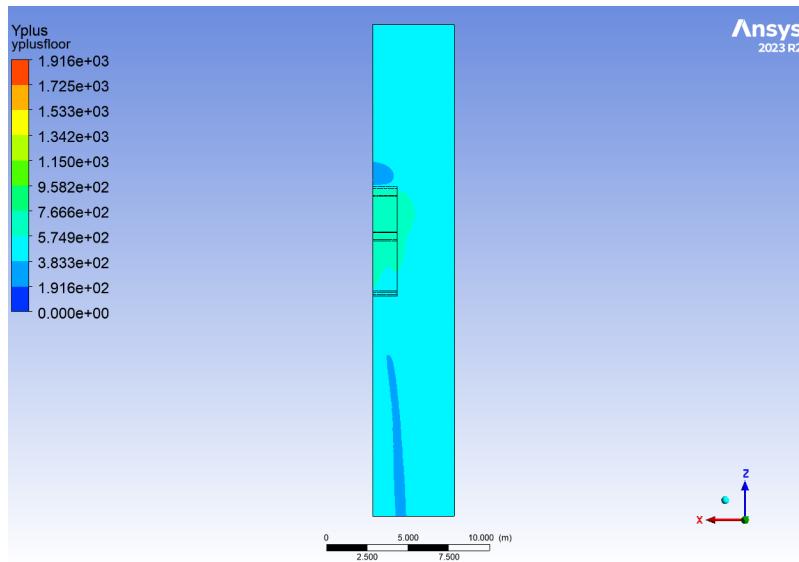


Figure 22: Y Plus on Floor for Fine Mesh

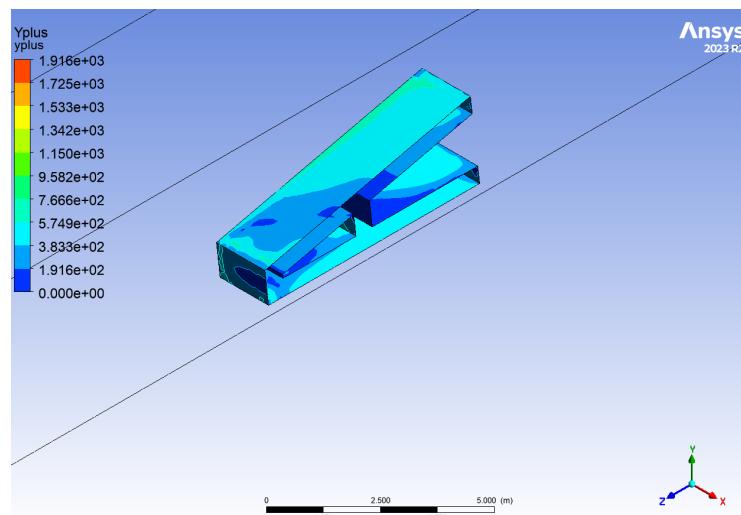


Figure 23: Y Plus on Body for Fine Mesh

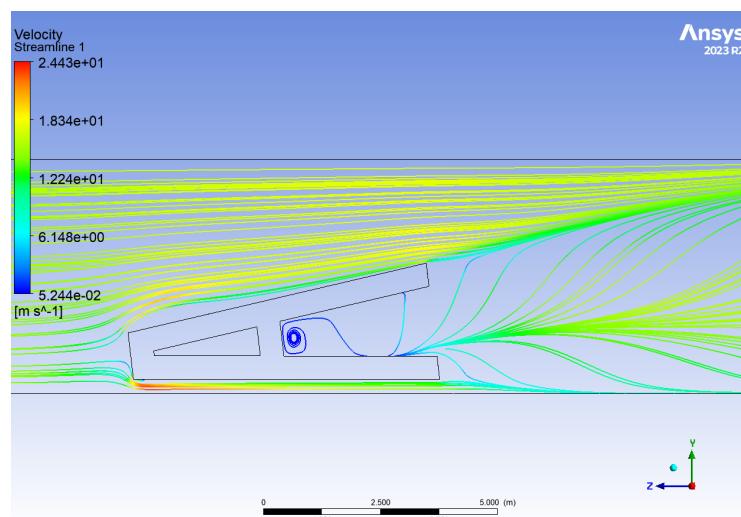


Figure 24: Streamlines on Slice for Fine Mesh

More pictures of the fine mesh flow can be found in the Appendix. The drag force was calculated in CFX Post using expressions and equations. First the drag force in the Z direction was found to be 380.9 [N]. The formula for drag coefficient:

$$C_D = \frac{2D}{\rho V^2 A} = \frac{2(380.9 \text{ [N]})}{1.185 \text{ [kg/m}^3\text{]} * 15.6^2 \text{[m/s]} * 3.75 \text{ [m]}} = 0.70$$

Results and Discussion

From the Y-Plus plots developed from both meshes, it can be seen that the wall treatment method satisfies Log Law of the Wall for both as the maximum values exceed 300.

Known drag coefficients are given from the problem and can be seen in Figure 25.

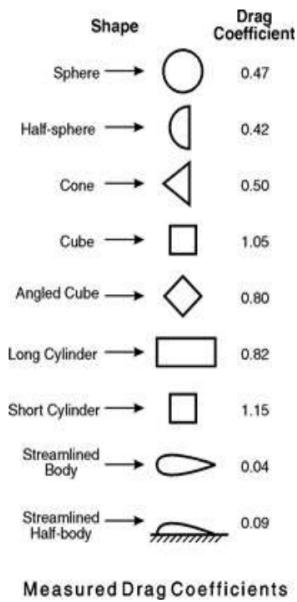


Figure 25: Known Drag Coefficients for Various Shapes

The most similar shape to the letter “A” is the angled cube, however the angle is less creating a more streamlined shape. The holes in the letter “A” will also induce some drag, but based on the plots from the simulation it is not the main generator of drag. As such a drag coefficient between the cone and the angled cube would be accurate, around 0.70. The original coarse mesh yielded a drag coefficient of 0.91, which is close to the estimated actual C_D . The fine mesh focused on refining the gaps in the letter as well as the wake region behind, theoretically improving the accuracy of the simulation. This fine mesh yielded a drag coefficient of 0.70, which is exactly what was predicted from similar shapes. The refined grid provided a more accurate drag coefficient as expected. Of course the predicted drag coefficient is a rough estimate and not exact, however it serves the general purpose of validating the flows.

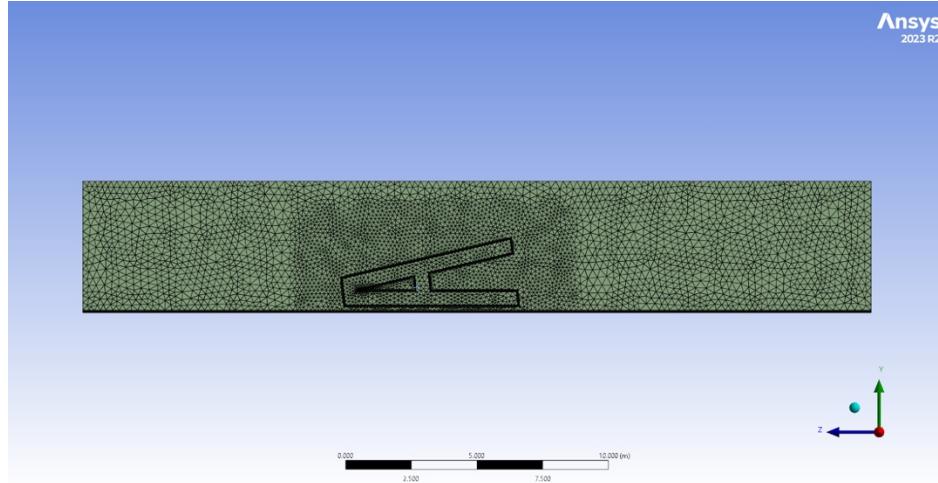
Appendix

Reynolds Number Calculation

$$\nu = \frac{\mu}{\rho} = \frac{1.831E - 05 [kg/(ms)]}{1.185 [kg/m^3]} = 1.545E - 05 [m^2/s]$$
$$D_h = \frac{4A}{l} = \frac{4 * 7.5}{11} = 2.7[m]$$

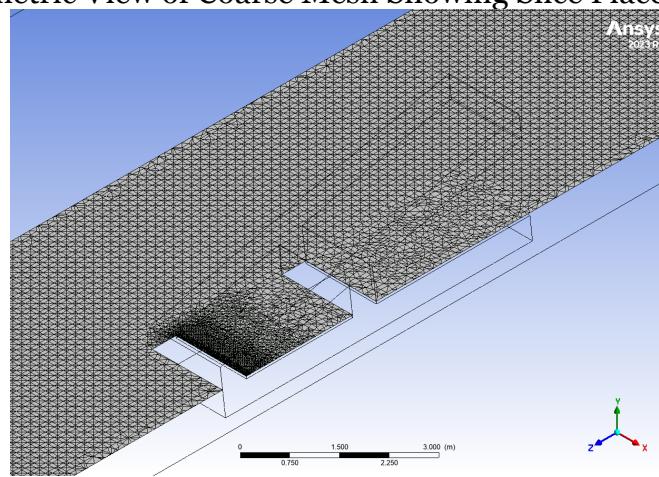
$$Re_{D_h} = \frac{VD_h}{\nu} = 2.75E6$$

Intermediate Mesh (236962 nodes)

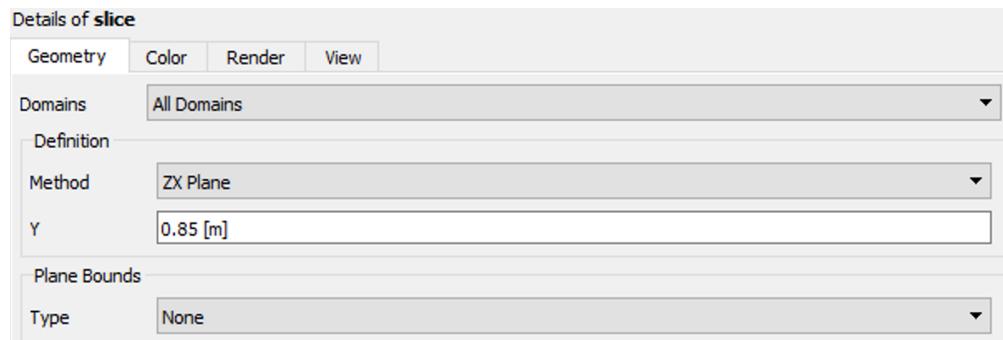


Slice view and coordinates

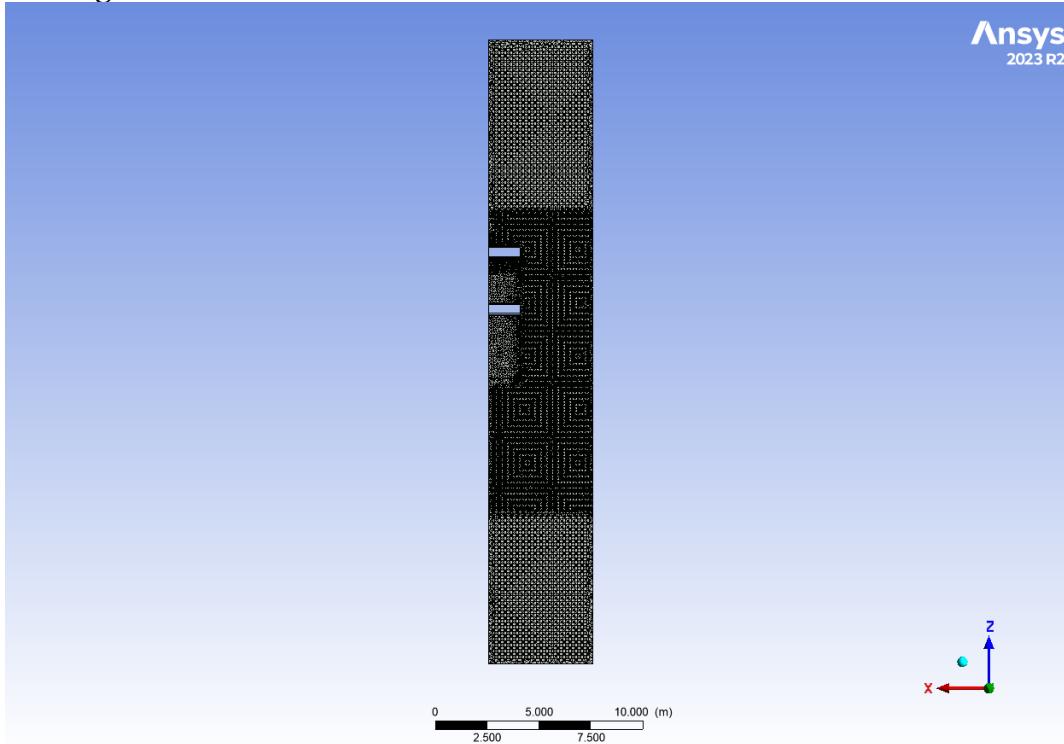
Isometric View of Coarse Mesh Showing Slice Placement



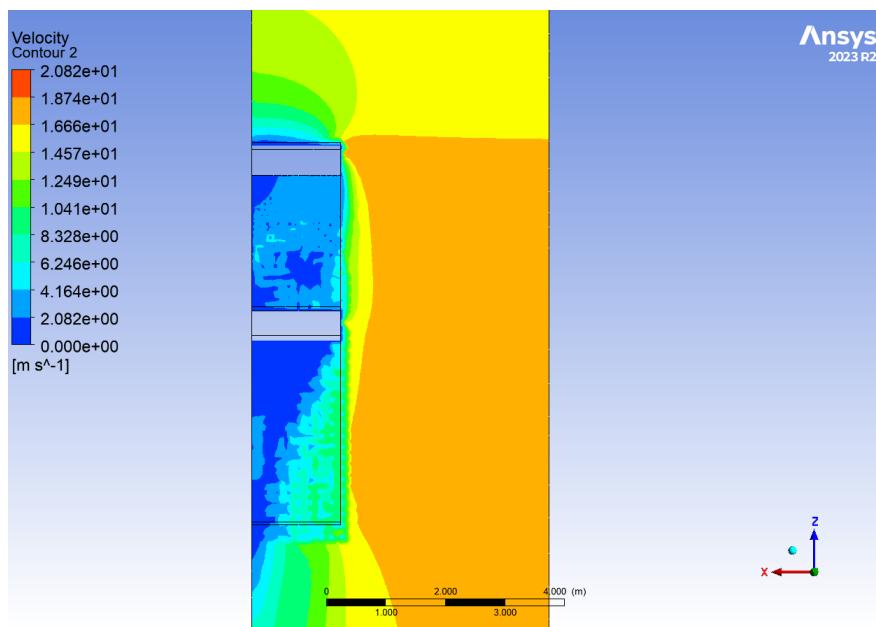
Slice Coordinates

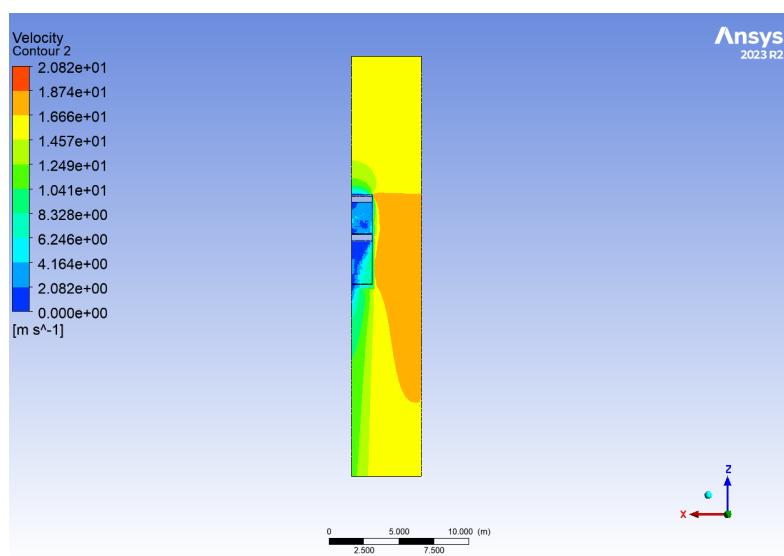
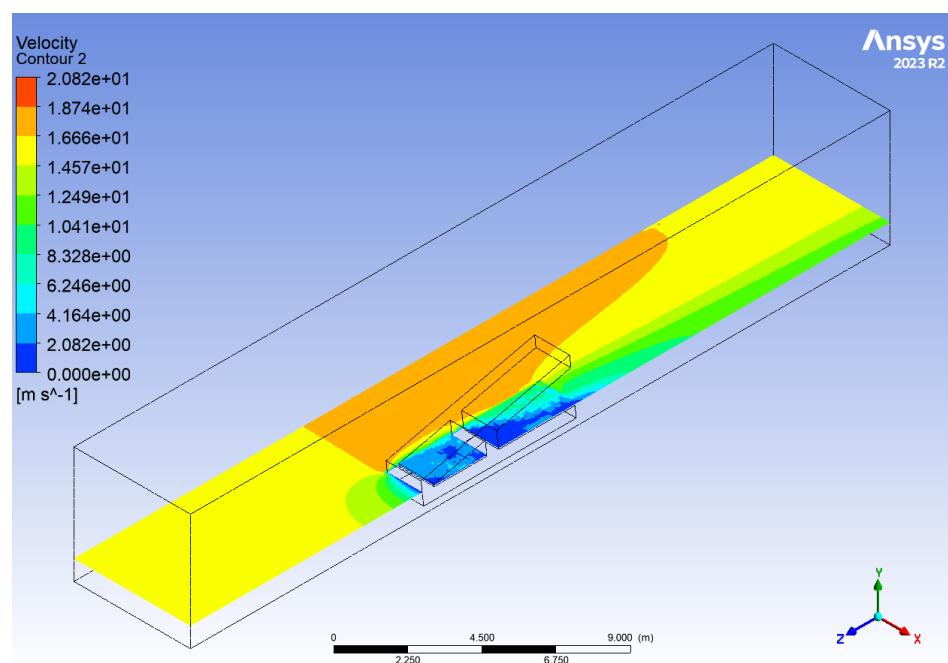
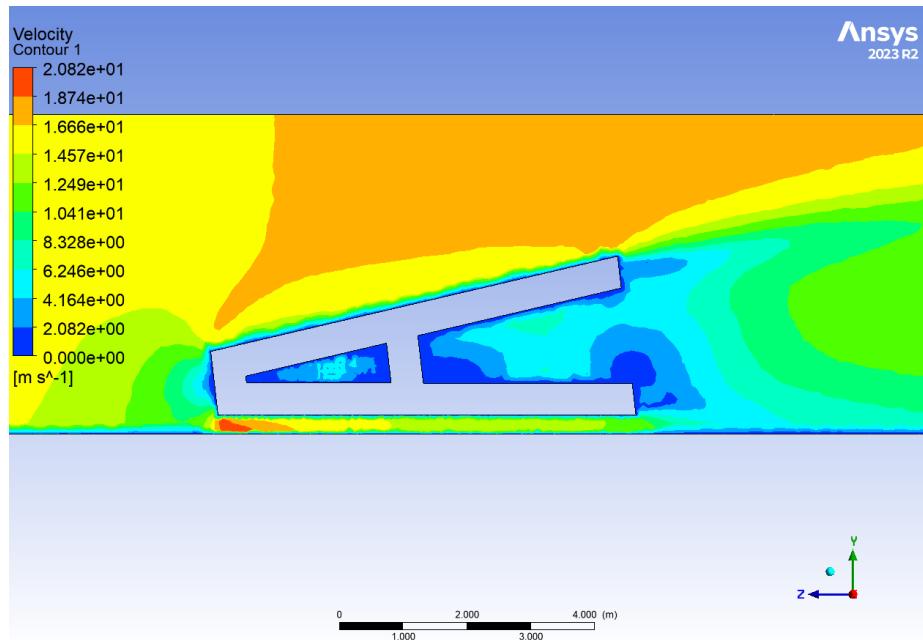


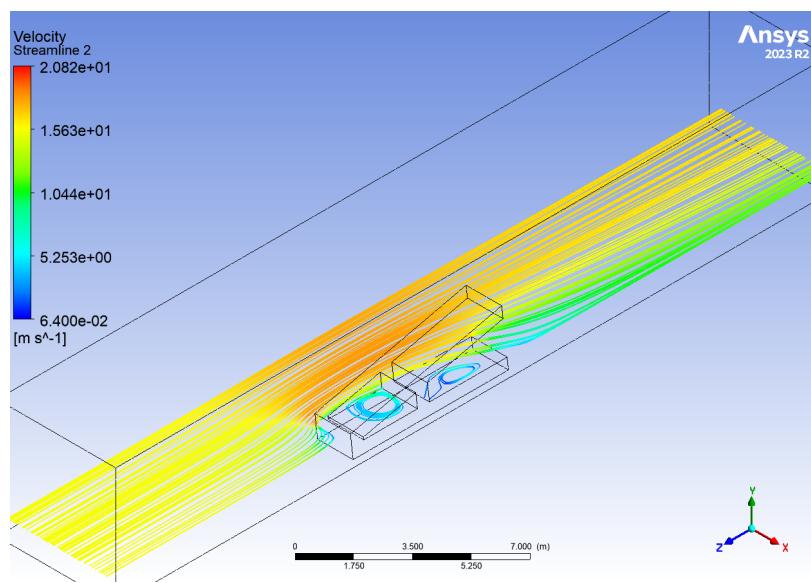
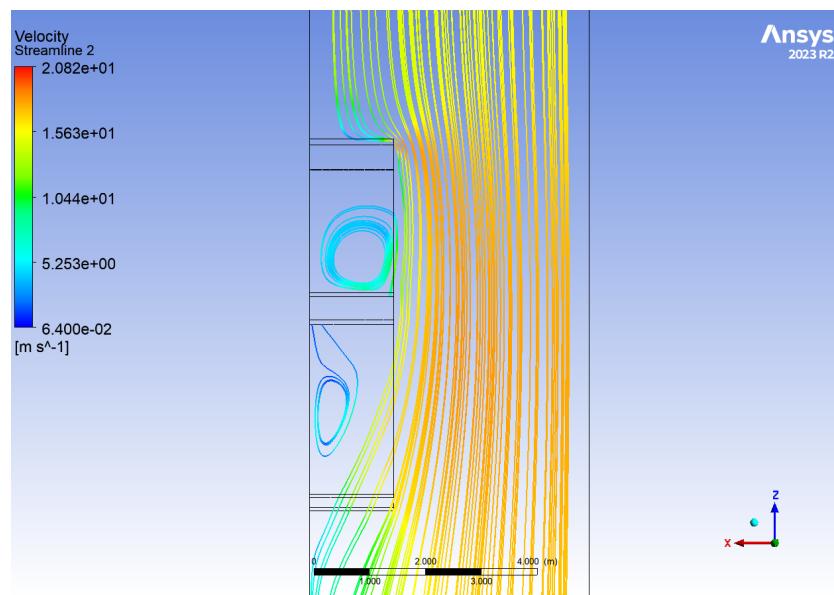
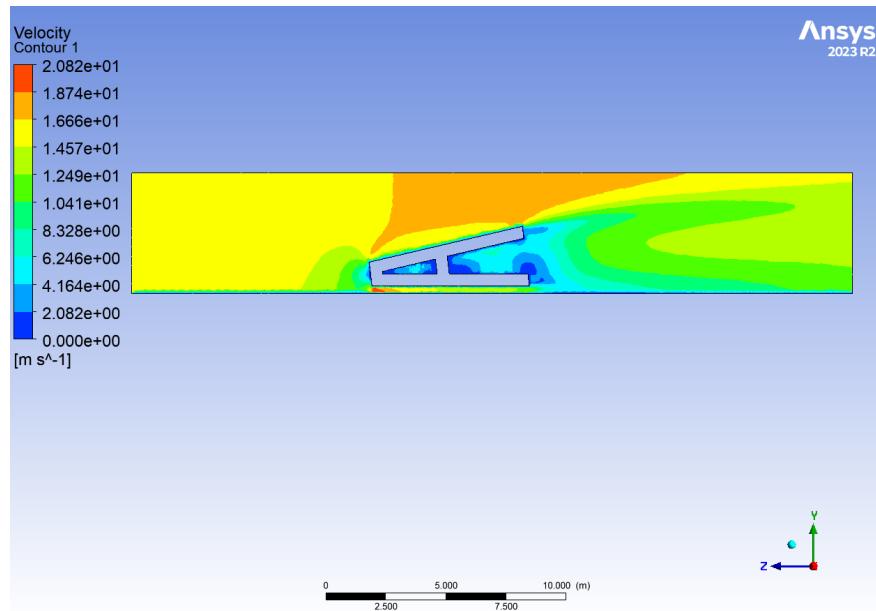
Full Fine Mesh Images

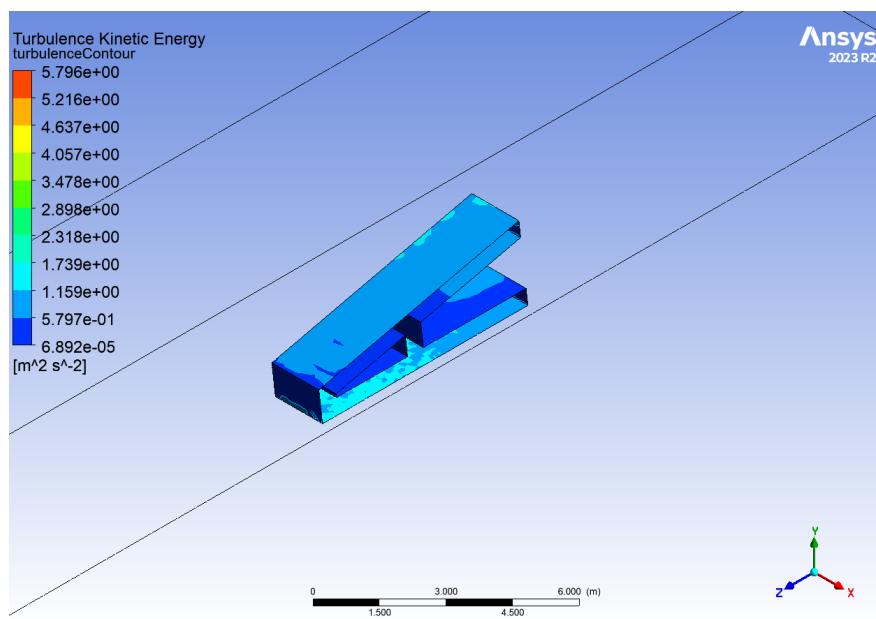
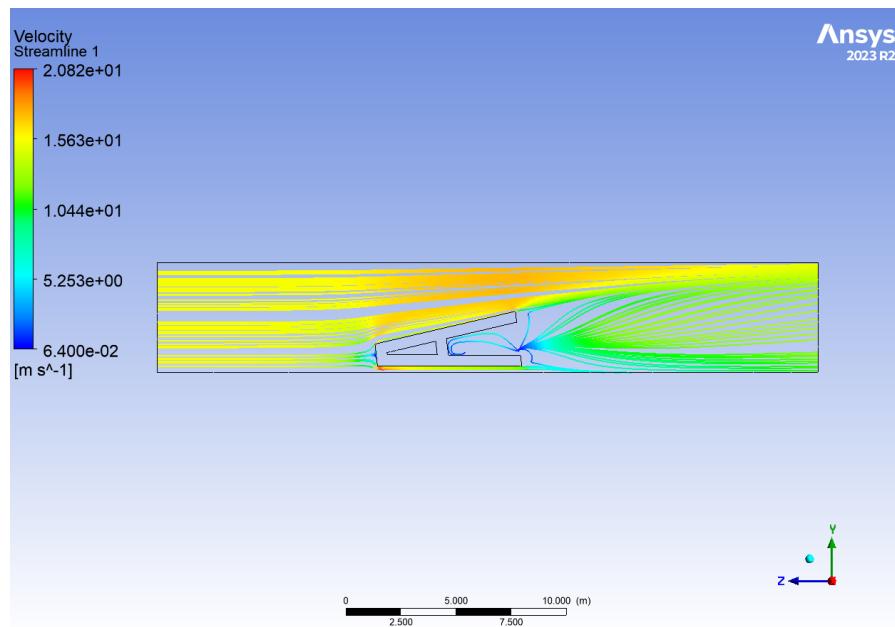
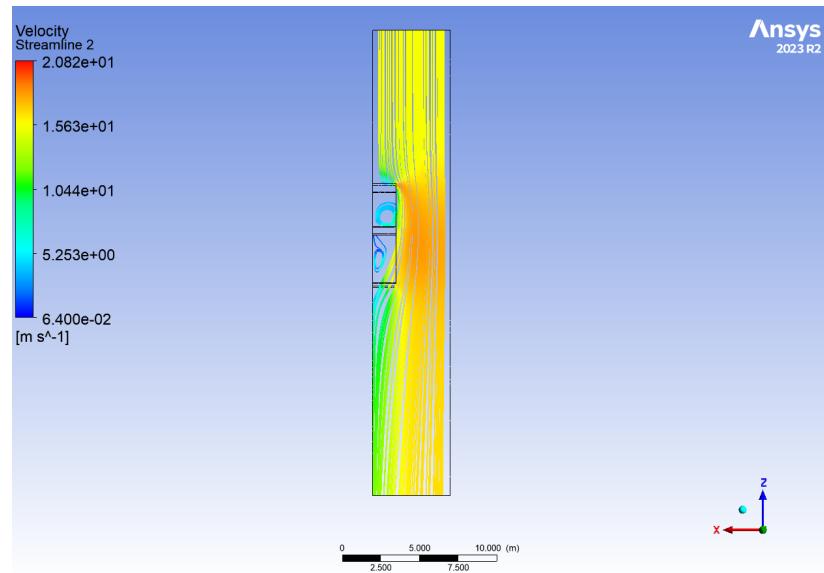


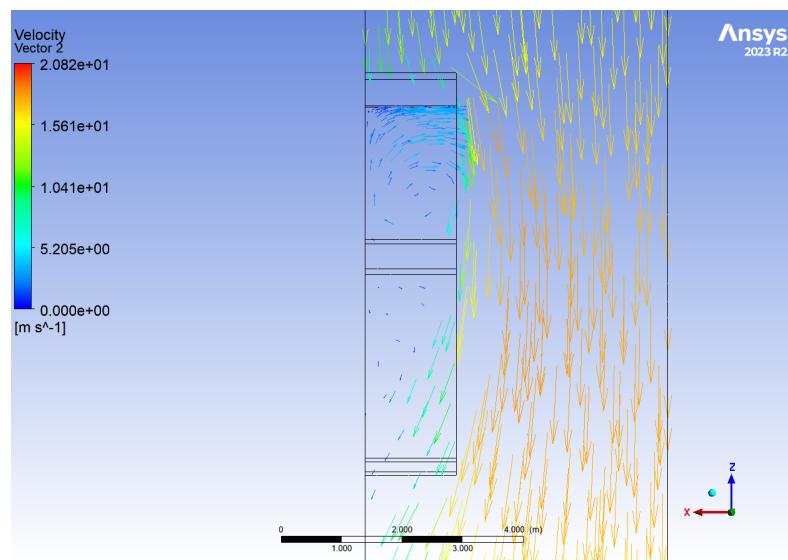
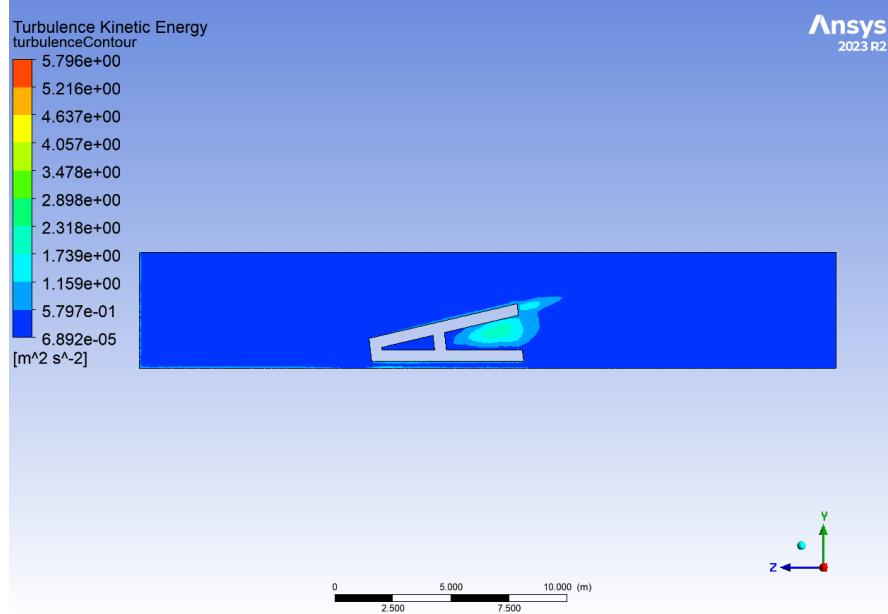
More coarse mesh figures:



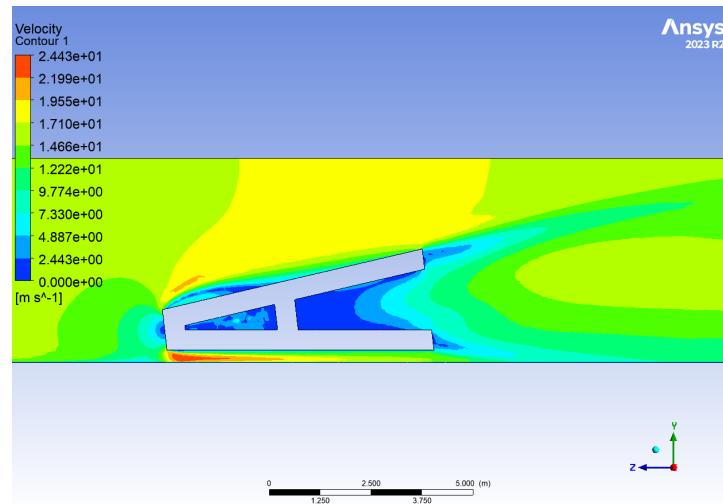


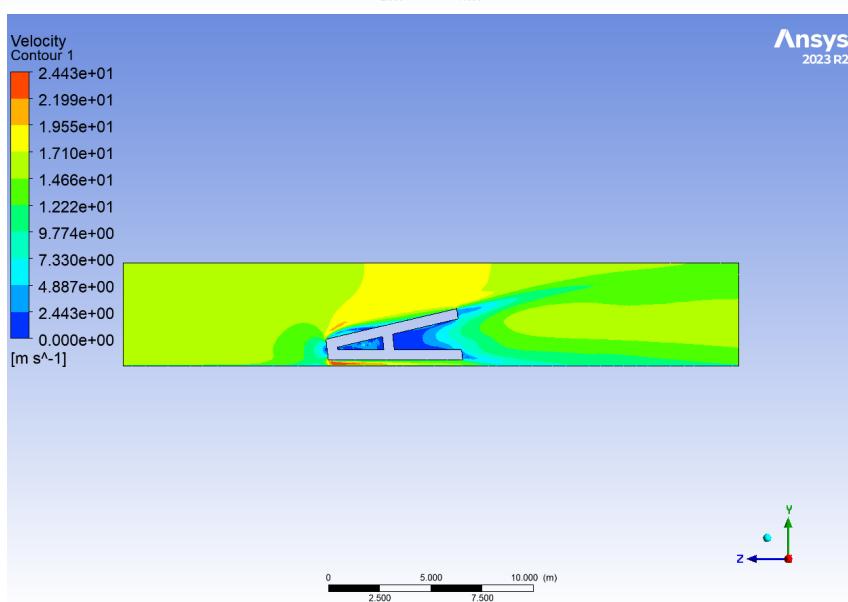
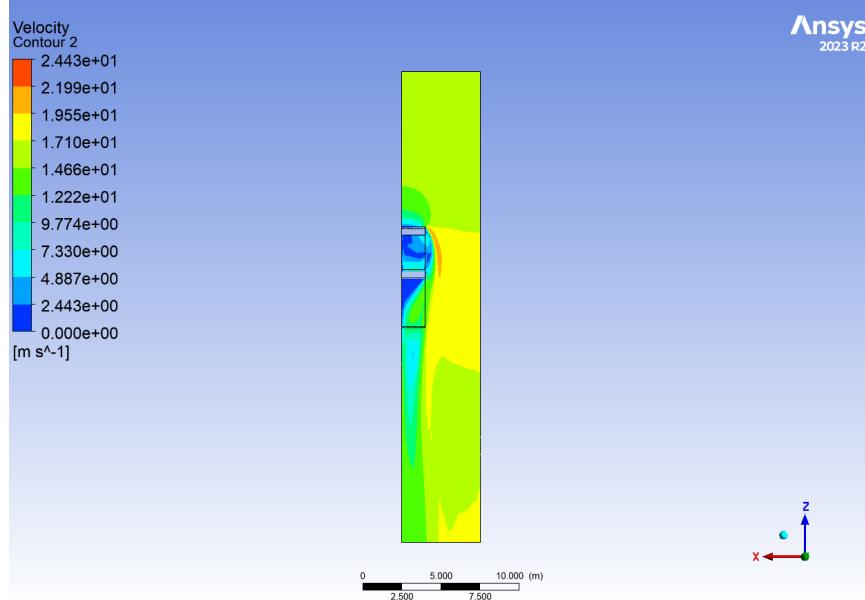
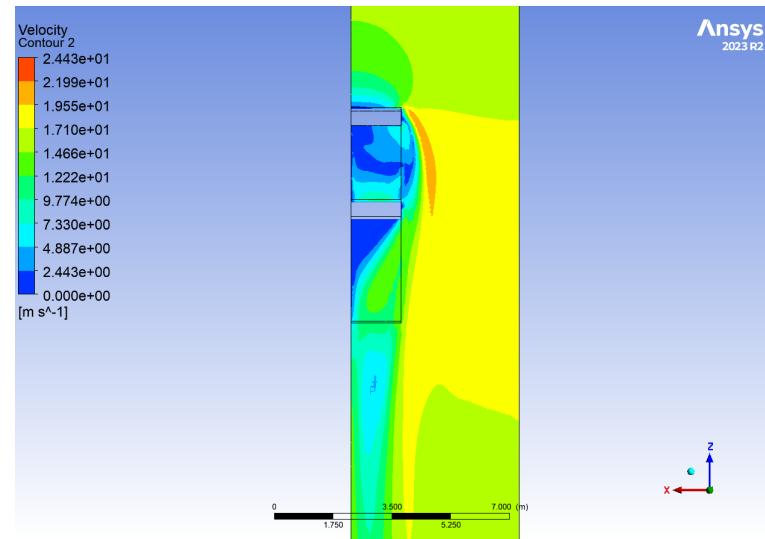


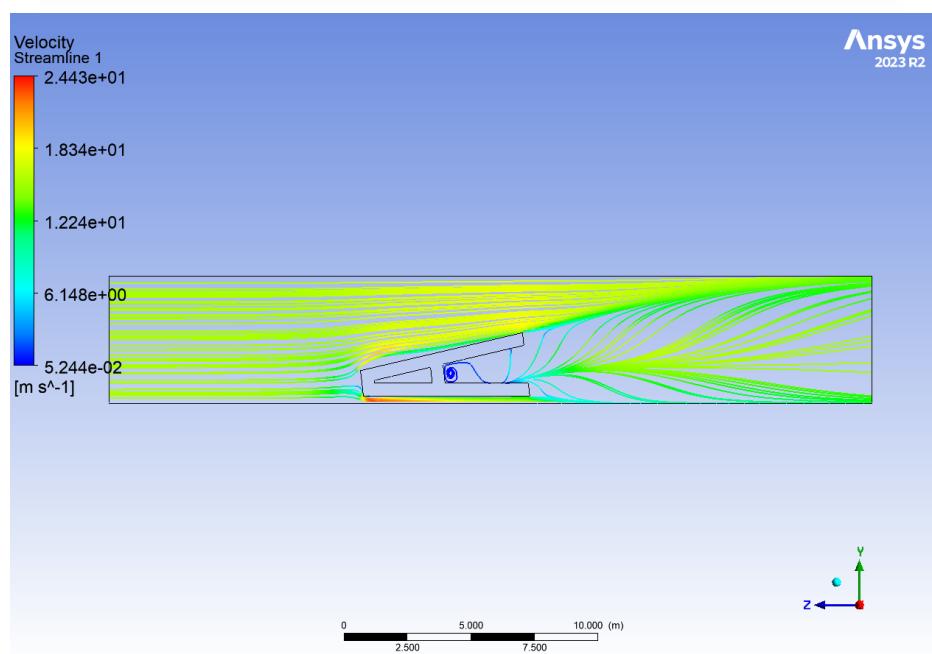
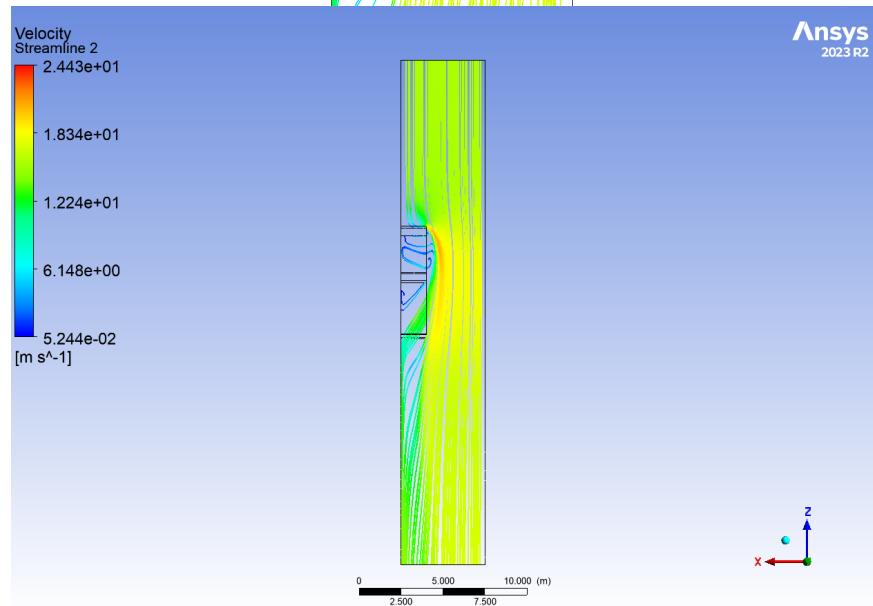
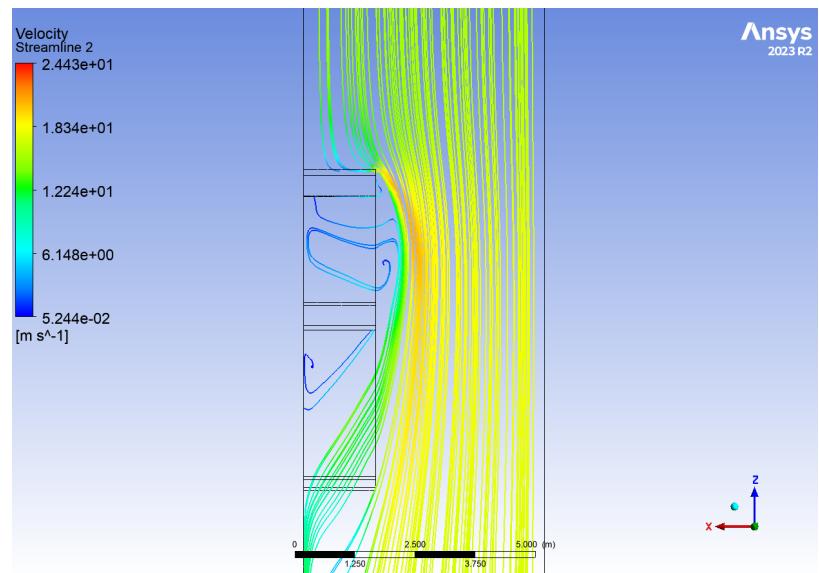


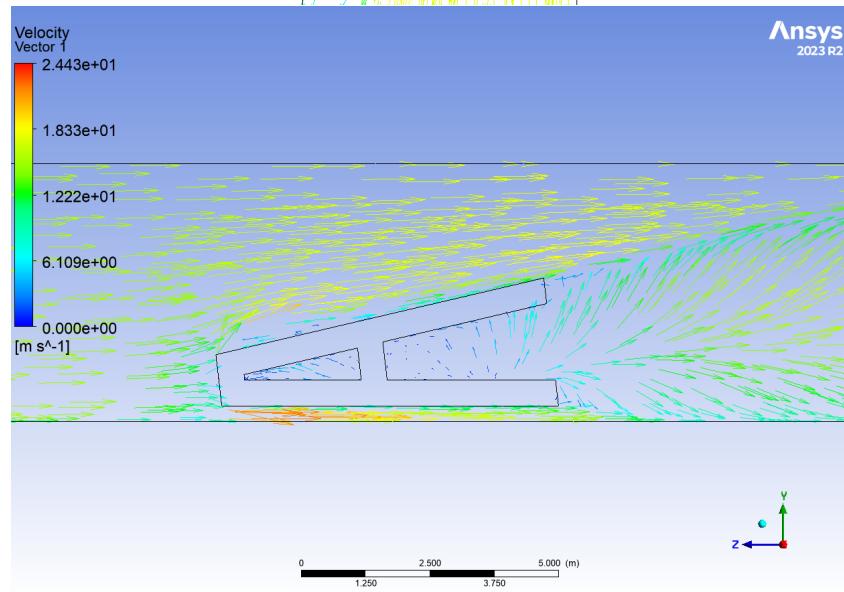
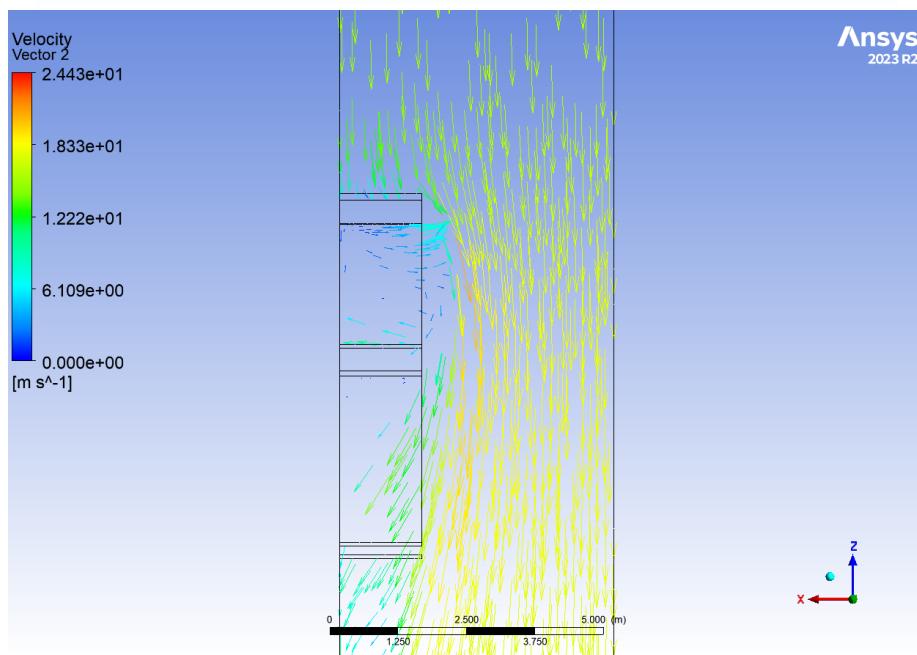
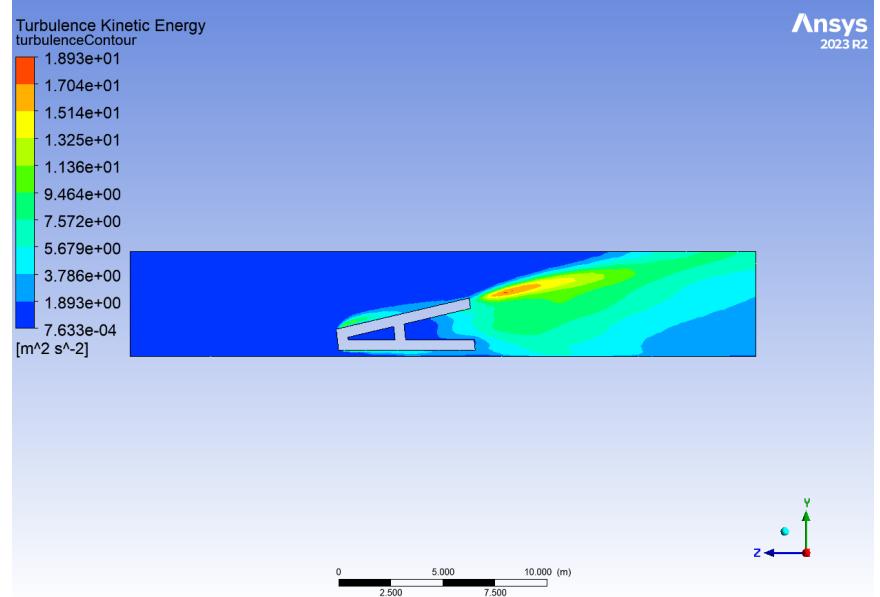


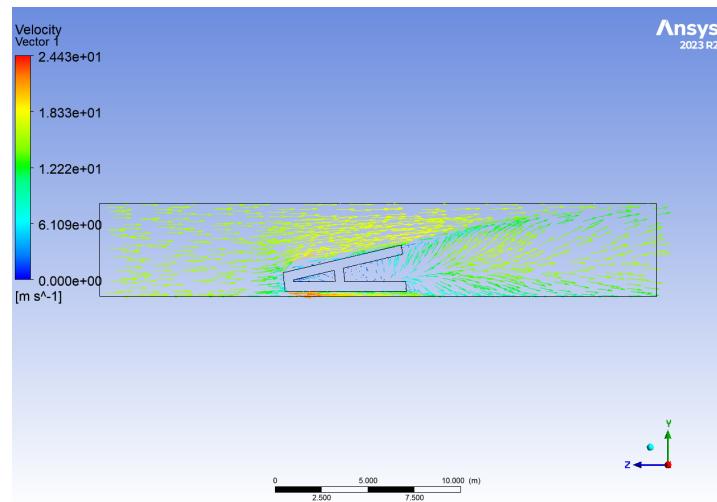
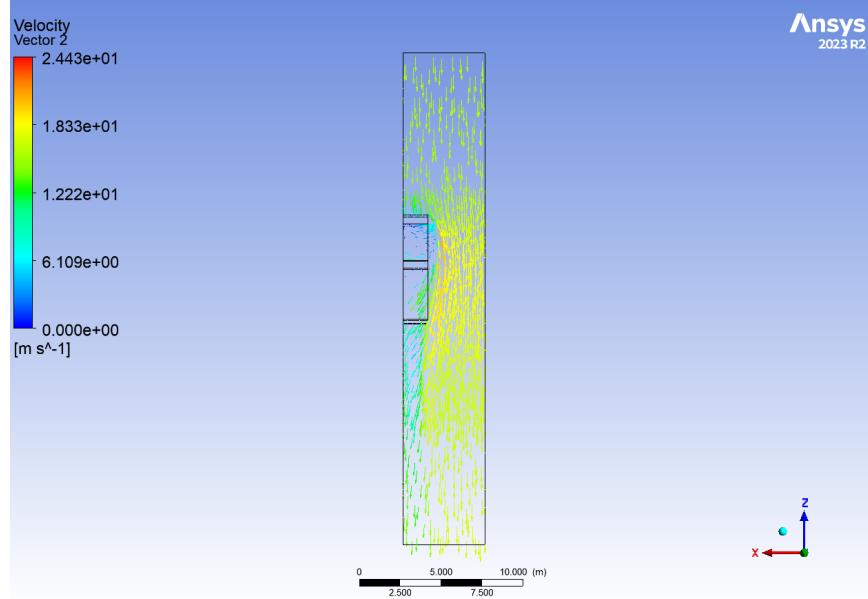
More Fine Mesh Figures:











References

- [1] "Ahmed Body" Notes
- [2] Class notes