Leaning Tower of Pisa

MME 4420 Report

Fiona Mackenzie

December 5, 2024

Table of Contents

1. **Overview**
2. **2D Scaling**
    2.1. Setup and Domain
    2.2. Reynolds Number Calculations
    2.3. Velocity Calculations
3. **2D Mesh Generation**
    3.1. Quality Analysis
    3.2. Mesh Sizing
4. **2D Simulations**
    4.1. Velocity 1
    4.2. Velocity 2
    4.3. Velocity 3
    4.4. Velocity 4
        4.4.1. K-Omega
        4.4.2. K-Epsilon
        4.4.3. K-Omega with 10% intensity
5. **3D Simulation**
    5.1. Setup
    5.2. Simulation results
    5.3. Analysis
6. **Results**
    6.1. Table of Results
    6.2. Force Coefficient vs. Reynolds Numbers
7. **Analysis**
    7.1. Discussion
    7.2. Conclusion

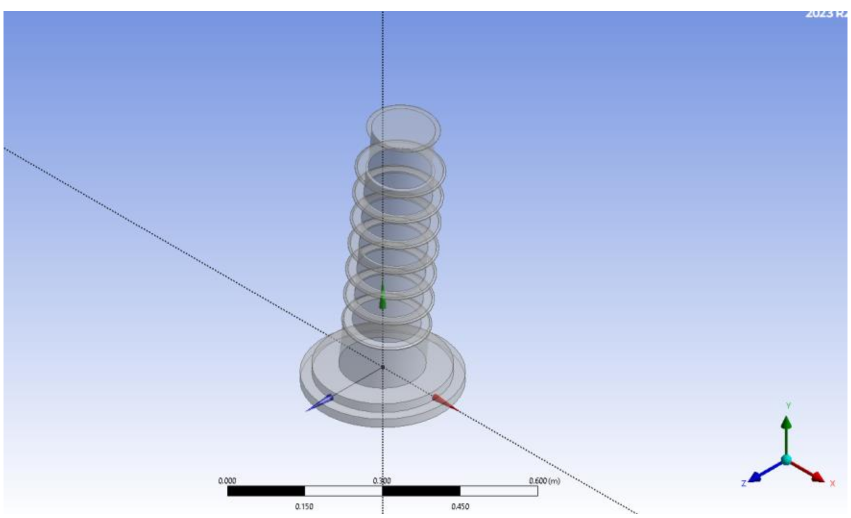
1. **Overview**

Analyzing the flow around buildings is an important step to ensuring safe structures, especially in areas which experience strong winds. In this project, the analysis of the aerodynamic forces caused by wind at different velocities are observed on the Leaning Tower of Pisa, using multiple computational fluid dynamics simulations. The objective is to simulate steady flow of wind around the Pisa Tower at specific velocities, and determine the pressure and viscosity forces acting on the building. From the forces determined in the 2D flow, the force coefficients are then calculated and compared to the Reynolds numbers. With the results from the 3D simulations, the 2D and 3D results are compared are discussed.

2. **Scaling**
    2.1. Setup and Domain:

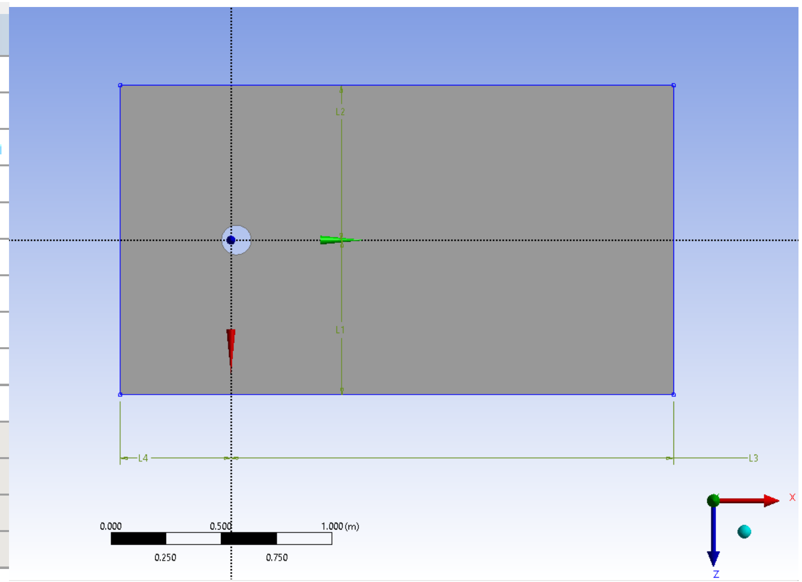
First, the stp file of the Leaning Tower of Pisa model was imported into fluent, and 2D geometry was set up in the DesignModeler. The diameter of the tower model was then found to be 0.32m. The domain was then created using a 2D horizontal slice of the smaller part of the diameter, as this is the geometry that makes up most of the building, representing the majority of the flow around the model. The dimensions were then set so that the domain was scaled large enough to capture the entire flow around the building, with about 15 times the diameter downstream of the model, and 5 times the diameter on either side of the model to ensure that the entire flow is captured during simulations.

Imported Model and Domain:



Sketching	Modeling
Details View	
Analysis Tools	
Analysis Tool	Bounding Box
Entity Set	1 Body
Length X	0.32 m
Length Y	0.62956 m
Length Z	0.32 m

Details View	
Details of Sketch1	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 4	
L1	0.7 m
L2	0.7 m
L3	2 m
L4	0.5 m
Edges: 4	
Line	Ln9
Line	Ln10
Line	Ln11
Line	Ln12



The diameter of the model was measured as 0.32m but the actual diameter of the Pisa Tower is 15.43m, so the Reynolds numbers were calculated at each velocity with the actual diameter. The velocities were then re-calculated with the new Reynolds numbers.

New Reynolds Numbers Calculations:

$$Re = \frac{U_o D}{v}$$

At the velocity = 0.12 m/s:

$$Re_1 = \frac{\left(0.12\frac{m}{s}\right)*(15.48m)}{\left(1.48e^{-5}\frac{m^2}{s}\right)}$$

$$Re_1 = 125\ 513.51$$

At the velocity = 2 m/s:

$$Re_2 = \frac{\left(2\frac{m}{s}\right) * (15.48m)}{\left(1.48e^{-5}\frac{m^2}{s}\right)}$$

$$Re_2 = 2\ 091\ 891.89$$

At the velocity = 4 m/s:

$$Re_3 = \frac{\left(4\frac{m}{s}\right) * (15.48m)}{\left(1.48e^{-5}\frac{m^2}{s}\right)}$$

$$Re_3 = 4\ 183\ 783.78$$

At the velocity = 8 m/s:

$$Re_4 = \frac{\left(8\frac{m}{s}\right) * (15.48m)}{\left(1.48e^{-5}\frac{m^2}{s}\right)}$$

$$Re_4 = 8\ 367\ 567.57$$

New velocities Calculations:

$$U_o = \frac{(Re)(v)}{(D)}$$

$$U_1 = \frac{(125\ 513.51) * (1.784e^{-5}\frac{m^2}{s})}{(0.32m)}$$

$$U_1 = 5.805\ m/s$$

$$U_2 = \frac{(2\ 091\ 891.89) * (1.784e^{-5}\frac{m^2}{s})}{(0.32m)}$$

$$U_2 = 96.75\ m/s$$

$$U_3 = \frac{(4\ 183\ 783.78) * (1.48e^{-5}\frac{m^2}{s})}{(0.32m)}$$

$$U_3 = 193.5\ m/s$$

$$U_4 = \frac{(8\ 367\ 567.57) * (1.48e^{-5}\frac{m^2}{s})}{(0.32m)}$$

$$U_4 = 387.0\ m/s$$

3. Meshing

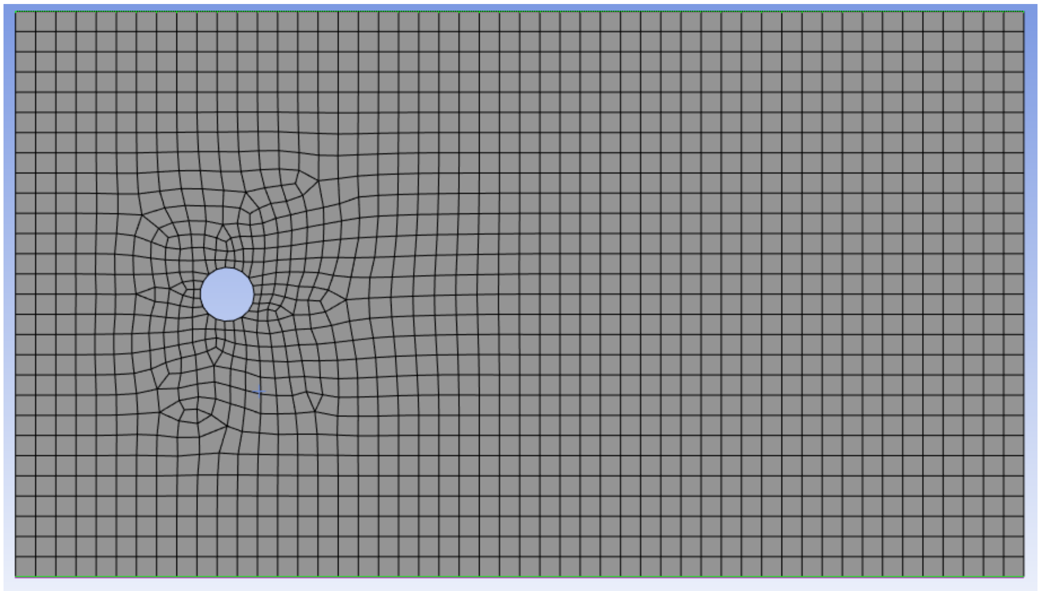
3.1. Quality Analysis

The initial mesh was generated using the automatic meshing method. From this mesh, the quadrilateral dominant method was generated, with an uneven pattern forming around the circular building cut-out with extremely skewed cells and poor orthogonality. The uneven cell structure indicated that triangles may be a better option for this geometry.

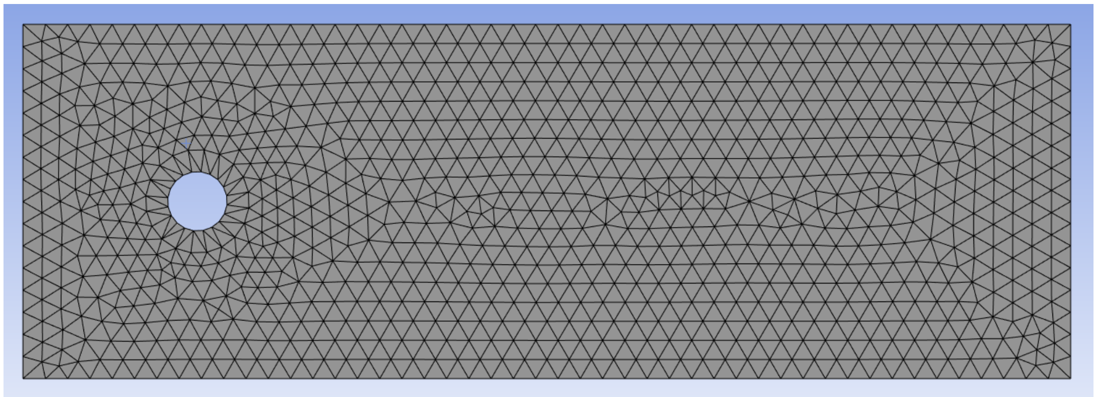
The second mesh generated was a triangular mesh. This mesh initially looks less deformed and the cells are less skewed than that of the quadrilateral, however the cells around the circle are skewed and elongated, due to the poor aspect ratio these cells would negatively impact the simulations. In order to mitigate these issues around the building cut-out, an inflation layer can be added to the circle, to provide better element quality and smoothness. When running the simulation for the triangles mesh, it was clear that symmetry must be applied to the walls of the domain so that only the geometry of the Pisa Tower will be simulated.

Generating the third mesh, an inflation layer was added to the circular boundary of the mesh, the transition ratio for the inflation layer was initially defaulted as 0.27, but it had to be increased to generate cells with a lower aspect ratio. The transition ratio was then increase to 0.7, this created a double layer of nice even cells, but it also generated some small distorted cells when transitioning to the triangular mesh. In order to smooth the transition, the ratio was lowered from 0.7 to 0.4. With a transition ratio of 0.4 the double layer has a slightly worse aspect ratio, but it does not generate the small distorted cell layer, as it transitions to the rest of the mesh.

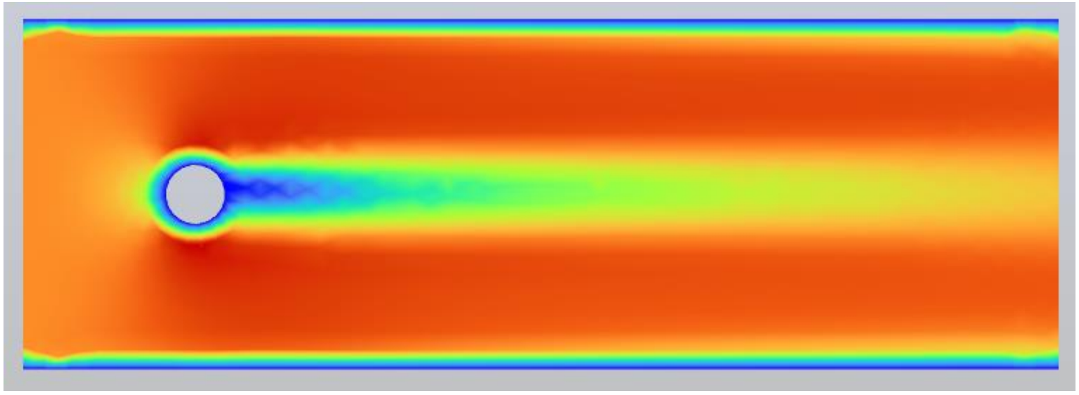
Quadrilateral Dominant Mesh:



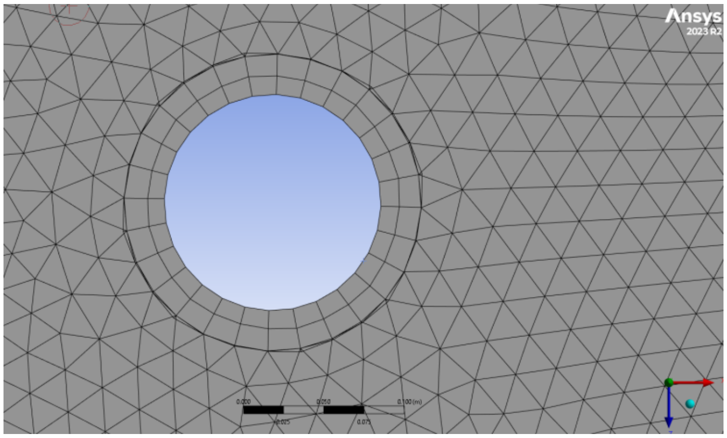
Triangles Mesh (without inflation layer):



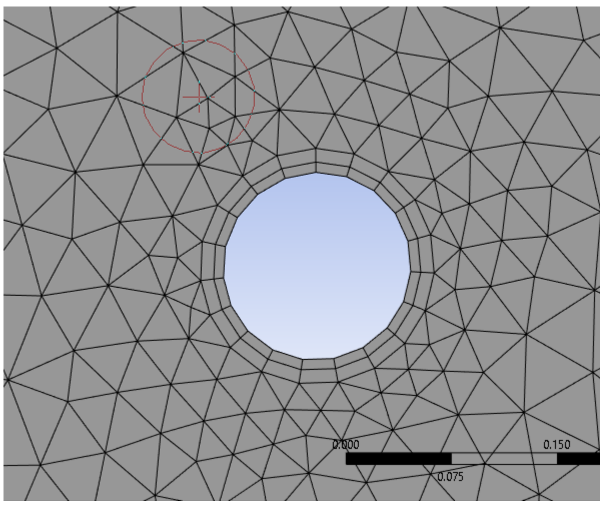
Triangles Mesh without symmetry:



Inflation layer, transition ratio = 0.7



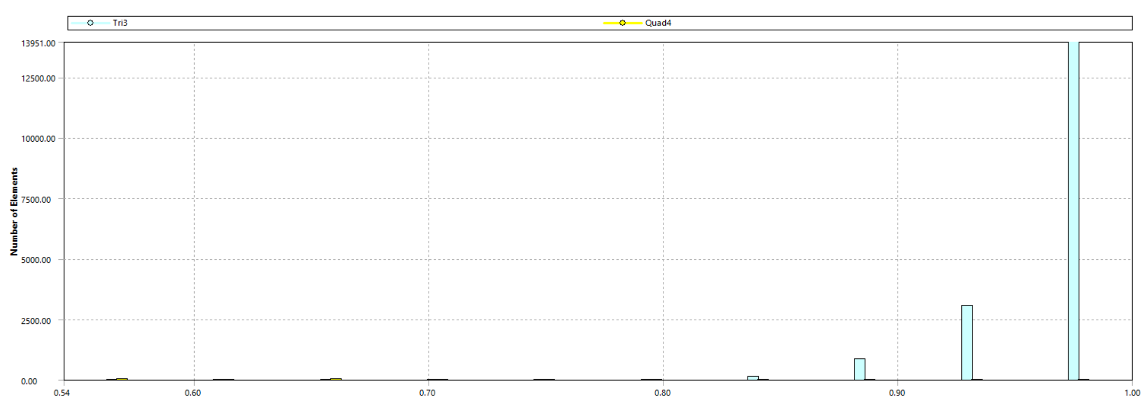
Inflation layer, transition ratio=0.4



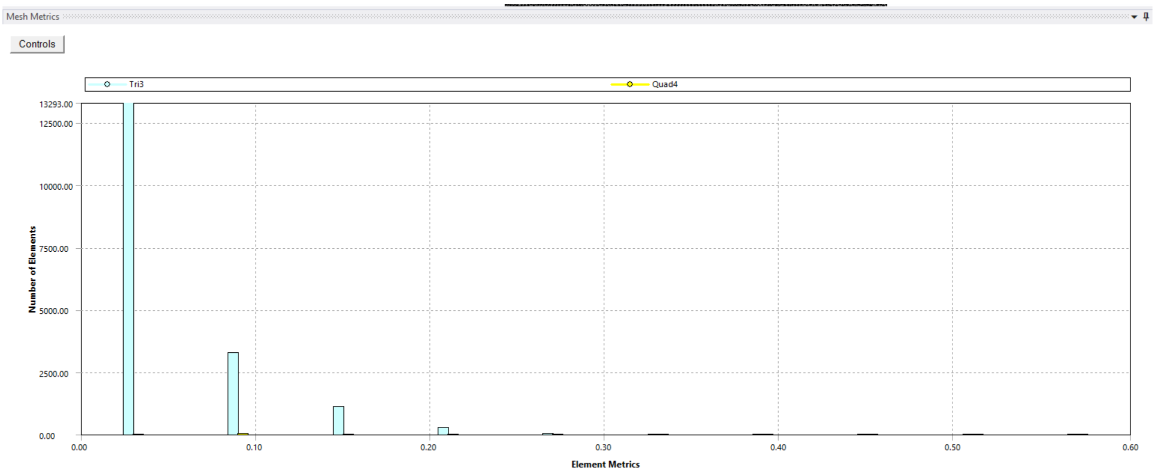
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Suppressed	No
Boundary Scoping Method	Geometry Selection
Boundary	1 Edge
Inflation Option	Smooth Transition
Transition Ratio	0.4
Maximum Layers	2
Growth Rate	1.2
Inflation Algorithm	Pre

Element Quality:

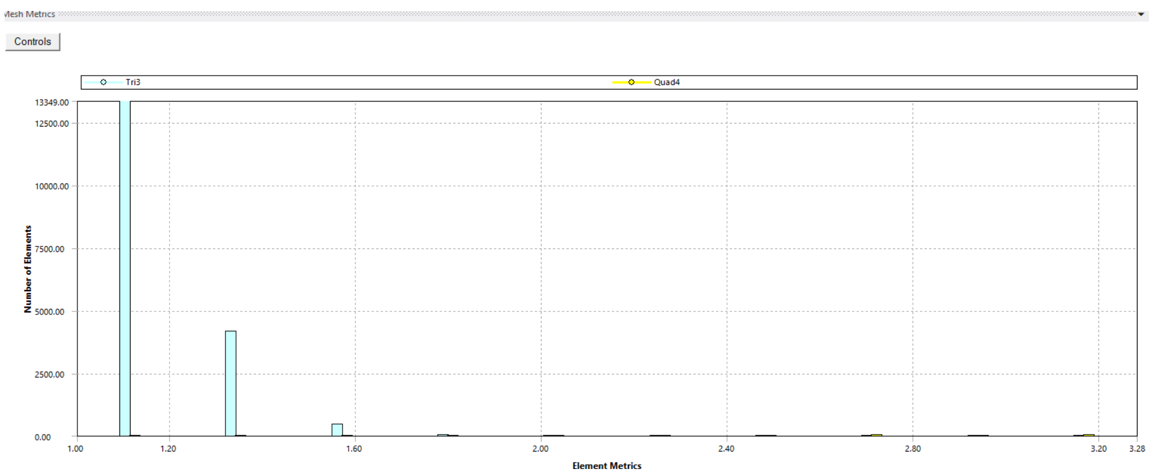
Skewness:



Aspect Ratio:



Orthogonal Quality:



3.2. Mesh Sizing

The mesh was initially generated with an automatic mesh, with the default element size which was way too big, causing the overall mesh to have extreme skewness problems. The mesh was initially set to a meshing size of 0.1. The mesh was then lowered and refined by 50% until the ideal mesh size was achieved. Once the mesh was refined enough to the point where it no longer changes the results in any significant way, grid-independence was achieved. In order to achieve grid-independence, a 5% or smaller change should be achieved between the two meshes. The first 50% drop between 0.1 and 0.05 achieves a 5.8853% difference, and the second 50% drop between 0.05 and 0.025 achieves a 0.5086% difference which is much less than 5%, indicating grid-independence. From these final results, a mesh with a size of 0.025m was selected, with triangles and an inflation layer with a transition ratio of 0.4, to be used for all of the 2D simulations.

50% reduction of mesh with percent difference calculation:

Mesh Size Range	Coarse Mesh - Force 1	Refined Mesh - Force 2	% Difference
0.1 - 0.05	2.05787	1.94349	5.8853
0.05 - 0.025	1.94349	1.9534251	0.5086

Forces for mesh size = 0.1m:

```
Forces - Direction Vector (1 0 0)
                          Forces [N]
Zone                      Pressure        Viscous         Total
pisa                      2.0287473       0.029122878     2.0578702
------------------------- --------------- --------------- ---------------
Net                       2.0287473       0.029122878     2.0578702
```

Forces for mesh size = 0.05m:

```
Forces - Direction Vector (1 0 0)
                          Forces [N]
Zone                      Pressure        Viscous         Total
pisa                      1.912164        0.03132765      1.9434916
------------------------- --------------- --------------- ---------------
Net                       1.912164        0.03132765      1.9434916
```

Forces for mesh size = 0.025m:

```
Forces - Direction Vector (1 0 0)
                          Forces [N]
Zone                      Pressure        Viscous         Total
pisa                      1.9238514       0.029573718     1.9534251
------------------------- --------------- --------------- ---------------
Net                       1.9238514       0.029573718     1.9534251
```

4. 2D Simulations

For each of the simulations, different velocities are used. The four different velocities were given as: 1.2m/s, 2m/s, 4m/s and 8m/s, but when scaled up to the actual building size these velocities became: 5.805m/s, 96.75m/s, 193.5m/s and 387m/s. For all four of the 2D velocity simulations, the same domain and mesh was used. The objective of these simulations is to compare and analyze how the air flows at each of these velocities, determined by forces, around the Pisa Tower horizontally.

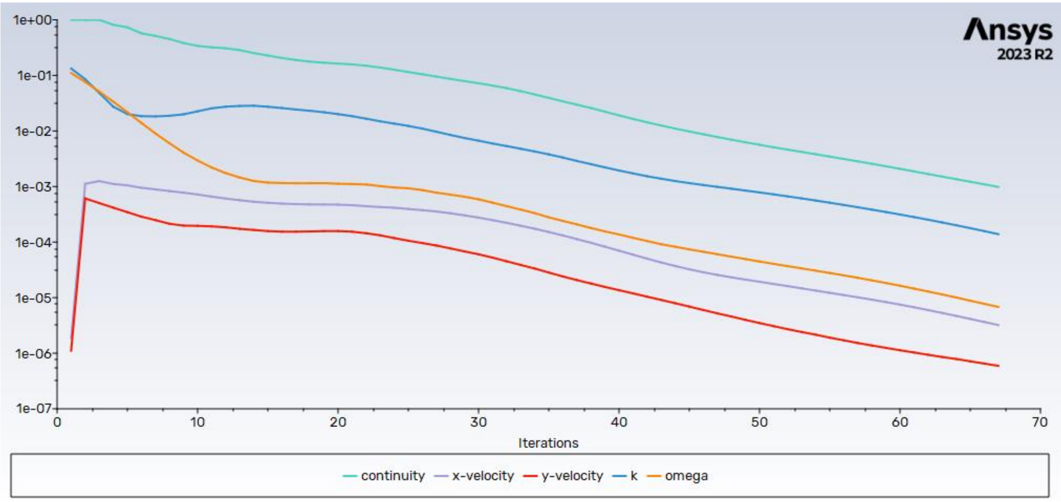
Each simulation is setup in its own way to ensure the maximum quality of the flow results for the viscosity and pressure forces in both the x and y directions. The results of each simulation provide a scaled residuals plot to show how the results converged and behaved. The results also show the x-velocity contour plot, a total pressure contour plot, a velocity vector plot and the forces in the x and y directions for pressure and viscosity.

4.1.Velocity 1

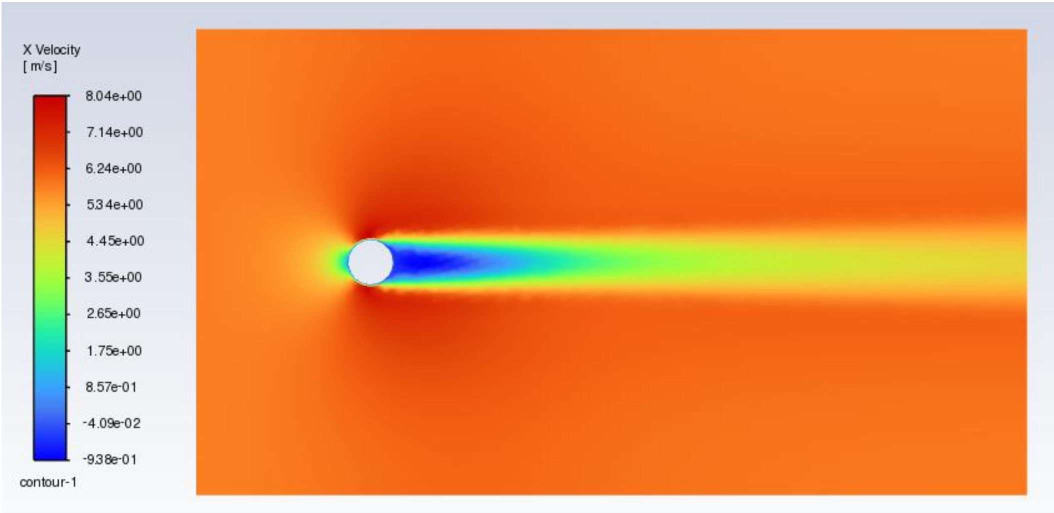
For the first velocity, due to the large Reynolds number of 125513.51, the flow was set to turbulent using k-omega. The material was selected as air for the cell zone conditions. The boundary conditions were then setup with the inlet set as a velocity inlet with a velocity of 5.805m/s in the x-direction. The right and left edges of the domain were set as symmetry in order to not disrupt the flow. The outlet was set as a pressure outlet, the cut-out tower was set as a wall and the inner-surface was set as interior. The method was kept as a COUPLED flow as it produces a higher-accuracy flow, as all of the Reynolds numbers used are extremely high.

After simulating the first velocity, the scaled residuals showed that the flow converges at about 67, which is where the solution was stabilized. This is a relatively low convergence number indicating that the flow is well-converged with little error producing reliable results. The appearance of X-velocity vector plot and the total pressure plot look well executed and even indicating that the correct domain and boundary conditions were selected. The arrows on the velocity vector plot represent the direction and magnitude of the velocity, showing clearly exactly what the velocity and direction of the flow is doing at all parts in the domain.

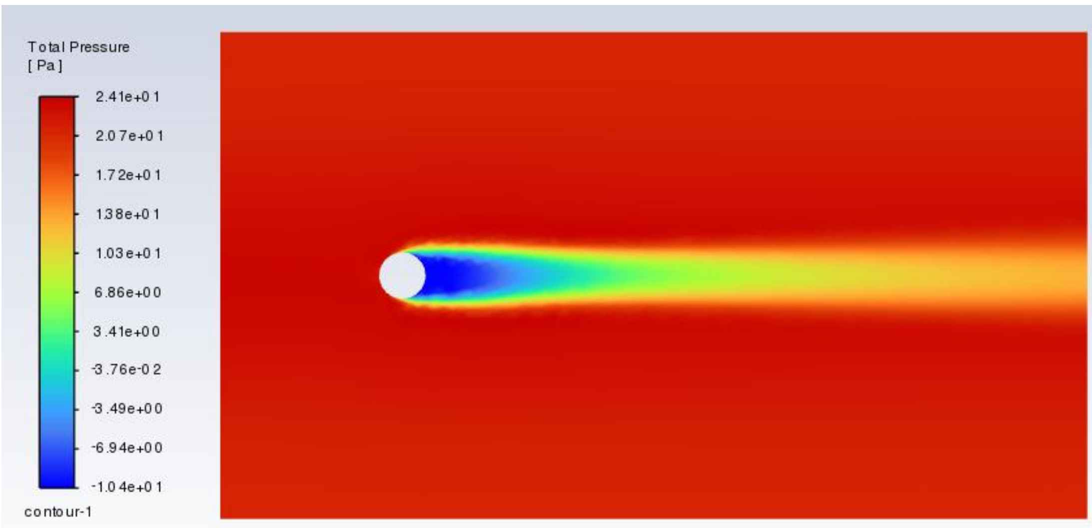
Scaled Residuals:



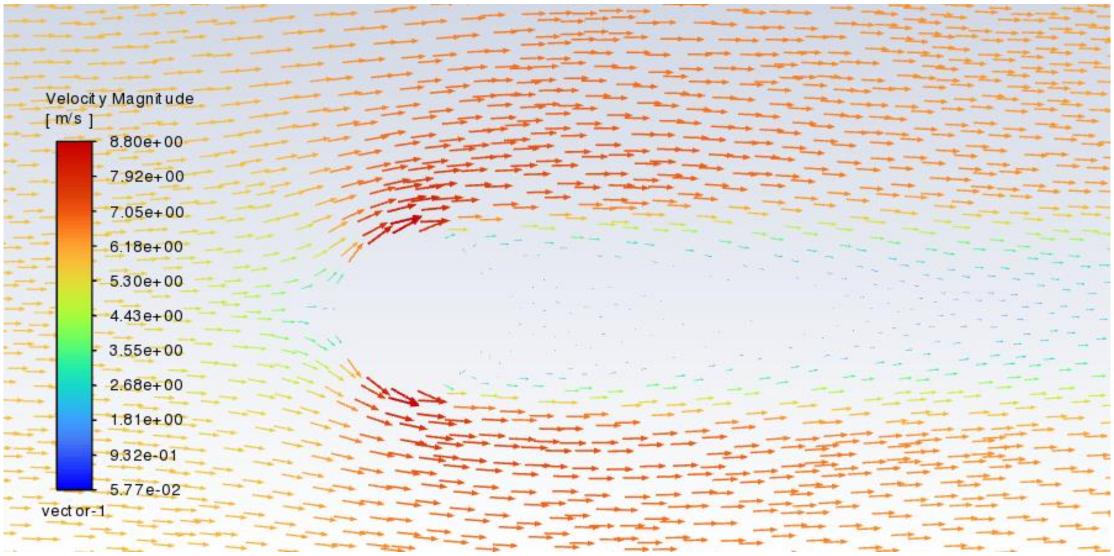
X-Velocity Contour Plot:



Total Pressure Contour Plot:



Velocity Vector Plot:



Forces:

Forces - Direction Vector (1 0 0)			
	Forces [N]		
Zone	Pressure	Viscous	Total
pisa	1.9238514	0.029573718	1.9534251
Net	1.9238514	0.029573718	1.9534251

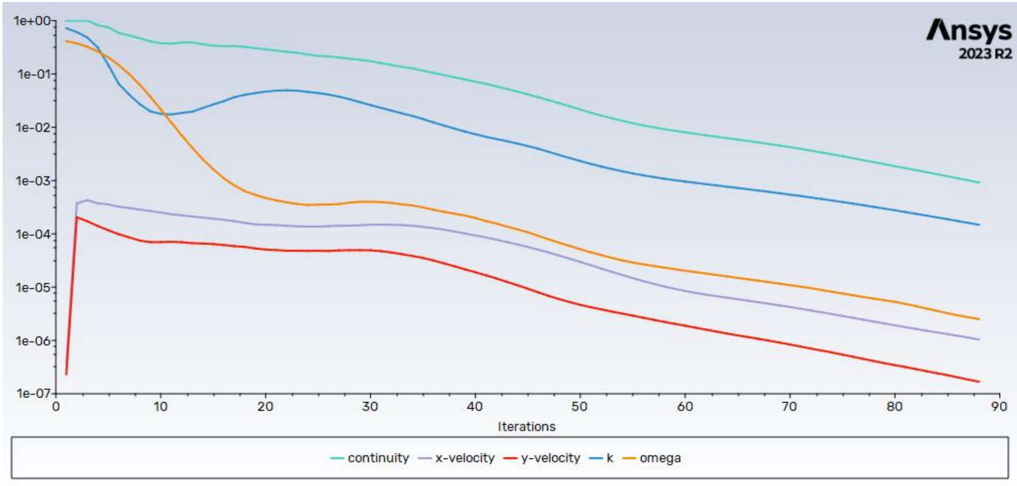
Forces - Direction Vector (0 1 0)			
	Forces [N]		
Zone	Pressure	Viscous	Total
pisa	0.11489546	0.00017460831	0.11507006
Net	0.11489546	0.00017460831	0.11507006

4.2.Velocity 2

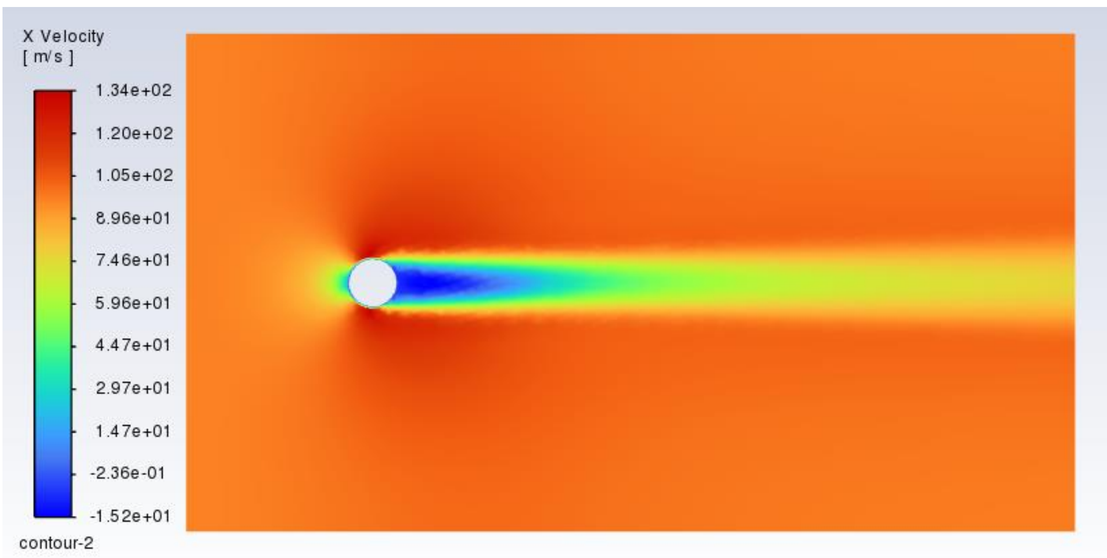
For the second velocity the flow was set as turbulent k-omega due to the high Reynolds number. The boundary conditions and set-up were then setup in the same way as the first velocity simulation with a coupled flow method as the velocity is quite high, and the inlet velocity was set to 96.75m/s.

From the results of this flow, looking at the scaled residuals shows that the simulation converged at 88, so the simulation was well converged. The x-velocity and total pressure contour plots appear similar to that of the first velocity, indicating that the flow is accurately represented.

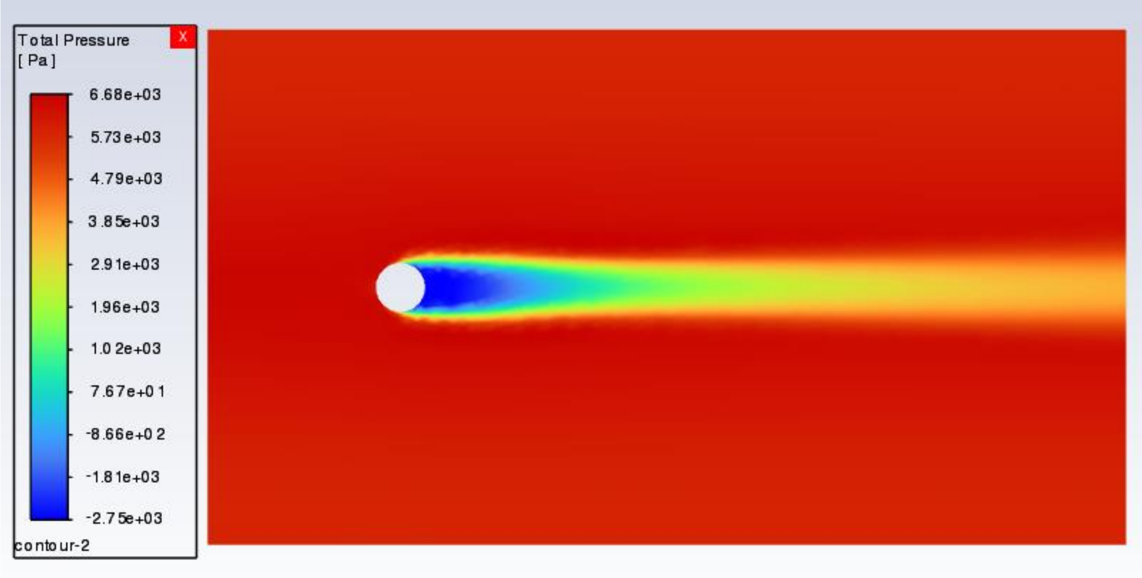
Scaled Residuals:



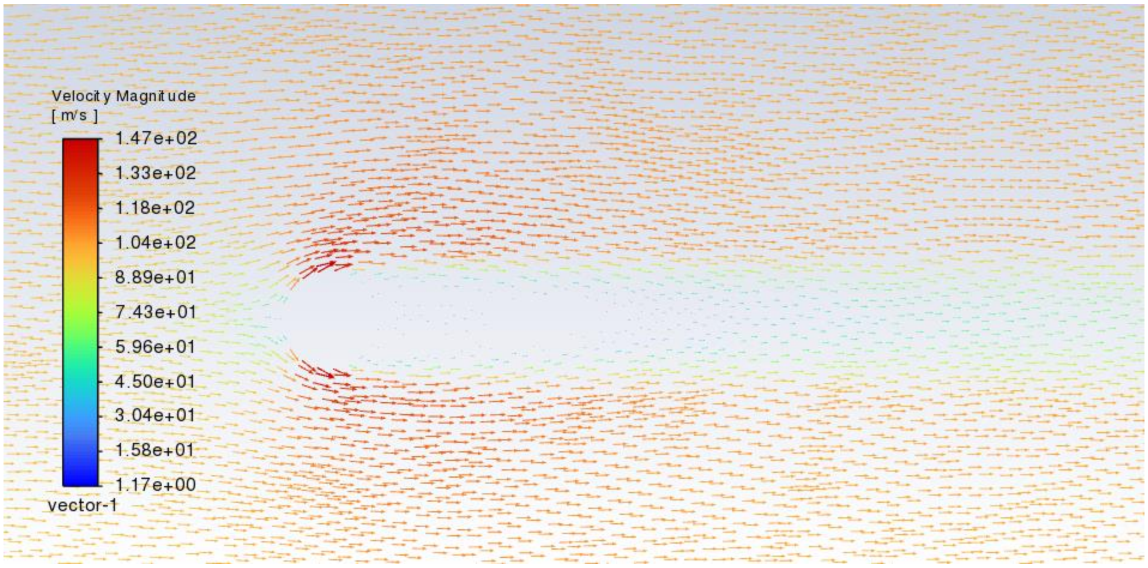
X-Velocity Contour Plot:



Total Pressure Contour Plot:



Velocity Magnitude Vector plot:



Forces:

```
Forces - Direction Vector (1 0 0)
                          Forces [N]
Zone                      Pressure        Viscous         Total
pisa                      516.4201        2.8427954       519.2629
------------------------- --------------- --------------- ---------------
Net                       516.4201        2.8427954       519.2629
```

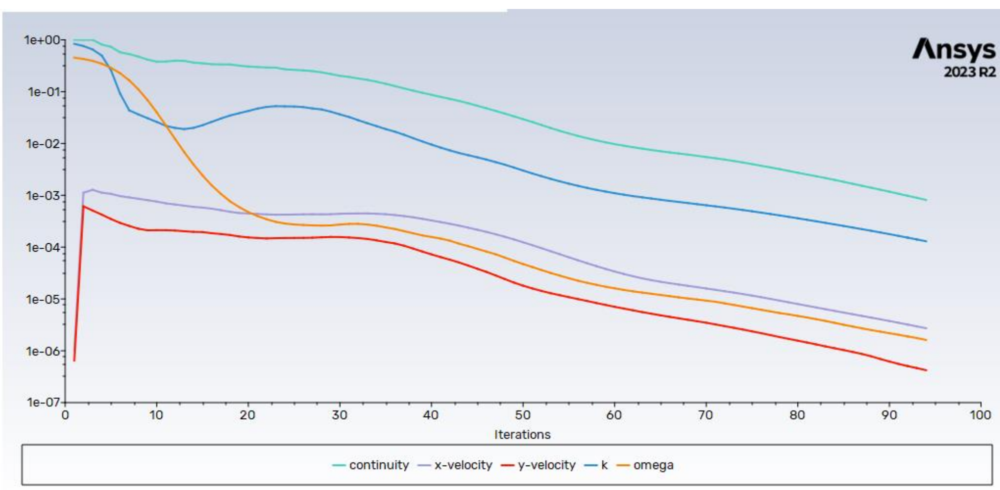
```
Forces - Direction Vector (0 1 0)
                          Forces [N]
Zone                      Pressure        Viscous         Total
pisa                      21.327484       0.020250246     21.347734
------------------------- --------------- --------------- ---------------
Net                       21.327484       0.020250246     21.347734
```

4.3.Velocity 3

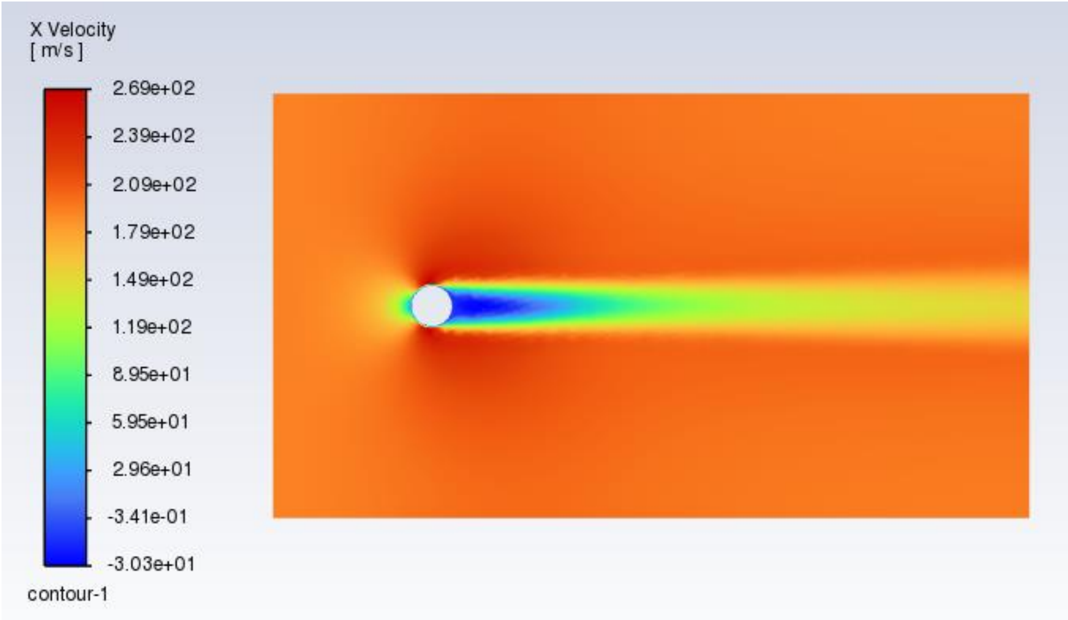
For the third velocity, the flow was set as turbulent k-omega. The boundary conditions and set-up were performed the same way as the first and second velocity simulations, except the inlet velocity was set to 193.5m/s in the x-direction.

From the results, the scaled residuals show that the simulation converged at about 94, which is higher than the first two but still converged very well. As the velocity is increased, the maximum total pressure is increasing as would be expected. For the contour plots, overall, they look similar but as the velocity is increased the streamline downstream is getting narrower.

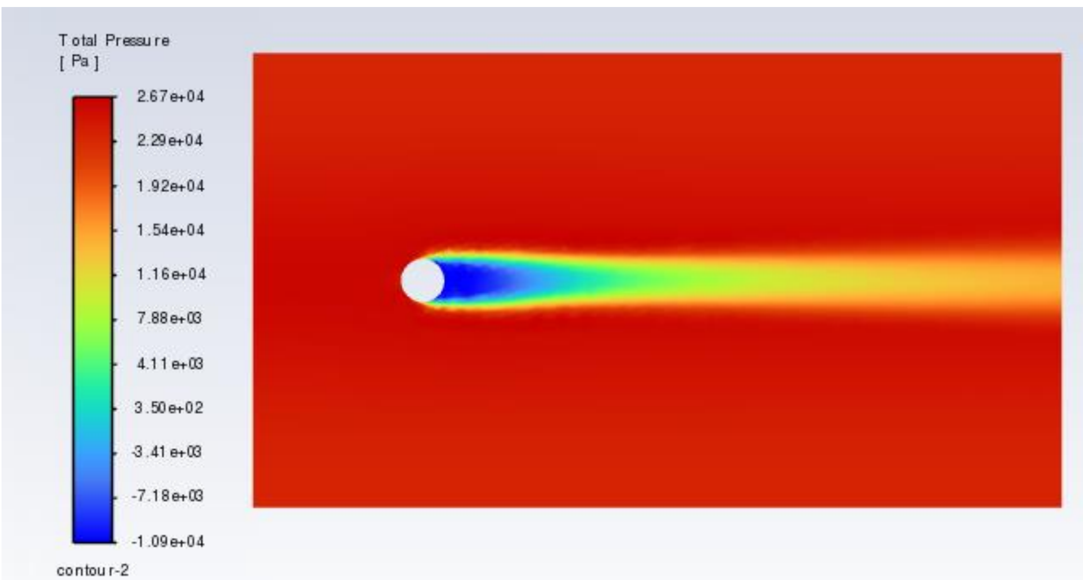
Scaled Residuals:



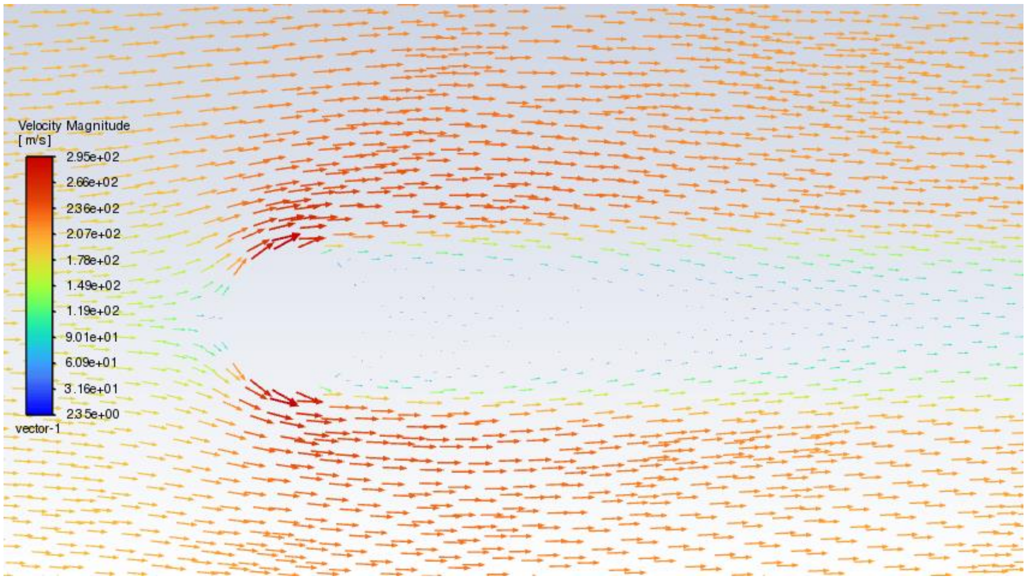
X-Velocity Contour Plot:



Total Pressure Contour Plot:



Velocity Magnitude Vector plot:



Forces:

Forces - Direction Vector (1 0 0)	Forces [N]		
Zone	Pressure	Viscous	Total
pisa	2055.5447	8.9231329	2064.4678
Net	2055.5447	8.9231329	2064.4678

Forces - Direction Vector (0 1 0)	Forces [N]		
Zone	Pressure	Viscous	Total
pisa	75.236923	0.072064191	75.308987
Net	75.236923	0.072064191	75.308987

4.4. Velocity 4

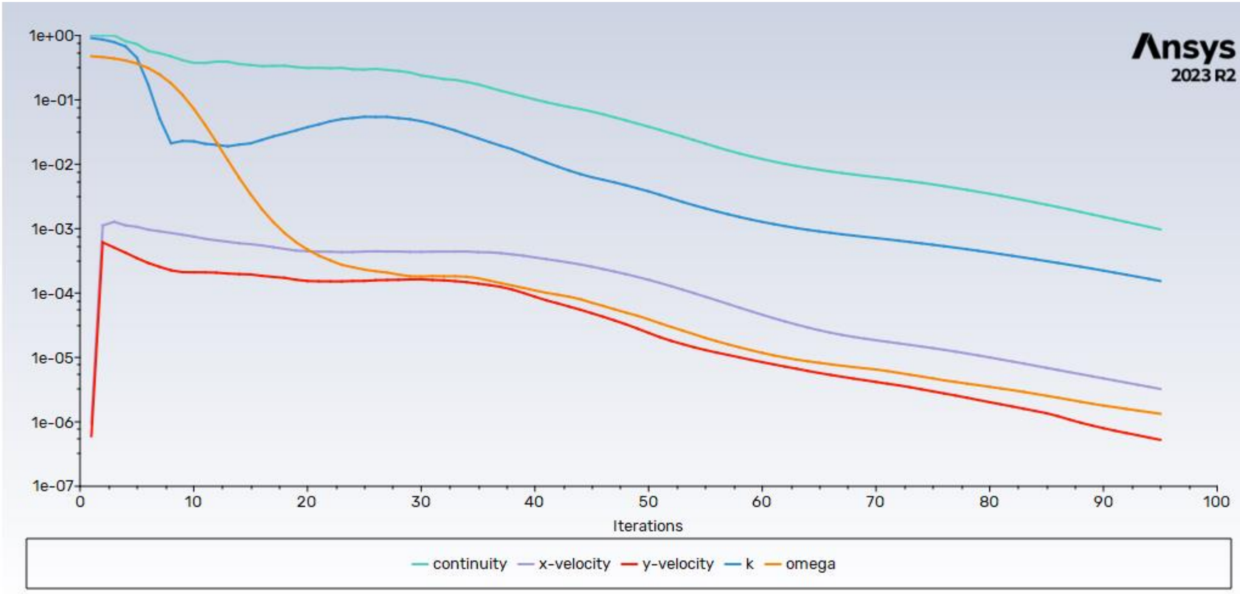
The final velocity is split into three different simulations. The first simulation is looking at the effects of the flow when using a regular k-omega turbulence, the second simulation uses a k-epsilon turbulence, and the final simulation uses a k-omega with 10% intensity turbulence. The results of the three simulations are then to be compared.

4.4.1. K-omega turbulence

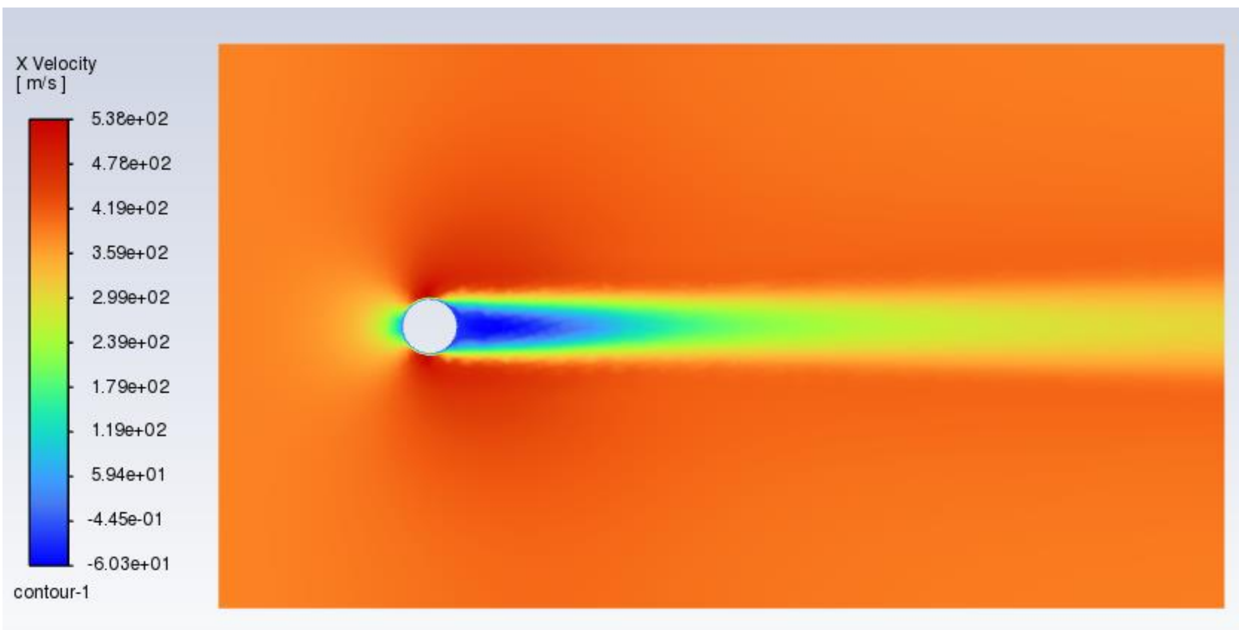
For the final velocity, the flow was initially set as turbulent k-omega. The boundary conditions and set-up the same way as the previous simulations, and the inlet velocity was set to 387m/s in the x-direction.

From the results, the scaled residuals show the simulation converged at about 96. The contour plots appearances were very similar to the previous flows.

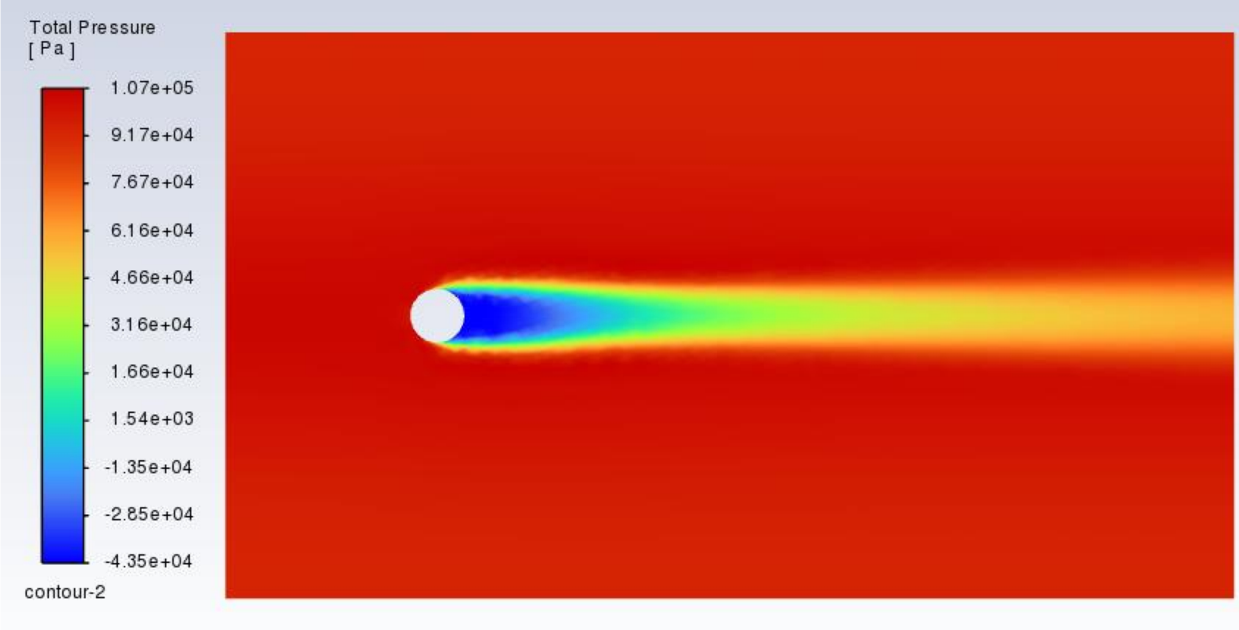
Scaled Residuals:



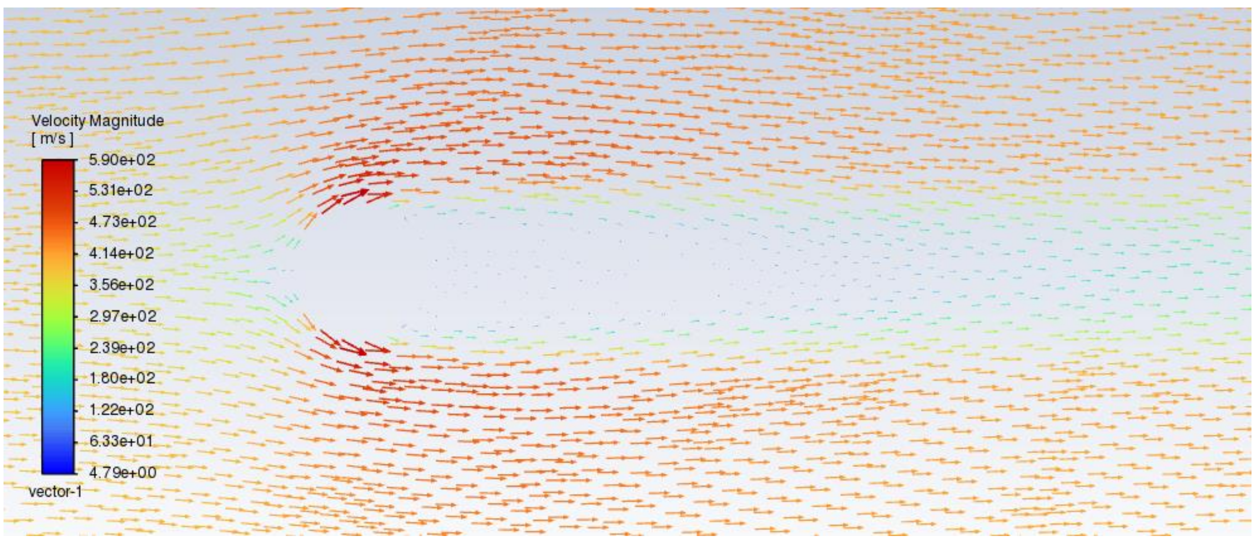
X- Velocity Contour Plot:



Total Pressure Contour Plot:



Velocity Magnitude Vector plot:



Forces:

```
Forces - Direction Vector (1 0 0)
                          Forces [N]
Zone                      Pressure        Viscous         Total
pisa                      8189.876        28.408573       8218.2845
------------------------- --------------- --------------- ---------------
Net                       8189.876        28.408573       8218.2845
```

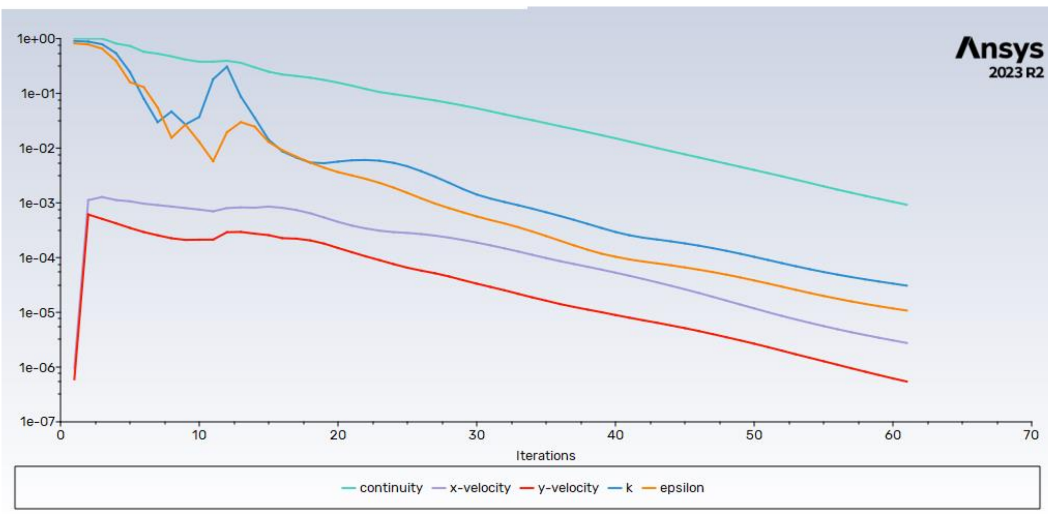
```
Forces - Direction Vector (0 1 0)
                          Forces [N]
Zone                      Pressure          Viscous           Total
pisa                      279.02054         0.26953065        279.29007
------------------------- ---------------   ---------------   ---------------
Net                       279.02054         0.26953065        279.29007
```

4.4.2. K-epsilon turbulence

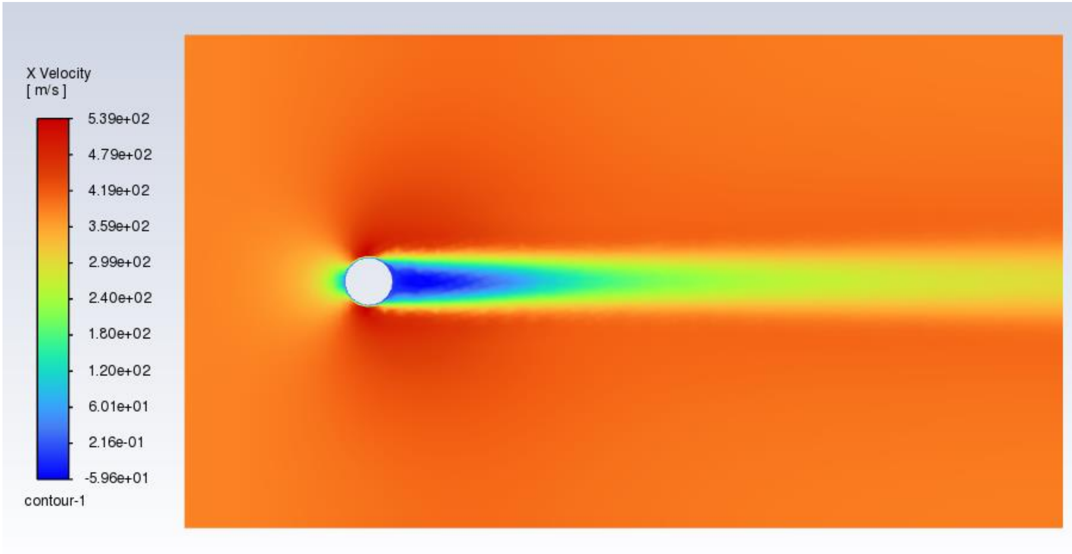
The next simulation was set as turbulent k-epsilon. The boundary conditions and set-up the same way as the previous simulation, and the inlet velocity was set to 387m/s in the x-direction.

From the results, the scaled residuals show the simulation converged at about 61, which is much lower than the k-omega simulation, this lower convergence value indicates that the results may be more accurate and precise as the early convergence means the change in iterations is smaller. The forces are also different, indicating a change in the flow's behavior. The contour plots are the same as the k-omega simulation.

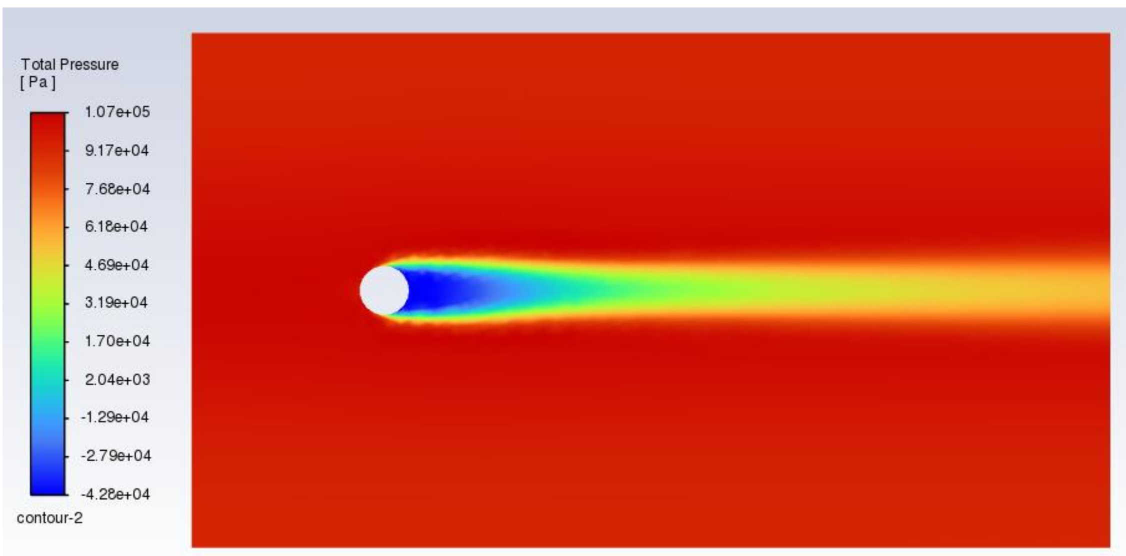
Scaled Residuals:



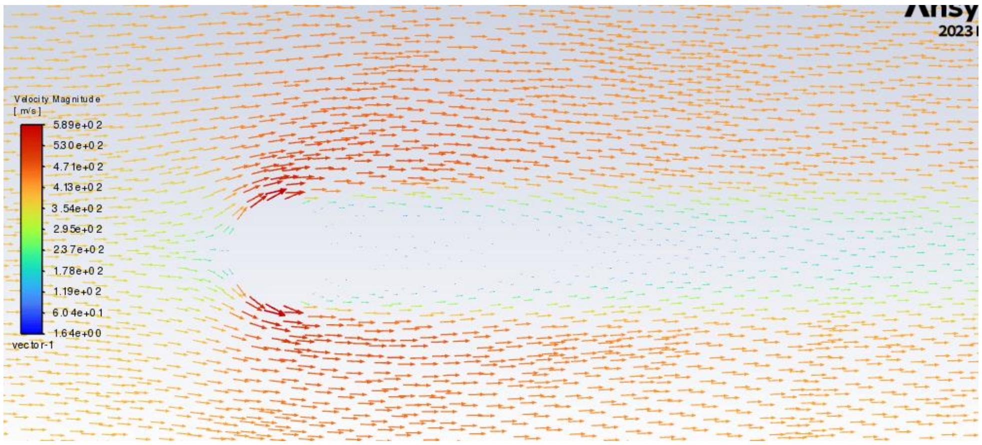
X-Velocity Contour Plot:



Total Pressure Contour Plot:



Velocity vector plot:



Forces:

Forces - Direction Vector (1 0 0)

Zone	Forces [N] Pressure	Viscous	Total
pisa	8145.3755	30.377964	8175.7535
Net	8145.3755	30.377964	8175.7535

Forces - Direction Vector (0 1 0)

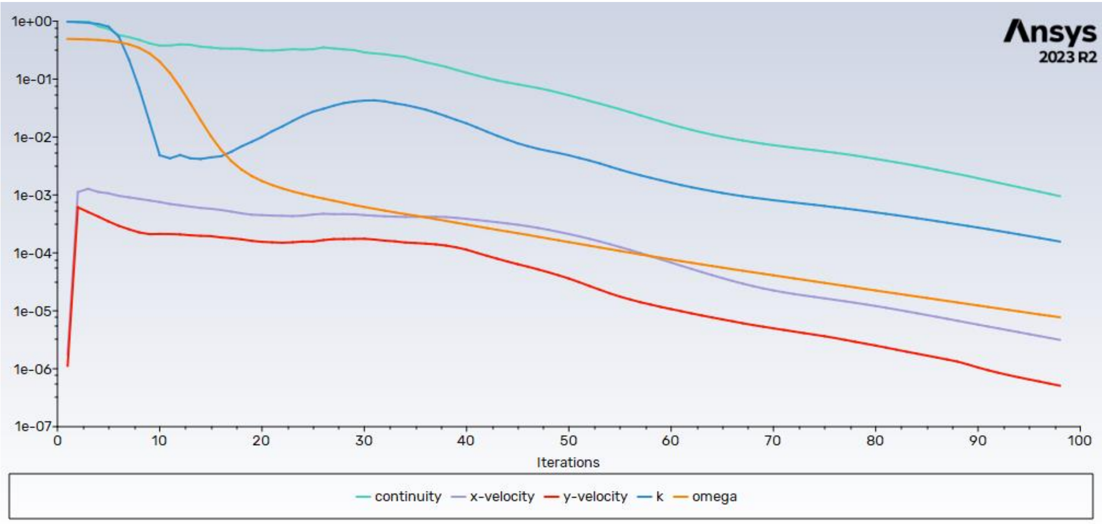
Zone	Forces [N] Pressure	Viscous	Total
pisa	-22.884998	0.35477841	-22.53022
Net	-22.884998	0.35477841	-22.53022

4.4.3. K-omega with 10% turbulence intensity

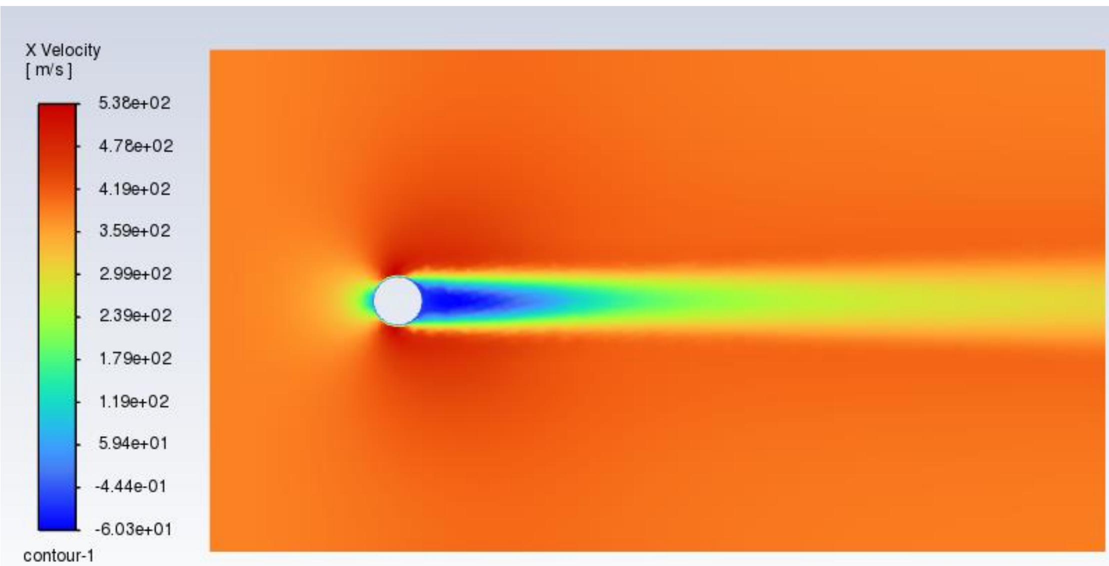
For the third simulation, k-omega was selected with a 10% turbulence intensity. The boundary conditions were set the same as the previous simulations, and the inlet velocity was set to 387m/s.

From the results, the scaled residuals show that simulation converged at 98, which is higher than the k-epsilon and the k-omega simulations. The higher convergence value may indicate that these results may less accurate than the previous simulations. The forces are different from the k-epsilon forces, but they are very similar to the k-omega forces. The contour plots are the same as the k-omega and k-epsilon simulations.

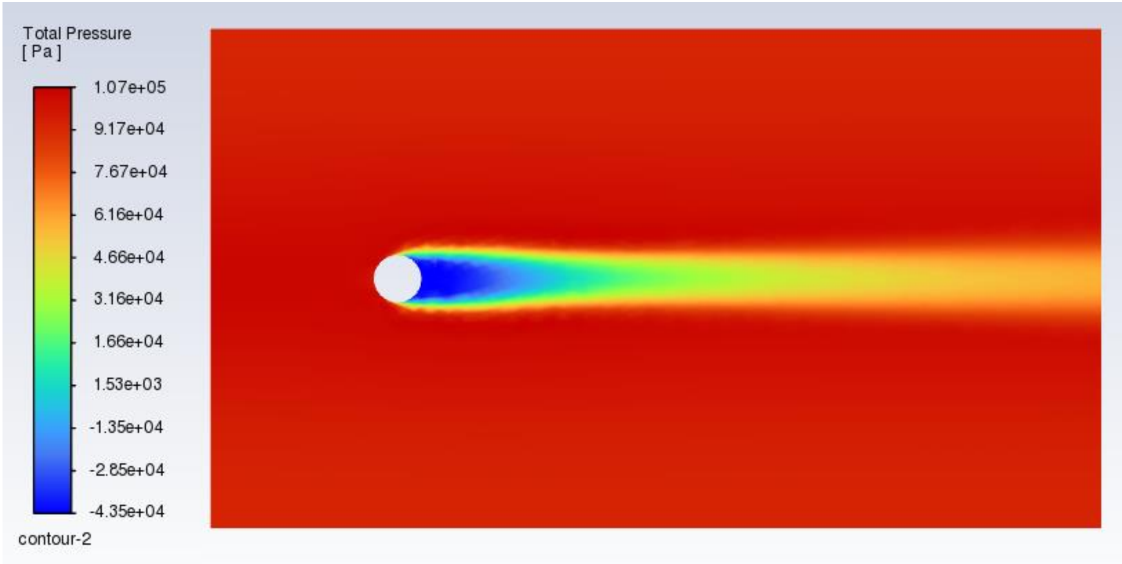
Scaled Residuals:



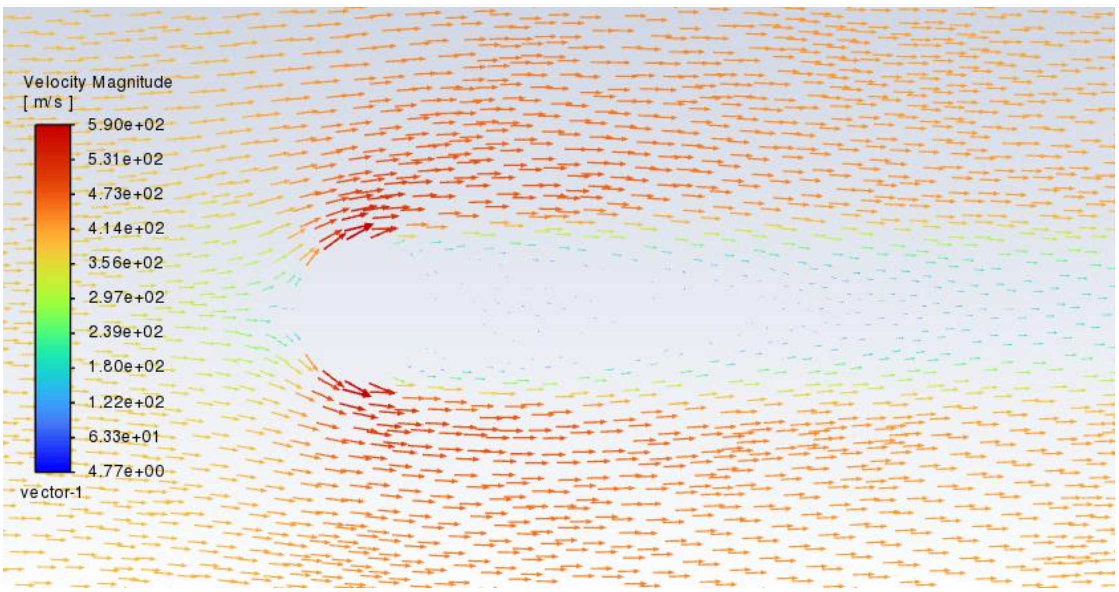
X-Velocity Contour plot:



Total Pressure Contour Plot:



Velocity Vector Plot:



Forces:

Forces - Direction Vector	(1 0 0)		
	Forces [N]		
Zone	Pressure	Viscous	Total
pisa	8191.0801	28.119444	8219.1995
Net	8191.0801	28.119444	8219.1995

Forces - Direction Vector	(0 1 0)		
	Forces [N]		
Zone	Pressure	Viscous	Total
pisa	282.62384	0.26192307	282.88576
Net	282.62384	0.26192307	282.88576

5. 3D Simulation

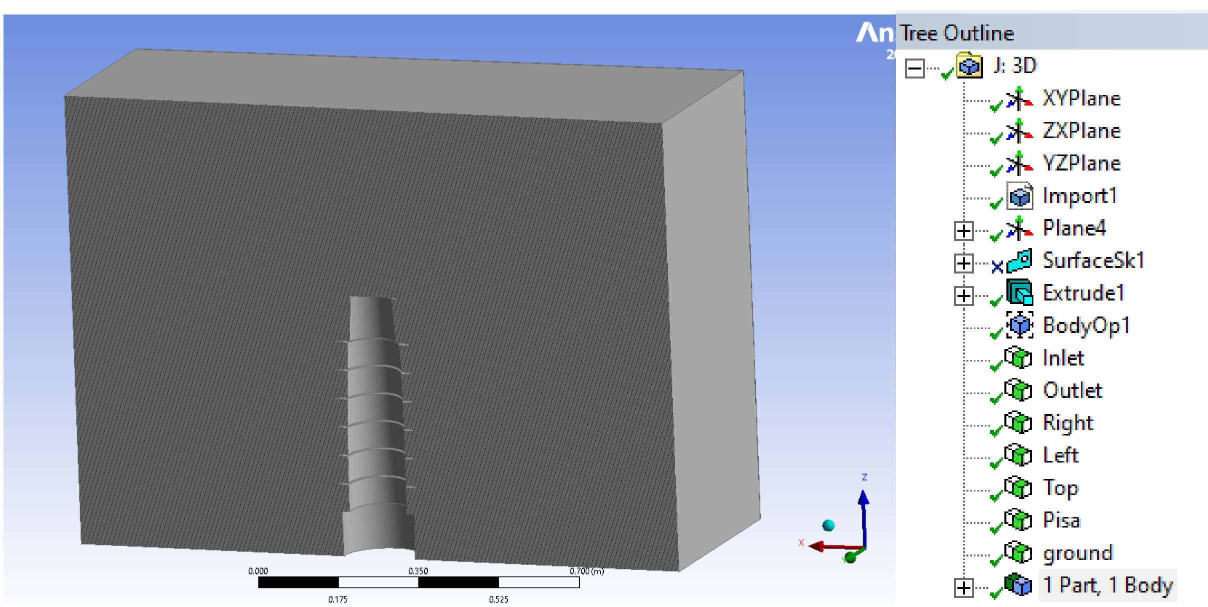
In the 3D simulation, initially the third velocity of u=193.5 m/s was used. The set up had the 3D Pisa Tower file imported into fluent and an extruded sketch was created. The operational body tool was used to generate the domain for the flow. The tower was placed at the bottom of the domain, so that the flow is to be measured around and over the building. All aspects of the domain were labeled in order to avoid confusing and allow for easily boundary condition selections.

The mesh for the 3D analysis was initially set as an automatic mesh with triangles. The flow was set to turbulent with k-omega, and the flow was set as coupled as it is at a high velocity. After running the first simulation, the results did not converge.

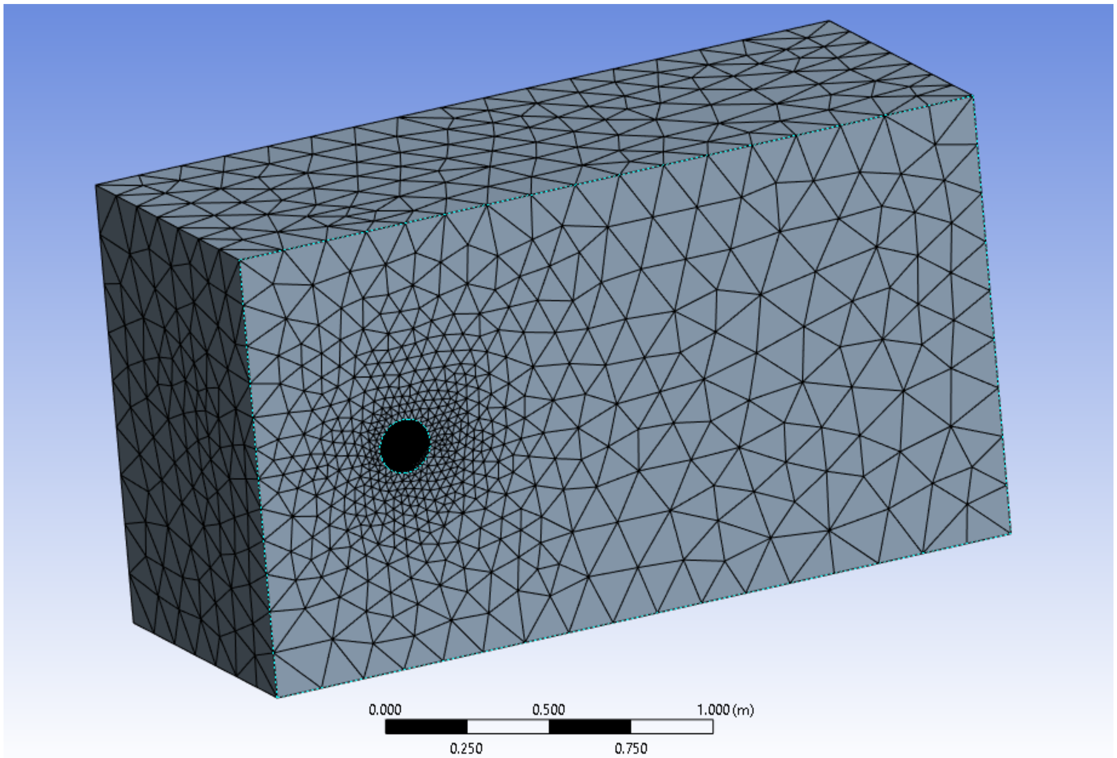
For the second simulation, the highest velocity, u=387m/s was used to see if the velocity change would make impact to the convergence. This was then realized as a bad ideal as the higher velocity numerical method may require iterations to reach convergence, so this simulation also failed to converge.

For the final simulation, the mesh was refined to the same mesh size as the 2D simulations, which was 0.025m while still kept at the velocity of 387m/s. The refinement of the mesh may provide higher accuracy and precision for capturing the important flow features and fine details. The results to the final simulation also did not converge. This is likely due to the inadequate grid resolution, as the physical model is a highly complex structure with 47 faces, causing the mesh cells to have high skewness around the boundary of the building.

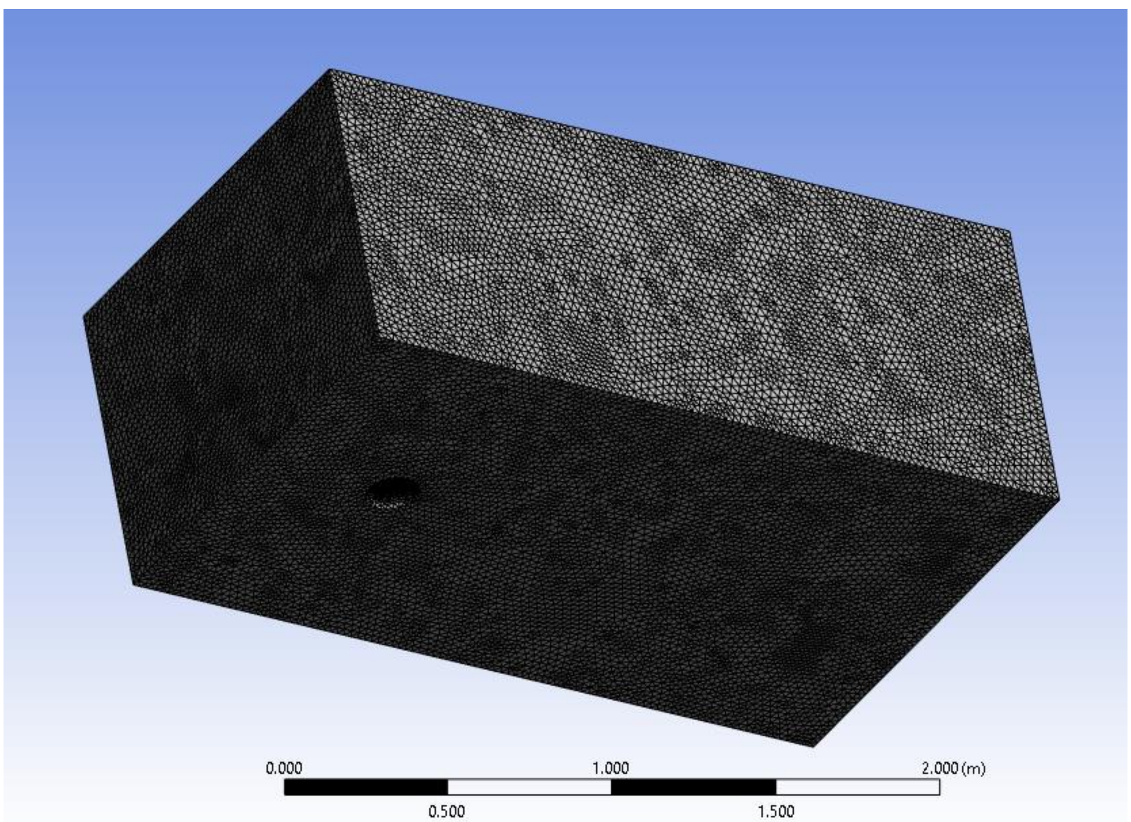
5.1.Setup



Default mesh size, automatic method:



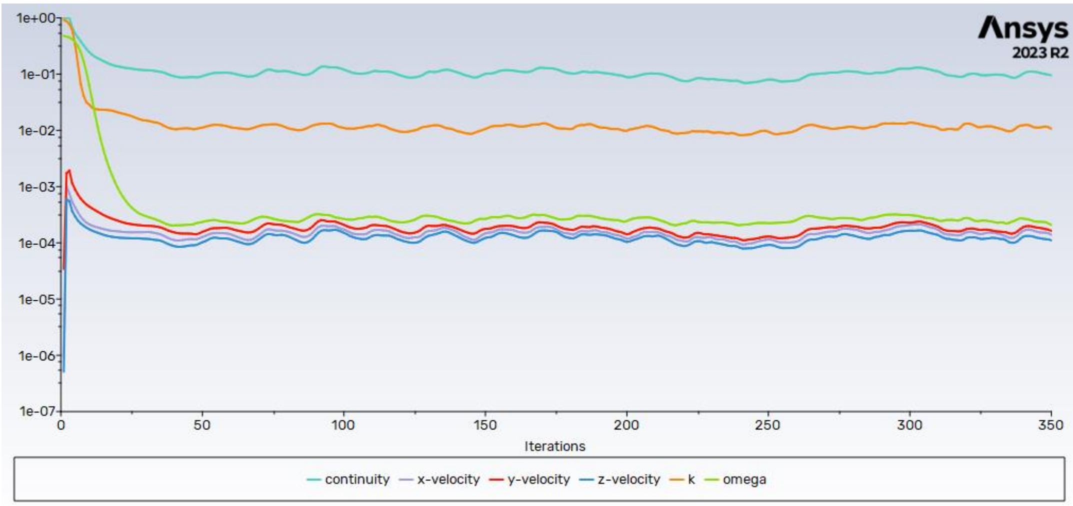
Mesh size = 0.025m:



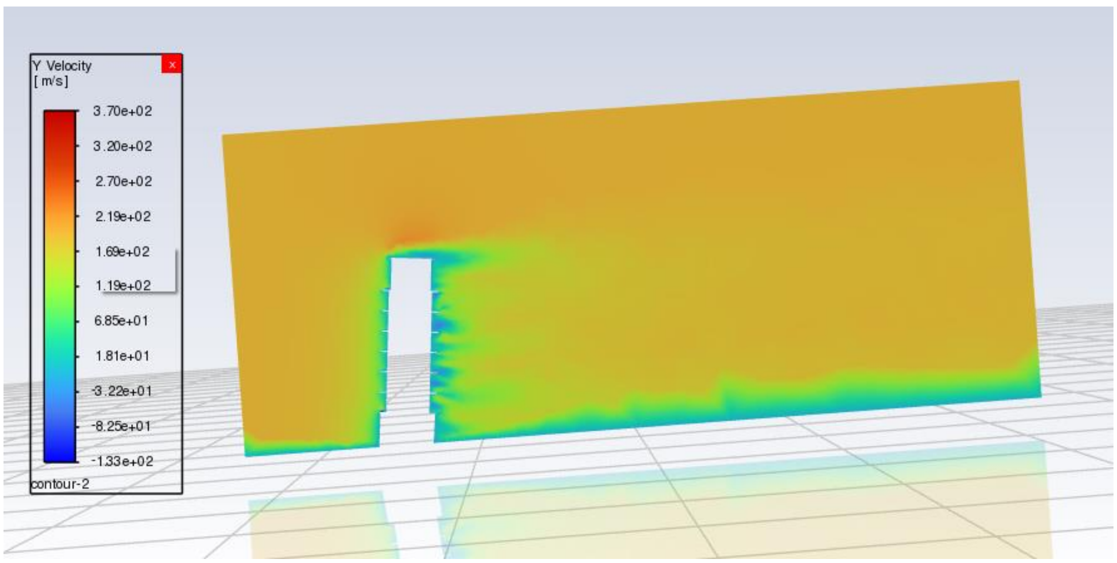
5.2.Simulation results

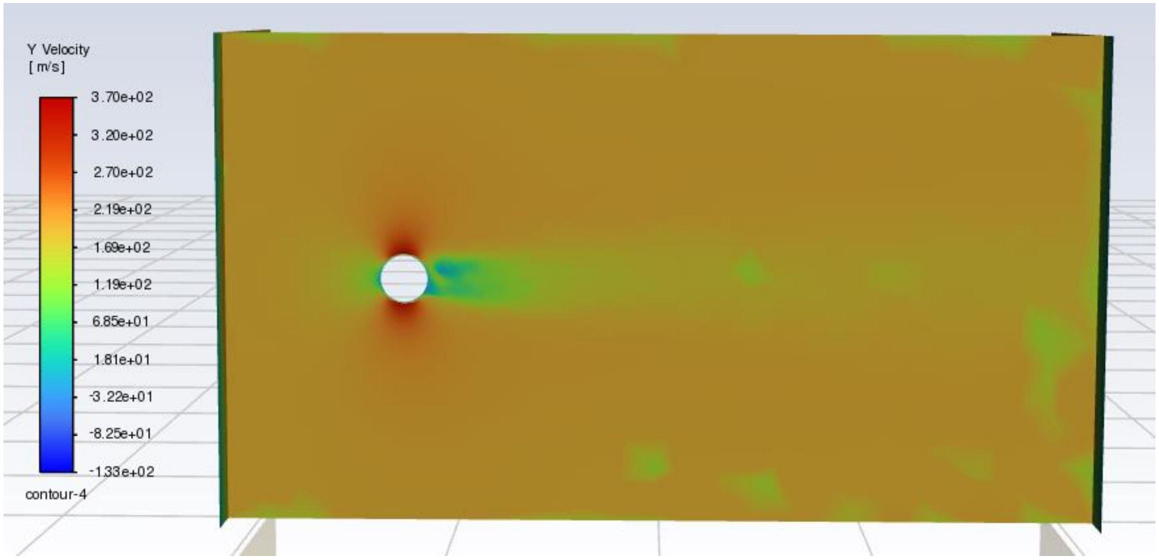
Run 1: With a velocity = 193.5m/s

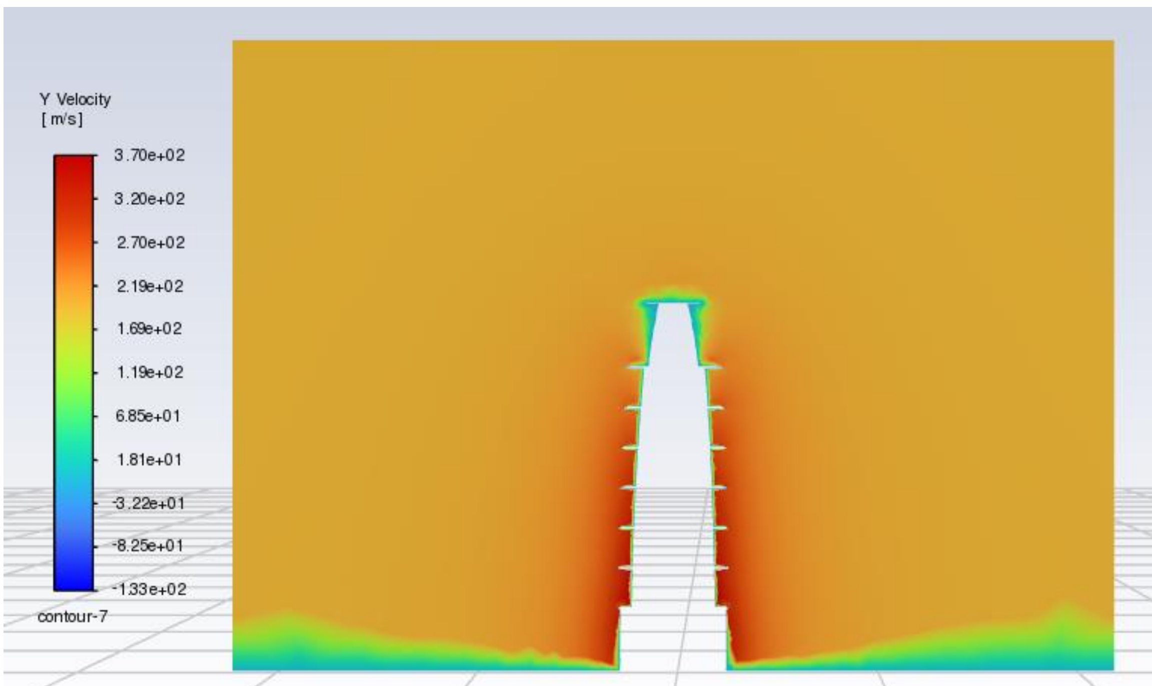
Scaled Residuals:



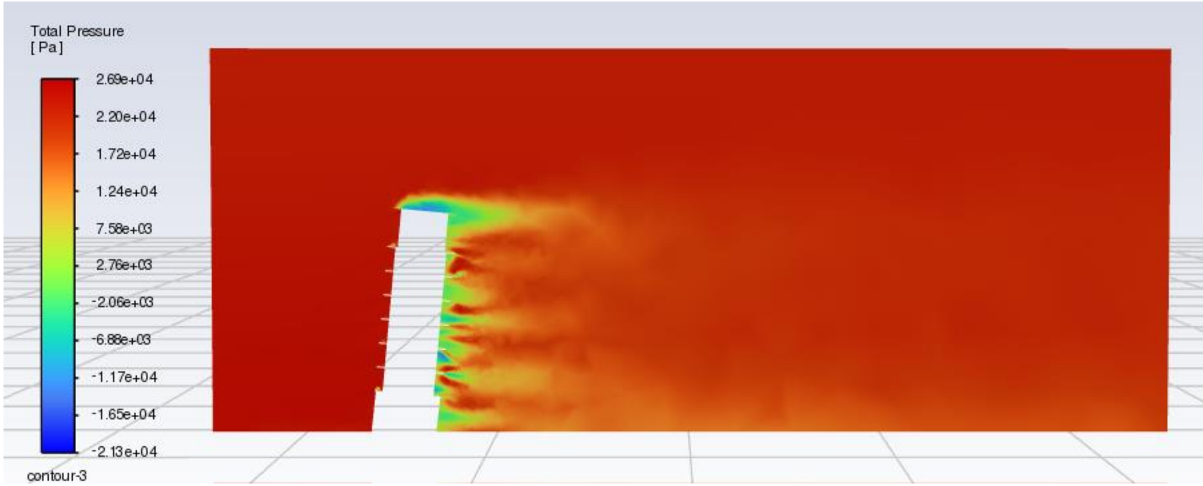
Velocity Contour Plots at 3 planes:

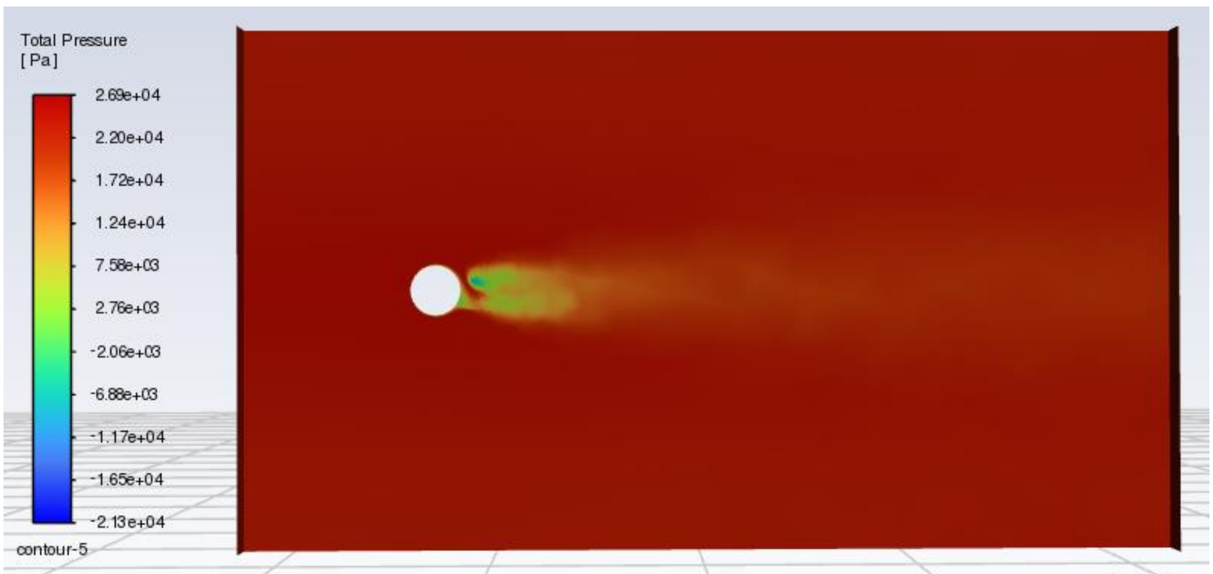


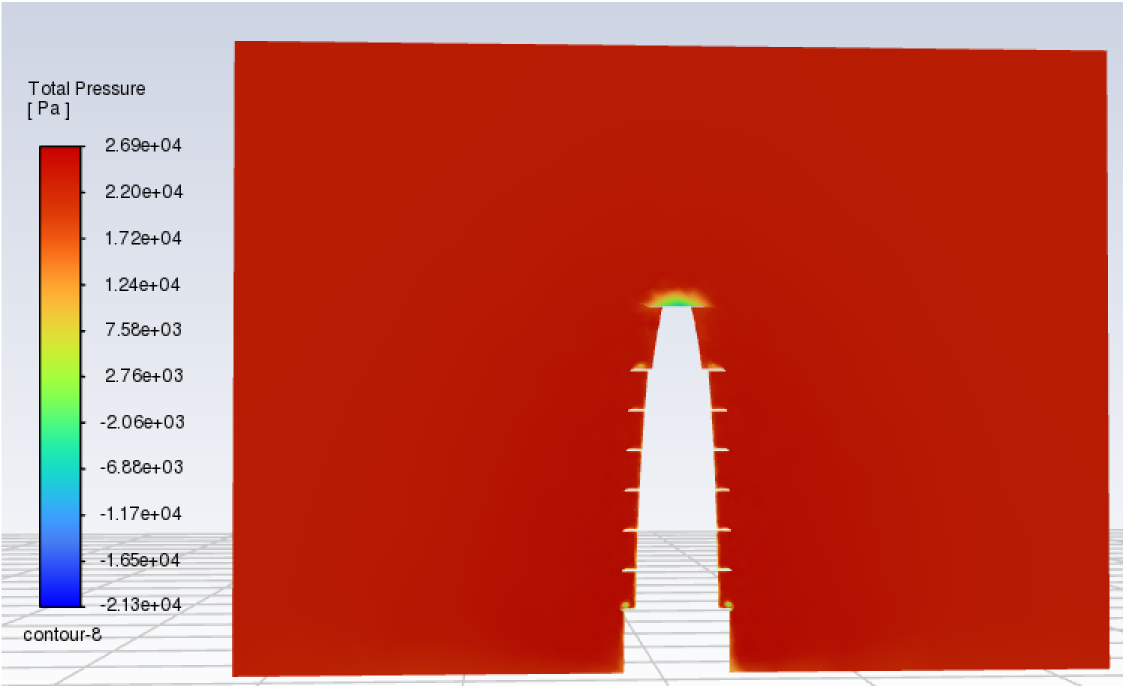

Y Velocity
[m/s]
3.70e+02
3.20e+02
2.70e+02
2.19e+02
1.69e+02
1.19e+02
6.85e+01
1.81e+01
-3.22e+01
-8.25e+01
-1.33e+02
contour-4


Y Velocity
[m/s]
3.70e+02
3.20e+02
2.70e+02
2.19e+02
1.69e+02
1.19e+02
6.85e+01
1.81e+01
-3.22e+01
-8.25e+01
-1.33e+02
contour-7

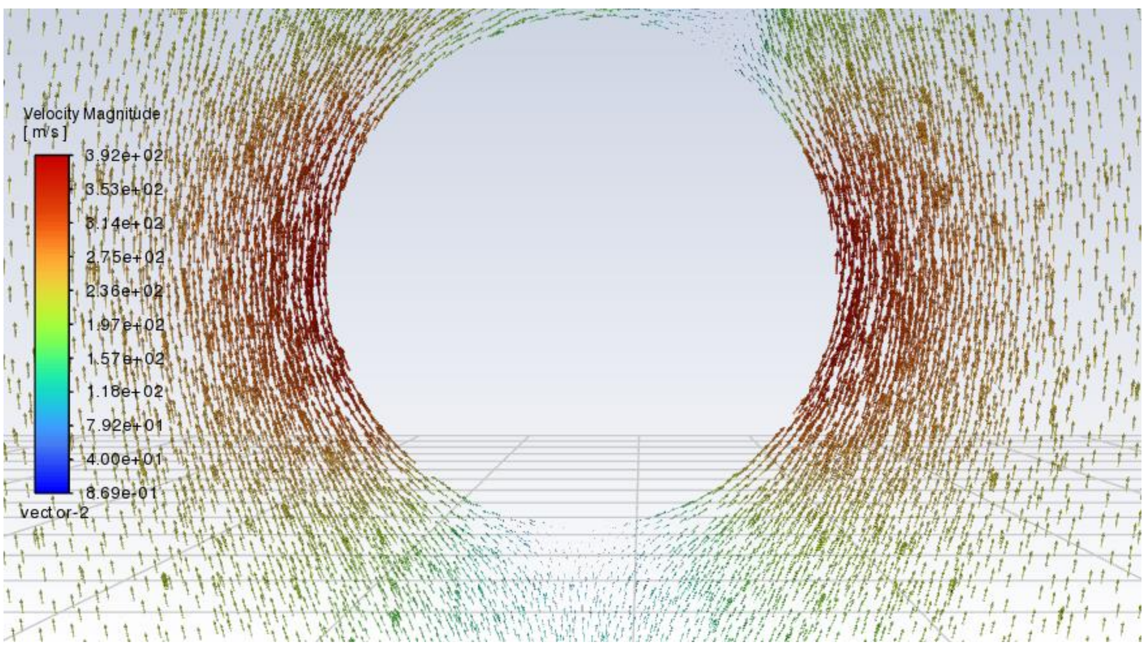
Pressure Contour Plots at 3 planes:







Velocity Vector Plot:



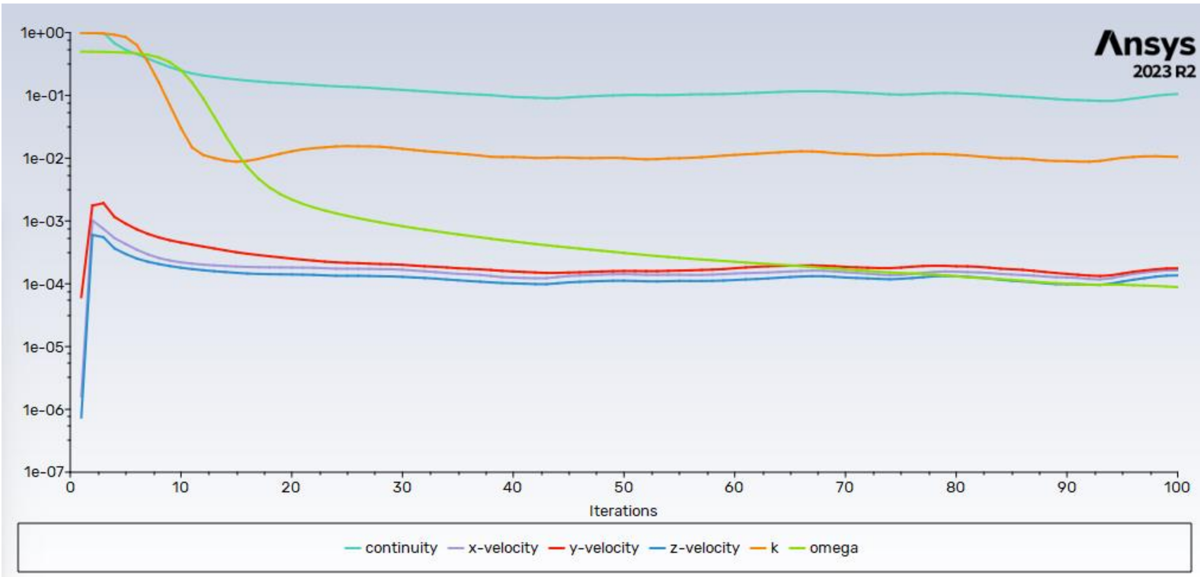
Forces:

```
Forces - Direction Vector (1 0 0)
                           Forces [N]
Zone                       Pressure         Viscous          Total
pisa                       -35.247009       0.060887866      -35.186121
-------------------------- ---------------- ---------------- ----------------
Net                        -35.247009       0.060887866      -35.186121
```

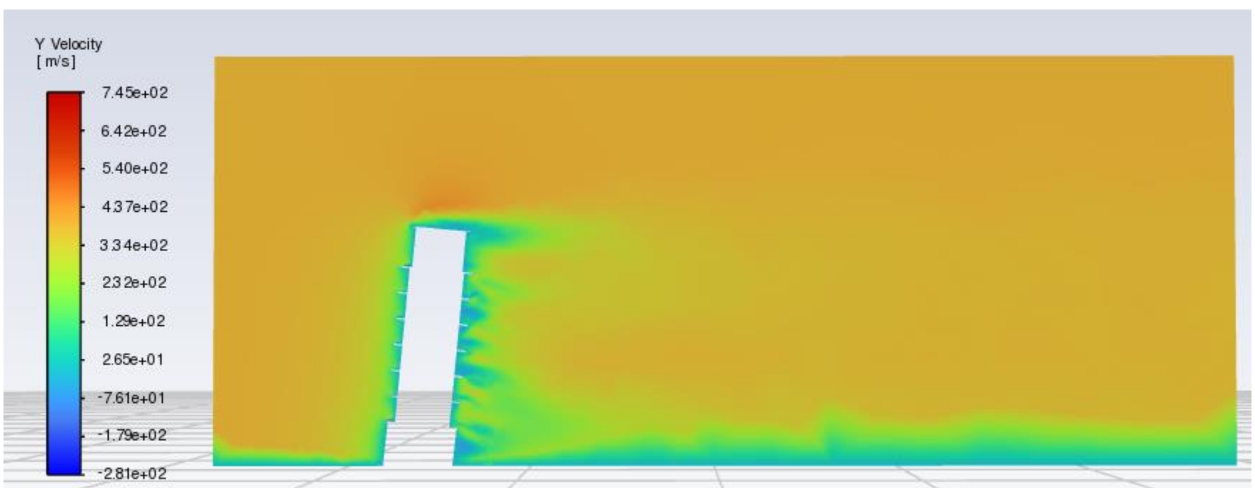
```
Forces - Direction Vector (0 1 0)
                           Forces [N]
Zone                       Pressure         Viscous          Total
pisa                       747.26074        29.325686        776.58643
-------------------------- ---------------- ---------------- ----------------
Net                        747.26074        29.325686        776.58643
```

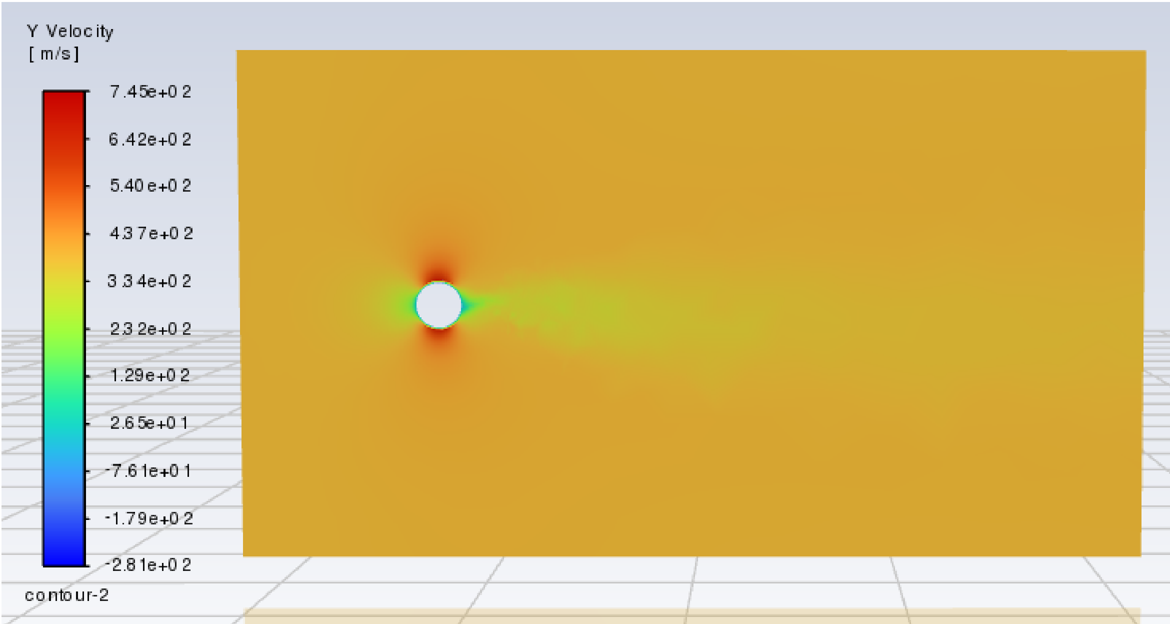
Run 2: with a velocity = 387m/s

Scaled Residuals:

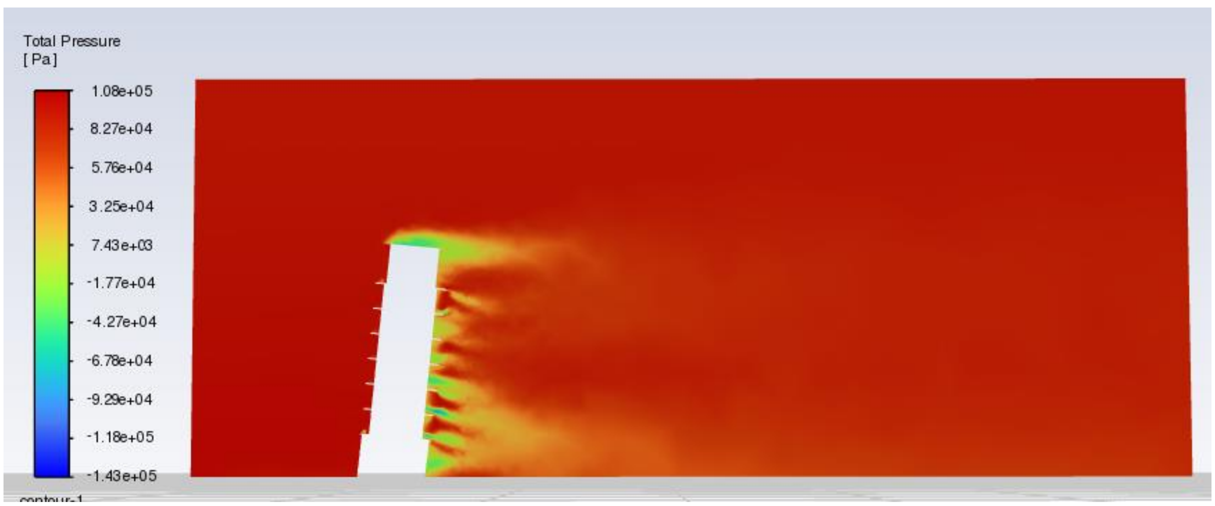


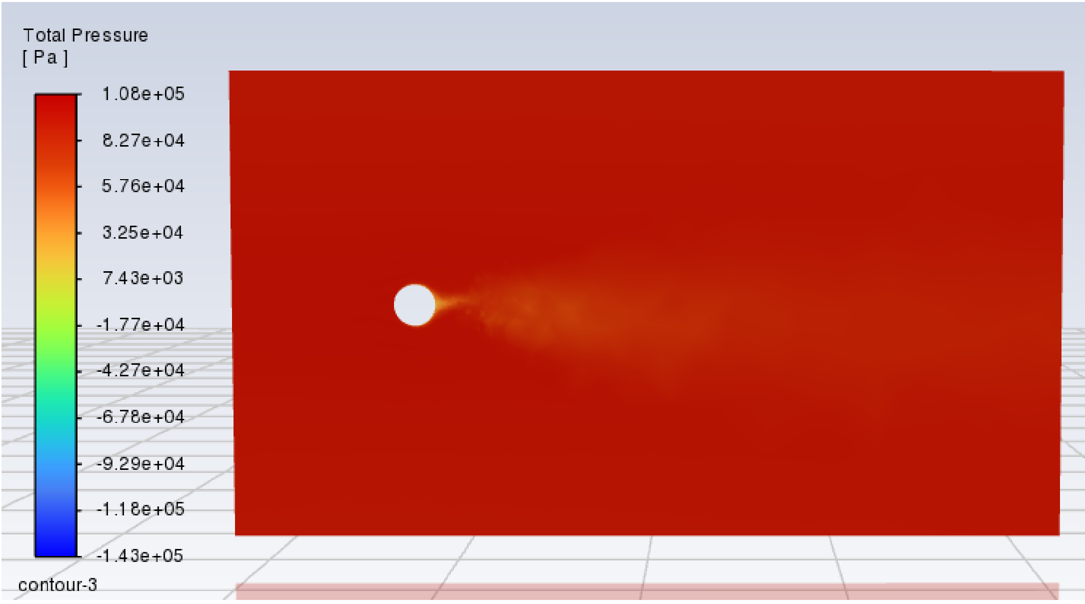
Velocity Contour Plots:





Pressure Contour Plots:





Forces:

Forces - Direction Vector (1 0 0)

	Forces [N]		
Zone	Pressure	Viscous	Total
pisa	45.136402	-0.089571938	45.04683
Net	45.136402	-0.089571938	45.04683

Forces - Direction Vector (0 1 0)

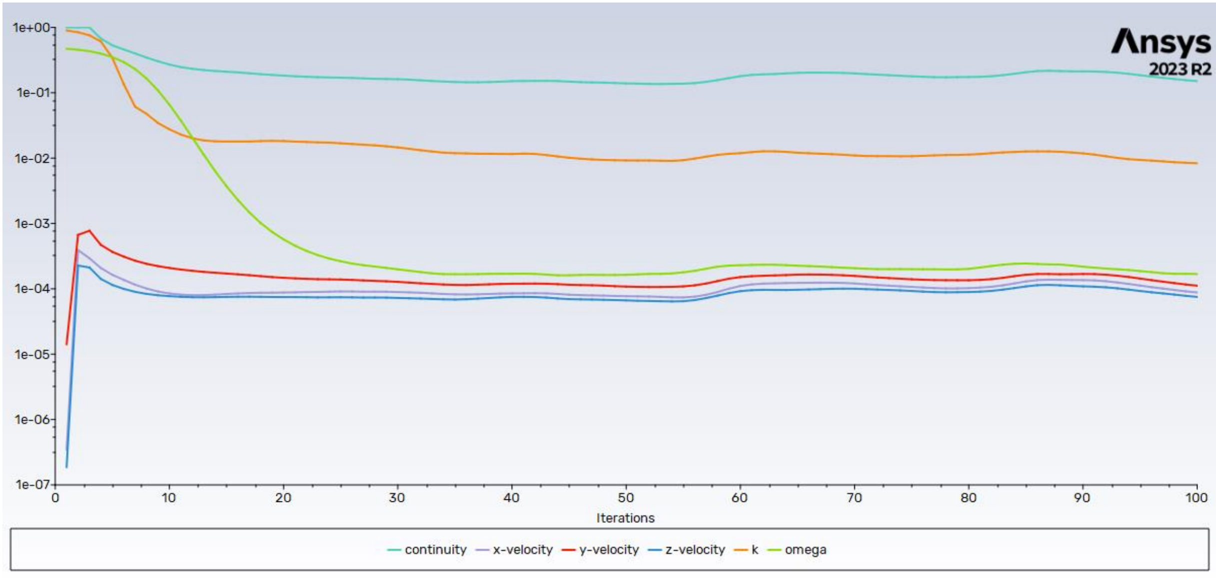
	Forces [N]		
Zone	Pressure	Viscous	Total
pisa	3620.9043	99.673973	3720.5783
Net	3620.9043	99.673973	3720.5783

Forces - Direction Vector (0 0 1)

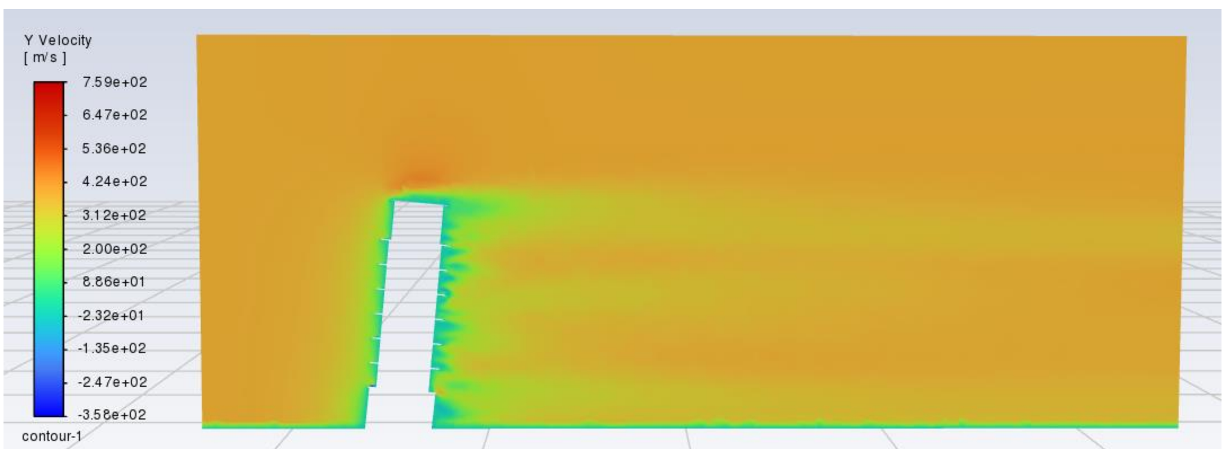
	Forces [N]		
Zone	Pressure	Viscous	Total
pisa	1715.1866	-9.0814009	1706.1052
Net	1715.1866	-9.0814009	1706.1052

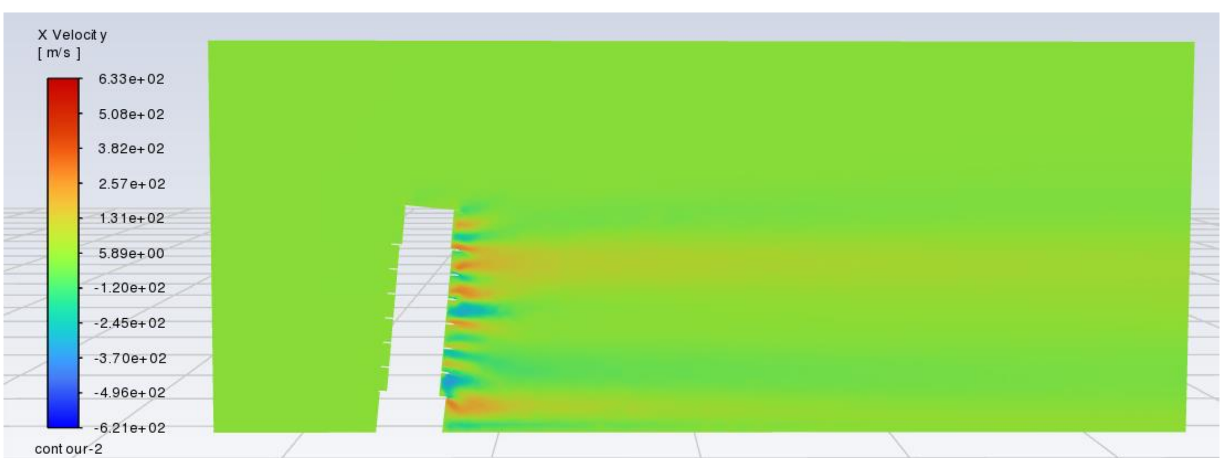
Run 3, with a velocity = 387m/s, and a mesh size of 0.025m:

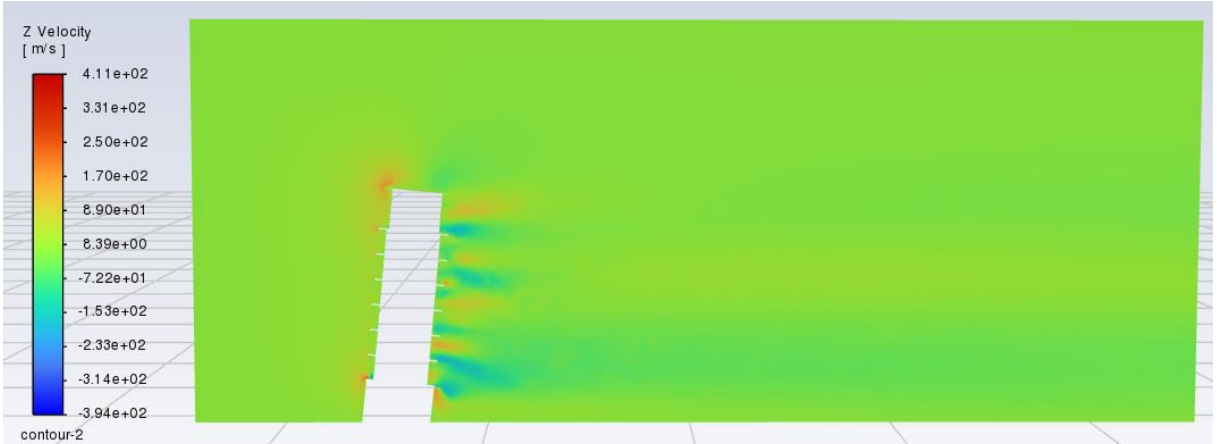
Scaled Residuals:



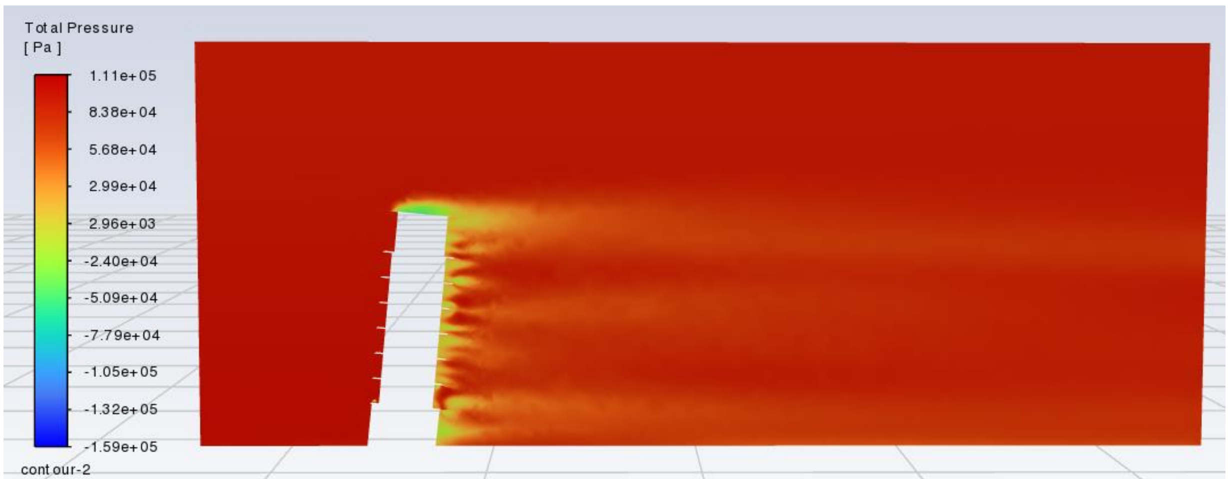
Velocity Contour Plots:







Total Pressure Contour Plot:



Forces:

```
Forces - Direction Vector (1 0 0)
                          Forces [N]
Zone                      Pressure        Viscous         Total
pisa                      -614.32147      -0.2738145      -614.59529
------------------------- --------------- --------------- ---------------
Net                       -614.32147      -0.2738145      -614.59529
```

```
Forces - Direction Vector (0 1 0)
                          Forces [N]
Zone                      Pressure        Viscous         Total
pisa                      2778.5864       111.08516       2889.6716
------------------------- --------------- --------------- ---------------
Net                       2778.5864       111.08516       2889.6716
```

```
Forces - Direction Vector (0 0 1)
                          Forces [N]
Zone                      Pressure        Viscous         Total
pisa                      1697.6615       -8.1135225      1689.548
------------------------- --------------- --------------- ---------------
Net                       1697.6615       -8.1135225      1689.548
```

6. **Results:**

6.1.Tables of Results:

Tests	**Velocities**	**X-Component Forces**			**Y-Component Forces**			**Resultant Force**
		Pressure	**Viscosity**	**Total Force**	**Pressure**	**Viscosity**	**Total Force**	
1	1.2	1.9238514	0.029573718	1.9534251	0.11489546	0.000174608	0.11507006	1.956811371
2	2	516.4201	2.8427954	519.2629	21.327484	0.020250246	21.347734	519.7015346
3	4	2055.5447	8.9231329	2064.4678	75.236923	0.072064191	75.308987	2065.840928
4	8	8189.876	28.408573	8218.2845	279.02054	0.26953065	279.29007	8223.028826

Force Coefficients and Reynolds Numbers:

Using the following equation, the force coefficients were calculated.

$$C = \frac{2F}{\rho u^2 d}$$

Where:

F = the resultant force for each test

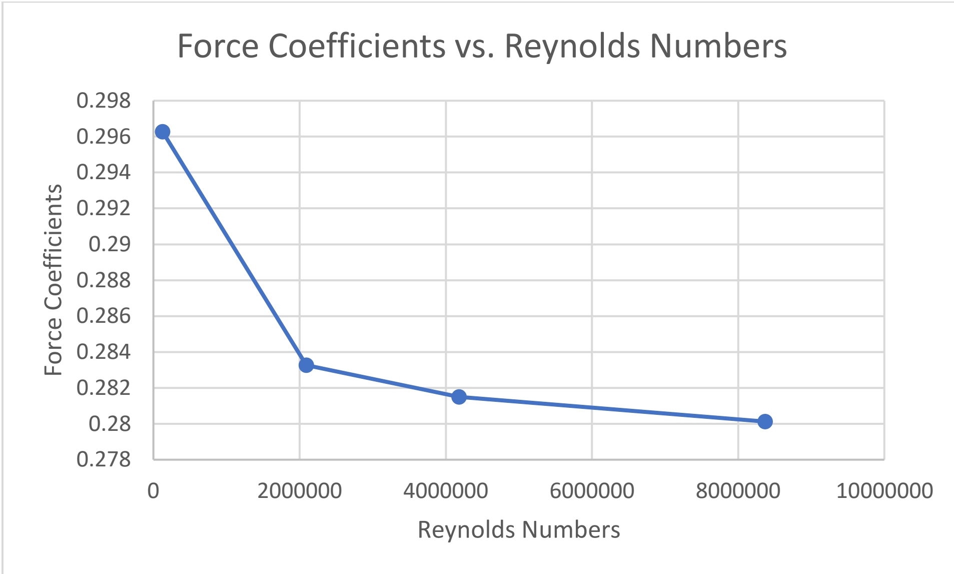
ρ = the density of air (1.225 kg/m^3)

u = the velocity for specific simulation

d = the diameter of the model (0.32m)

Tests	**Velocities**	**Force**	**Force Coefficients**	**Reynolds Number**
1	5.805	1.9568114	0.296270523	125513.51
2	96.75	519.70153	0.283266998	2091891.89
3	193.5	2065.8409	0.281500303	4183783.78
4	387	8223.0288	0.280126252	8367567.57

6.2.Force Coefficient vs. Reynolds Number Plot:



The plot of force coefficients vs. Reynolds numbers accurately shows the relationship between the behavior of the force coefficients vs. the Reynolds number as the velocity increases overtime. This graph clearly shows the laminar, transitional and turbulent regions of the flow overtime as the velocities were increased.

7. Analysis:

7.1.Discussion

From the 2D simulation results, it is shown that the force exerted by the wind on the building increases with wind velocity. The x-component of the forces are higher, indicating that the wind primary exerts its force in the direction of the flow. The force coefficients decrease as the wind increases velocity, meaning that the force experiences a less proportionally increasing force. This is likely due to the turbulence effects, at these increasingly high turbulent flows. The Reynolds number increases with the wind velocity, transitioning the flow through the laminar to turbulent phases. The transition shows the increased complexity of the flow and the complex interactions as the flow becomes more turbulent, generally making the simulations converge slower as the velocity of the wind is increased.

Reviewing the 2D and 3D simulations, the results were outputted very differently. The 2D simulation provides a simplified approximation of the wind forces acting on the building, whereas the 3D simulation offers a more detailed and accurate representation, capturing complex aerodynamic effects that are an accurate assessment of wind loads on the structure. The results from the 3D simulation would usually be more reliable and realistic, but in this case the 3D simulation did not converge, meaning there is still error that needs to be resolved. The 2D results all converged and clearly show the forces around the 2D building slice, but this is not enough to fully represent how the flow is behaving. The 3Dmodel of the Pisa Tower is unique as it is on an angle of 5 degrees with

47 faces and an extremely complex geometry. This makes the 3D model very difficult to mesh, without high skewness and a poor orthogonality, which slows the convergence rate. To perform these 3D simulations again, I would simplify the geometry to avoid these errors due to complexity.

To perform this experiment in-person, a wind tunnel could be used to replicate the simulations in a physical experiment. The wind tunnel would replicate the domain, and the computational simulation could be replicated by testing an appropriately scaled model of the building in a controlled wind flow with all the same velocities. The pitot tubes in the domain and attached to the model would then be used for measuring forces, pressures, and flow patterns to understand the aerodynamic forces acting on the building in all directions. The experimental setup would provide real-world validation and a further study of wind behavior around the building.

7.2.Conclusion

This project successfully studied the forces caused by wind velocity on the Pisa Tower through 2D and 3D simulations. The 2D simulations provided useful analysis on the behavior of the flow horizontally at a 2D slice of the building, further showing how the velocity and forces behave with the Reynolds numbers and the force coefficients. Analysis was performed into different mesh quality and sizes and knowing how they affect the flow as well as boundary condition selection and turbulence changes. The 3D simulations provided an interesting look at the flow for the entire building, even though the results for the 3D model did not converge, the complex domain allowed for insightful analysis of why convergence may not occur, and how a mesh may be altered to achieve grid independence.

Overall, this project demonstrated the importance of using both numerical simulations and experimental methods to analyze wind forces on buildings, and it has provided insights into the challenges and opportunities in modeling and testing these forces for real world engineering applications.