Applied CFD Project 4

Brent Skabelund

MAE 598

November 27, 2018

Name of Collaborator: Robert Chimel	
Task(s)	**Contribution to Collaborative Effort**
Task 1	Worked together to determine the correct time step for the problem.
Task 2	Compared and collaborated on the drag and lift forces as a function of the flying saucer angle.
Task 3	Discussed the different reasons why task 3b had a higher drag force than task 3a.

Task 1:

A) To estimate the Reynolds number, we take the density and the dynamic velocity of water found in ANSYS and use the equation of the Reynolds shown below. It is important to note that v_x is the x-velocity and the L_c is the characteristic length which in this problem is 10 cm. The calculations are done below.

$$Re = \frac{\rho v_x L_c}{\mu}$$

$$Re = \frac{(998.2\frac{kg}{m^3})\left(0.003\frac{m}{s}\right)(0.1\,m)}{0.001003\frac{kg}{m*s}}$$

$$Re = 298.564$$

The problem was modeled as a 2-D flow with the inlet velocity to be .3 cm/s and the side was modeled as an outflow. The hole in the plate was modeled as a wall-cylinder and the sides were walls as well. The flow was modeled as a water laminar flow and ran to 1 hour. The time step was 0.5 seconds and 8 iterations per time step. The contour plots of the x-velocity and static pressure are shown in figure 1 and 2 below.

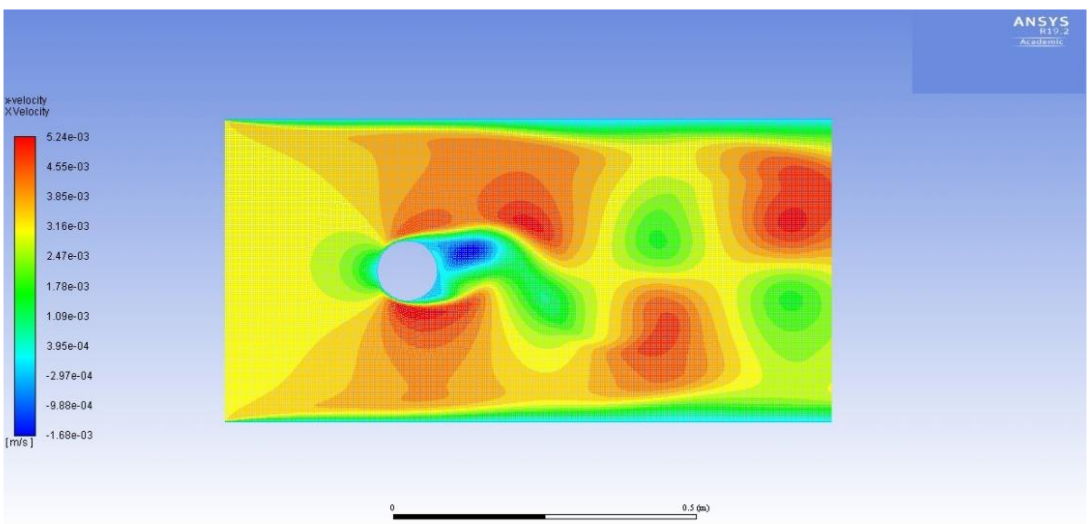


Figure 1- X-Velocity at 0.3 cm/s

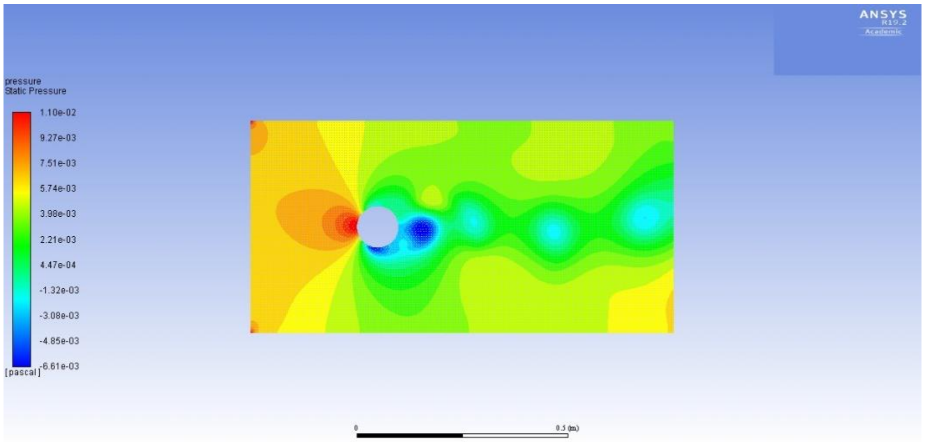


Figure 2 -Static Pressure at 0.3 cm/s

B) The same precedure as in task 1a was done in task 1b. The difference is that the velocity was increased from .3 cm/s to 2 cm/s. The change in Reynolds number is shown below.

$$Re = \frac{\rho v_x L_c}{\mu}$$

$$Re = \frac{(998.2\frac{kg}{m^3})\left(0.02\frac{m}{s}\right)(0.1\ m)}{0.001003\frac{kg}{m*s}}$$

$$Re = 1990.429$$

Like in task 1a, the hole in the plate in task 1b was modeled as a wall-cylinder and the sides as walls. The flow was modeled as a water laminar flow and ran to 1 hour. The time step was 0.5 seconds and 8 iterations per time step. The contour plots of the x-velocity and static pressure are shown in figure 3 and 4 below.

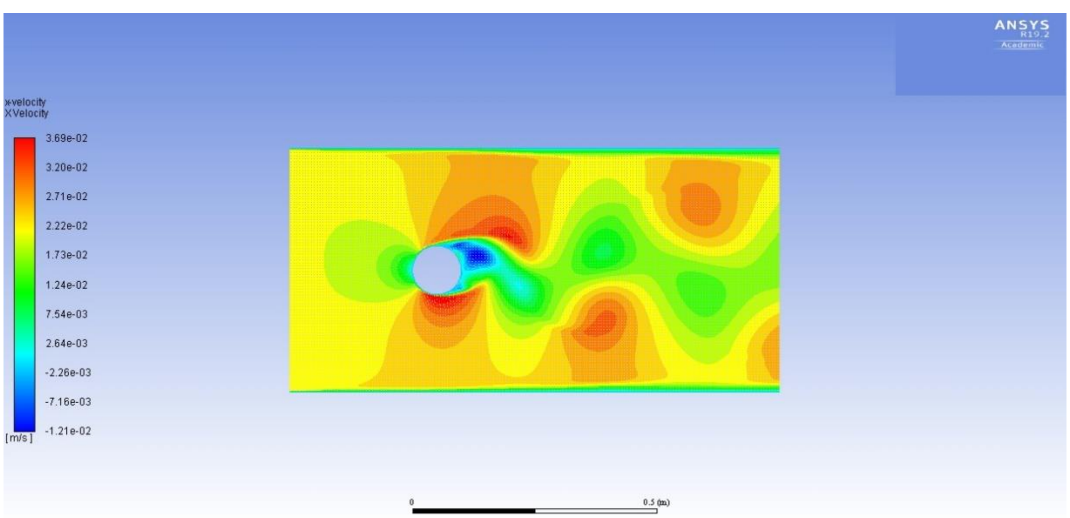


Figure 3 - Contour plot of X-Velocity at 2 cm/s

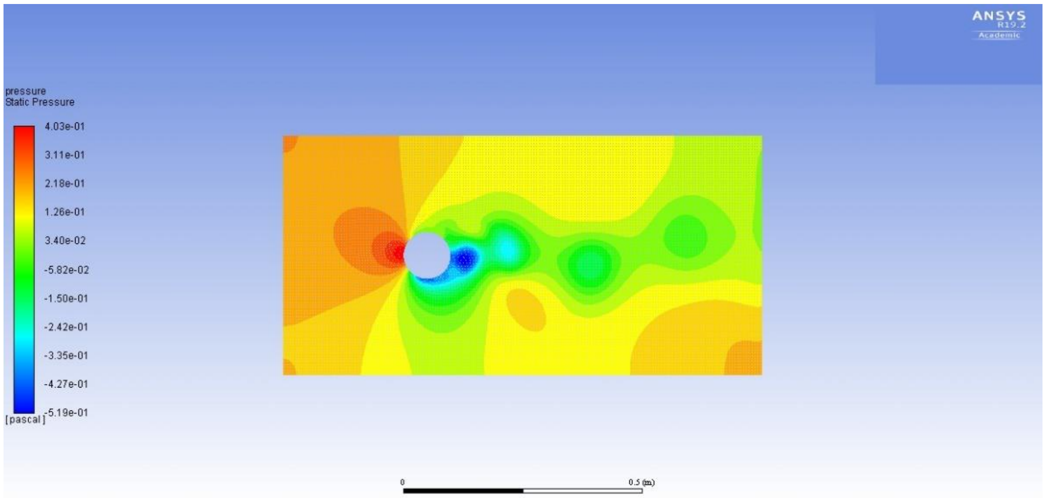


Figure 4 - Contour Plot of Static Pressure at 2 cm/s

The next step in this task was to plot the Lift Coefficient as a function of flow time. In the task, it was asked to plot between 50 mins and 1 hr. The resulting plot is shown below in figure 5. The period of the oscillation of the flow and the amplitude of the wave is also described below the plot.

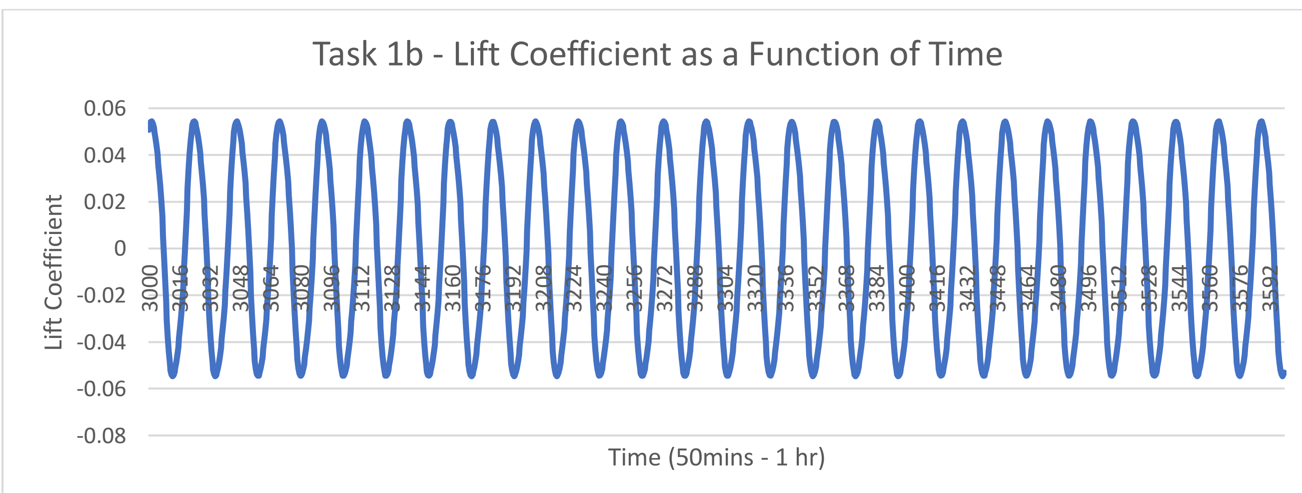


Figure 5 - Plot of Lift Coefficient as a Function of Time (Task 1b)

Period = 22 s

Amplitude = 0.0545

C)

For task 1c, it asked to use the same conditions and set up as task 1b. However, the circle was modeled as an eclipse and asked to run two different scenarios. One run has the major axis of the ellipse along the y-axis and the other run have the major axis of the ellipse along the x-axis. The run with an ellipse elongated along the y axis was ran at a time step of .5 seconds and 20 iterations per time step to one hour. The run with the ellipse elongated along the x axis was ran at a time step of 1 second and 20 iterations per time step to one hour. The results between the two runs are shown below in figure 6 and 7.

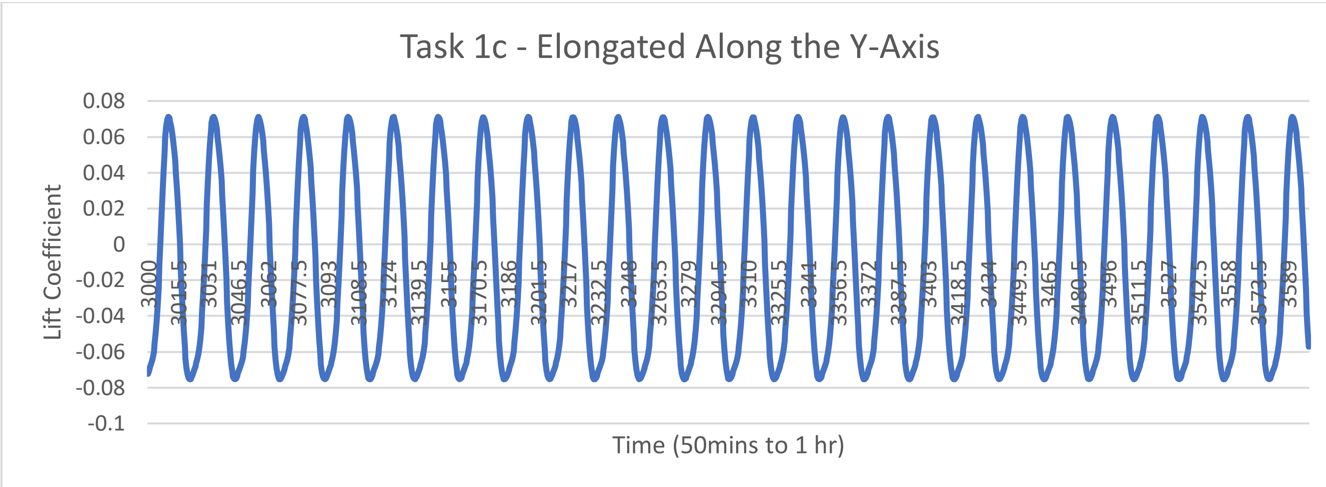


Figure 6 - Lift Coefficient with the Ellipse Elongated Along the Y-Axis.

Period = 23.5 s

Amplitude = 0.0711

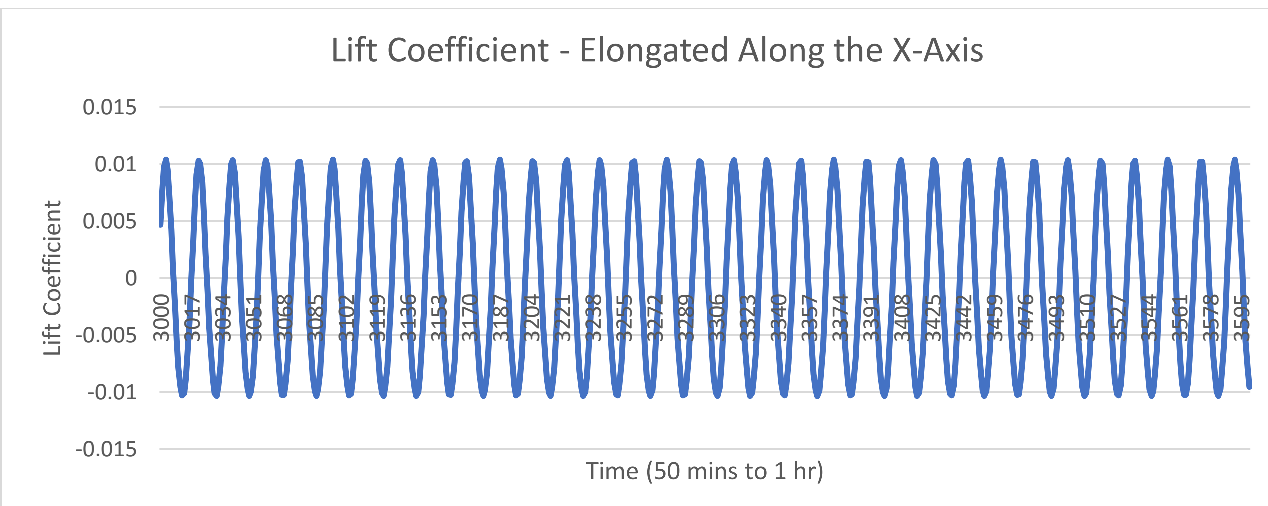


Figure 7 - Lift Coefficient with the Ellipse Elongated Along the X-Axis

Period = 19 s

Amplitude = 0.010357

	Task 1b (Cylinder)	Task 1c (Elongated in the Y-Axis)	Task 1c (Elongated in the X-Axis)
Period	22 s	23.5 s	19 s
Amplitude	0.0545	0.0711	0.010357

In comparing between the three runs, we can see that the ellipse elongated in the Y-Axis had a bigger amplitude and period. The ellipse elongated along the x axis had the smallest period and amplitude. The cylinder was in between the two different ellipse runs. This makes sense that the ellipse elongated in the y direction will have the biggest period and amplitude. It has the most surface area hitting the flow.

Task 2:

In this task, the goal was to model a flying saucer in a virtual wind tunnel and determine the lift force and drag force at angles 0 degrees, 15 degrees, 30 degrees and 45 degrees. This was done by modeling the flying saucer and insert a cylinder surrounding the flying saucer. Next, the bouillon command was used to subtract the fly saucer from the cylinder, effectively making a cut out of the flying saucer in the cylinder. Air was then flowed through the cylinder and the lift and drag forces along the wall of the saucer cut out were monitored. The desired deliverables are discussed below. The inlet velocity of the air flow was 50 m/s and the exit was set to be an outflow. The flow was modeled as a turbulent flow and searching for a steady solution.

(i) The first deliverable for task 2 was to plot the mesh of the plane symmetry with the flying saucer at 45 degrees. The mesh is shown below in Figure 8.

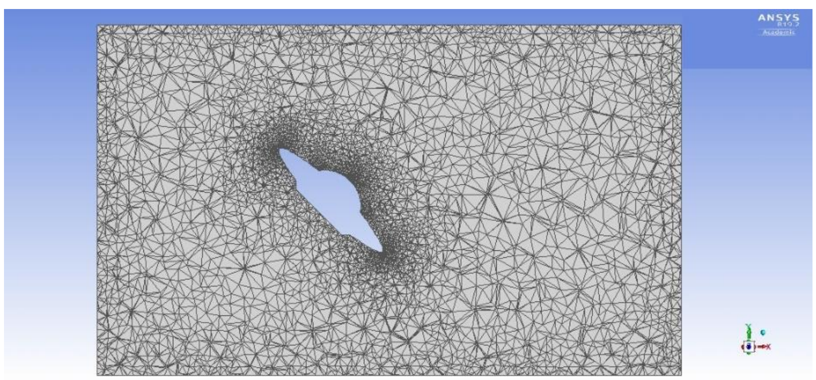


Figure 8 - Mesh of the Plane of Symmetry with the Flying Saucer at an Angle of 45 Degrees

(ii) The next deliverable was to show the x-velocity at 0 degrees and 45 degrees. These contour plots are shown in figure 9 and 10. These figures were tilted by a little bit forward in order to see the different bands of the contour plot. In ANSYS, there is a shine that drowns out the smaller, more subtle bands that are important to note.

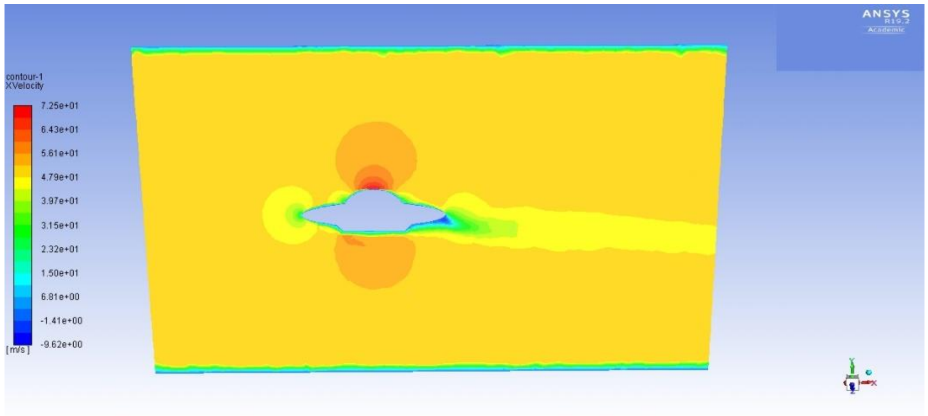


Figure 9 - Contour Plot of the X-Velocity at 0 degrees

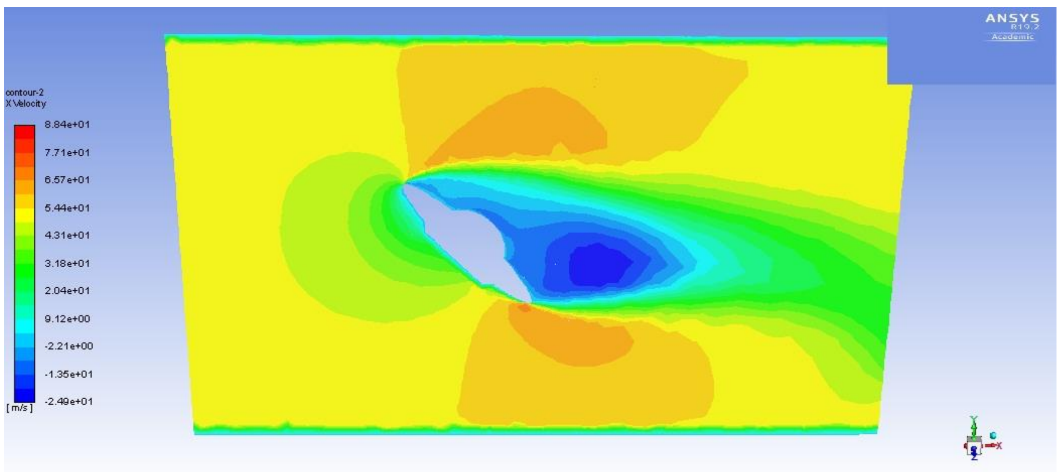


Figure 10 - Contour Plot of the X-Velocity at 45 degrees

(iii)

The final deliverable required was to plot the lift force and drag force as a function of the saucer angle. The calculations of the flying saucer at 0 degrees converged to a lift force of 16.556 Newtons and 9.882 Newtons as a drag force.

The flying saucer at 15 degrees ran to have a lift force of 117.992 Newtons and a drag force of 27.778 Newtons. This value did not coverage after 5000 iterations but stayed relatively linear as iterations increased.

The flying saucer at 30 degrees ran up to 5000 iterations. The drag force and lift force oscillated by a little bit, so the average was taken when the values started to plateau. The graph of the drag force and lift force as iterations increased are shown in figure 11 and 12.

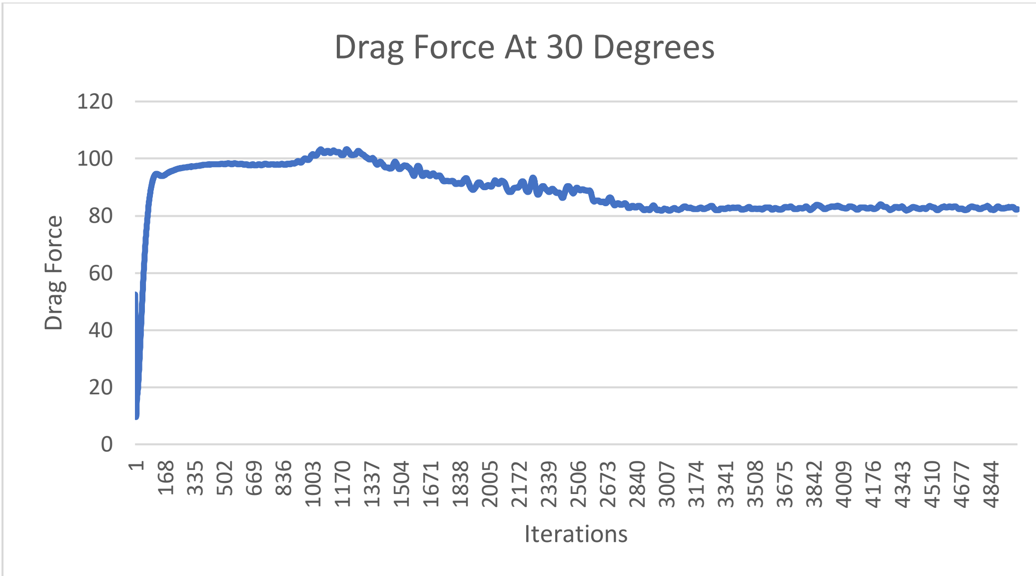


Figure 11 - Drag Force at 30 Degrees

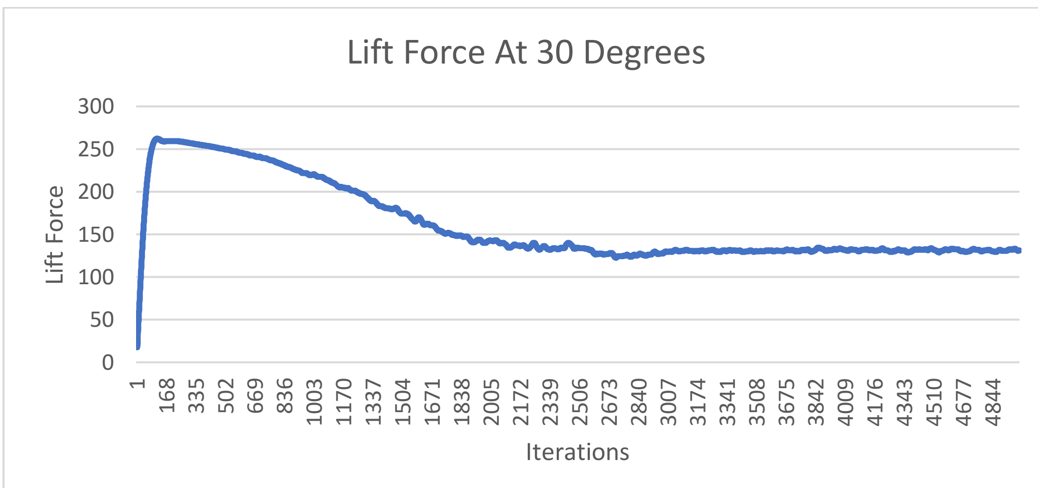


Figure 12 - Lift Force At 30 Degrees

The flying saucer at 45 degrees ran up to 5000 iterations as well. The drag force and lift force oscillated, so the average was taken when the values started to oscillate at a consistent wave. The graph of the drag force and lift force as iterations increased are shown in figure 13 and 14.

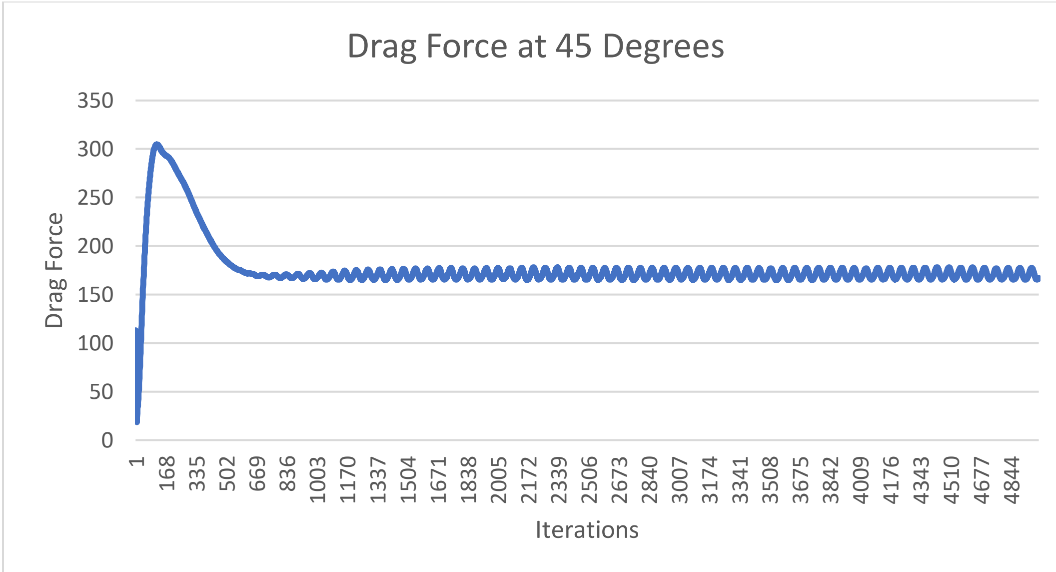


Figure 13 - Drag Force at 45 Degrees

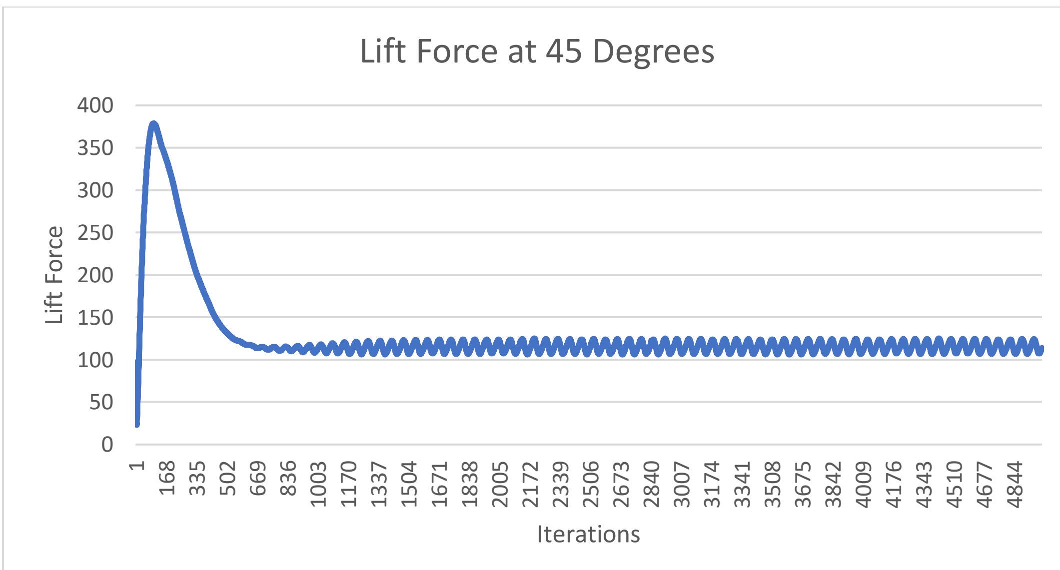
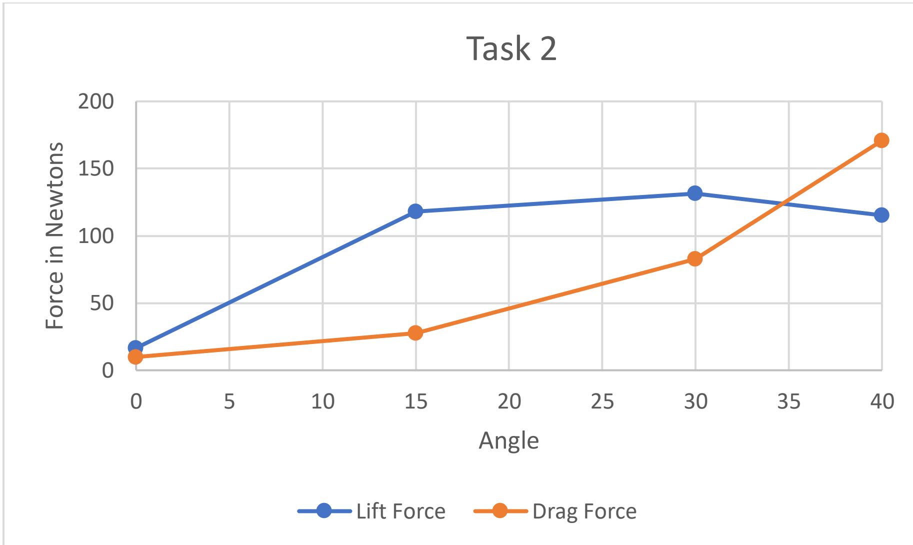


Figure 14 - Lift Force at 45 Degrees

The values for the lift and drag forces were compiled into a table below.

	0°	15°	30°	45°
Lift Force	16.55607	117.992	131.43	115.4468
Drag Force	9.881634	27.7779277	82.7543458	170.870358

With these values and angles, a line plot for both lift force and drag force was plotted. This line plot is shown below.



Task 3:

In this task, the goal was to test the aerodynamics of a pentagon shaped building in a virtual wind tunnel. The model was set up like the previous models; create the structure, create a boundary, use the bouillon command to subtract the structure from the boundary; name every surface except the subtracted surface and run the calculations. By doing this, the flow of the fluid around the structure can be calculated. In this task, the inlet velocity was set to an uniform 50 m/s and the exit was set to outflow. All the walls were named, and the building was named by ANSYS as wall-fluid-body. It was set to a turbulent model and seeking a steady solution. The simulation ran to 10,000 iterations for both task 3a and 3b.

A) In task 3a, the simulation was performed with the wind coming from the inlet attacking the flat edge of the pentagon.

(i) The deliverables that were required were the contour plots of static pressure and y-velocity along the horizontal plane with z = 0.5 m. The following contour plots are shown in figure 15 and 16.

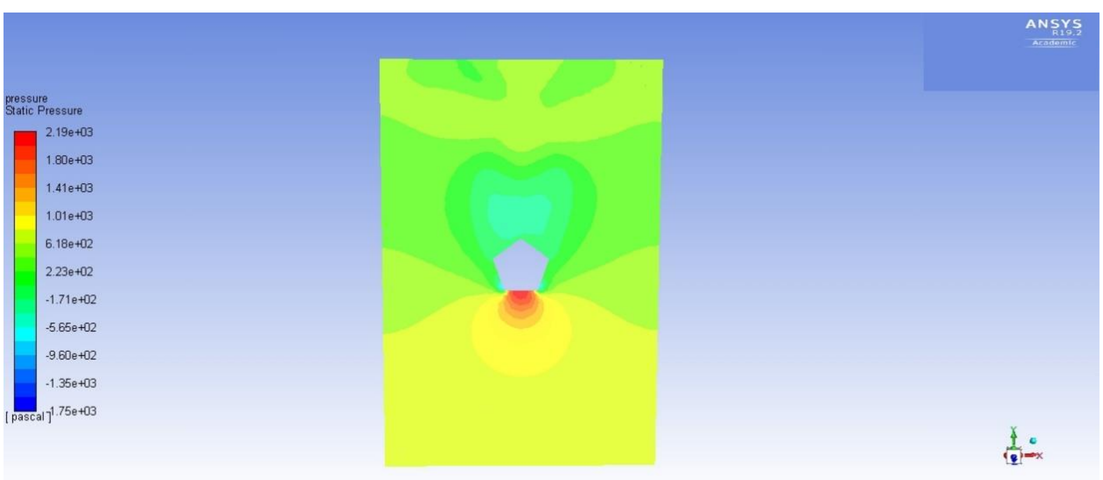


Figure 15 - Static Pressure Along the Horizontal Plane

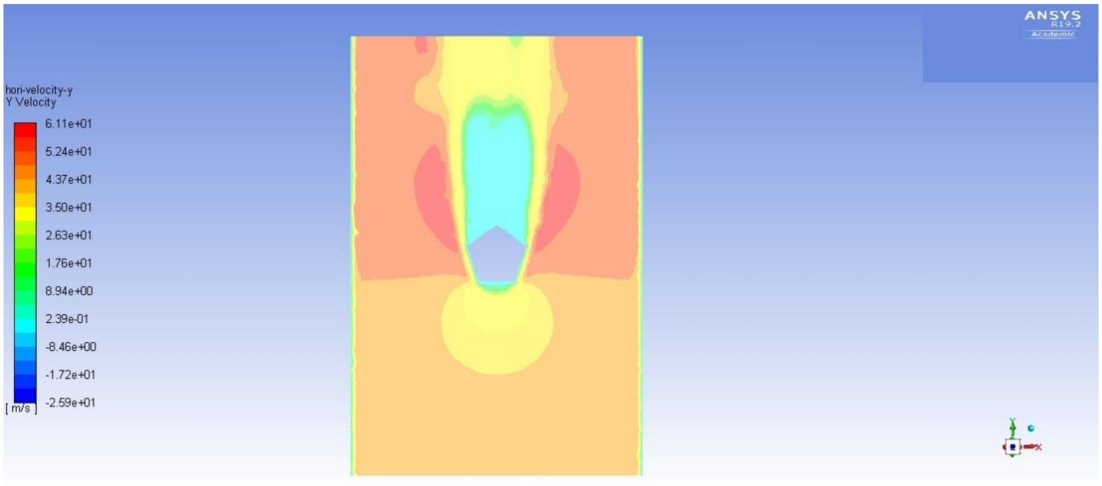


Figure 16 - Y-Velocity Along the Horizontal Plane

(ii) The next deliverable for task 3a was to show the contour plot of the y-velocity along the plane of symmetry. This contour plot is shown below in figure 17.

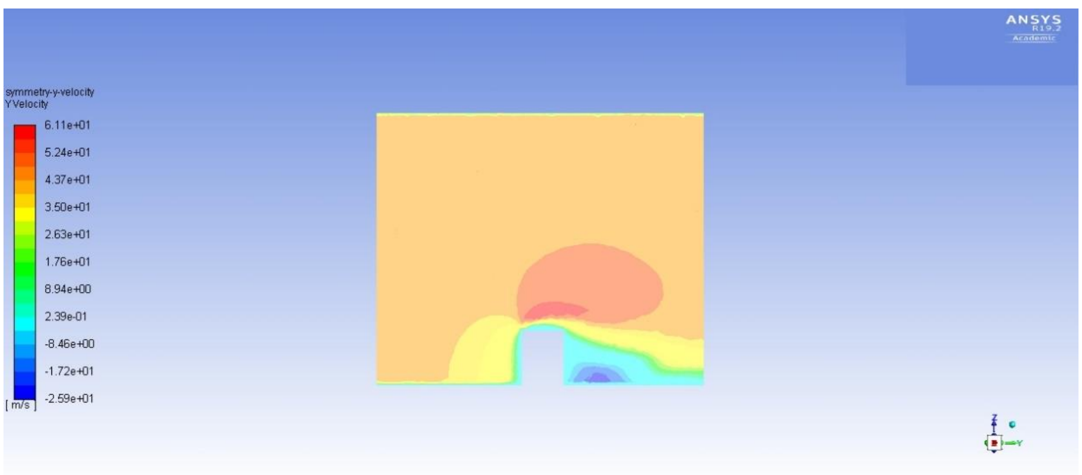


Figure 17 - Y-Velocity Along the Plane of Symmetry

(iii) The final deliverable for task 3a was to state the values of total drag force and the individual contributions to the drag force from the pressure and viscous terms. These terms are shown below in the ANSYS chart.

```
Forces - Direction Vector (0 1 0)
                            Forces (n)
Zone                        Pressure         Viscous          Total
wall-fluid-body             905.21057        2.600831         907.8114
--------------------------- ---------------- ---------------- ----------------
Net                         905.21057        2.600831         907.8114
```

Total Force = 907.8 Newtons

Force Caused by Viscous = 2.601 Newtons

Force Caused by Pressure = 905.21 Newtons

These force values were taken at the last point of the run. It was oscillating and therefore, an estimate of the middle of the wave was stopped and data was taken.

B) Task 3b was almost identical to task 3a. The key difference is that the flow was reversed going in the negative y-direction instead of the positive direction. This effectively means that the wind would be hitting the point of the pentagon first instead of a flat surface. All the deliverables required for task 3b are the same as task 3a. These deliverables are found in figures 18, 19, and 20. It is important to note that the y-velocities are negative because the flow is flowing in the negative y direction.

(i)

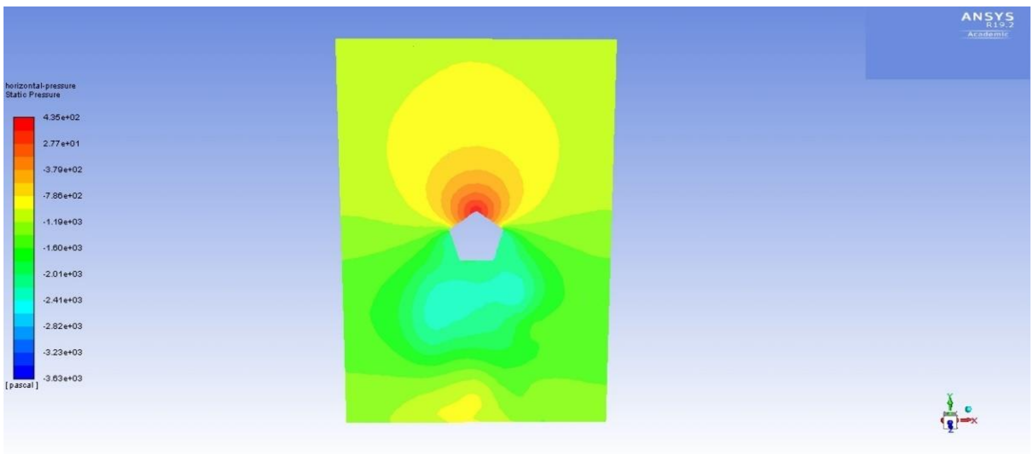


Figure 18 - Static Pressure Along the Horizontal Plane (Reversed Air Flow)

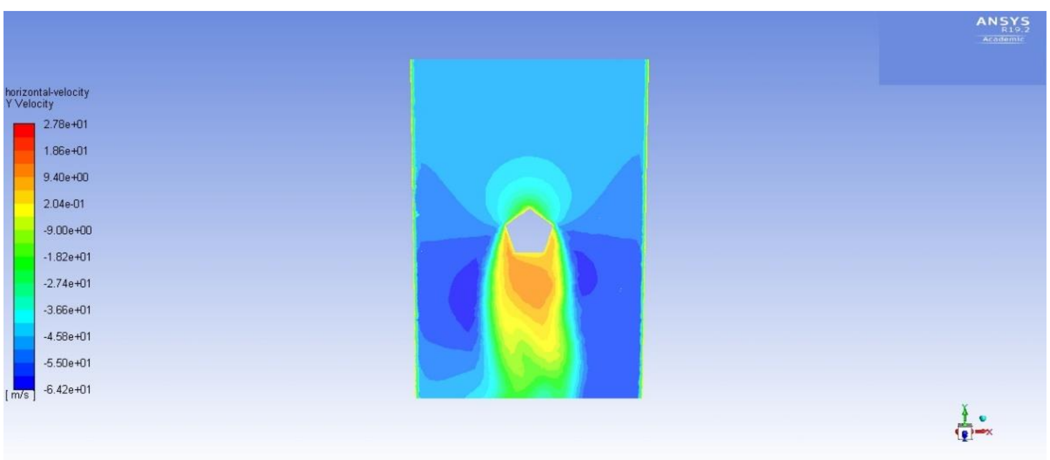


Figure 19 - Y-Velocity Along the Horizontal Plane (Reversed Air Flow)

(ii)

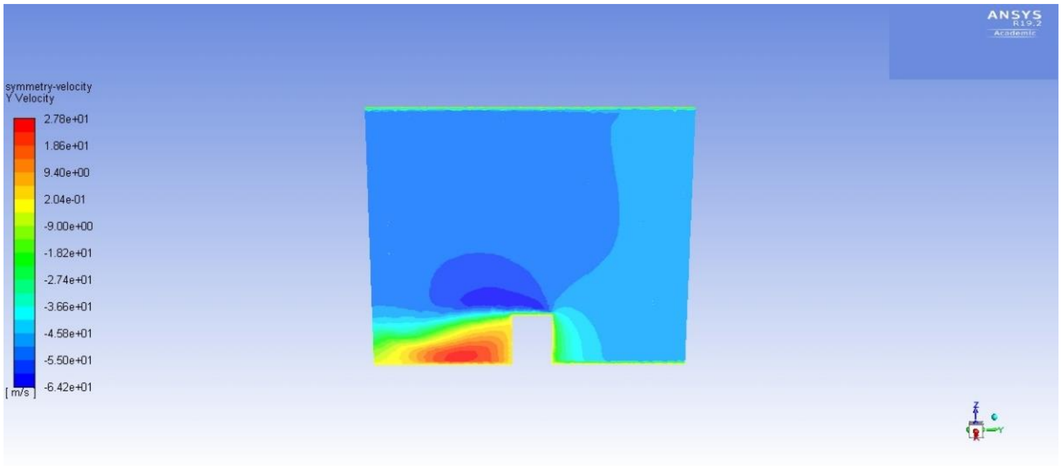


Figure 20 - Y-Velocity Along the Plane of Symmetry (Reversed Air Flow)

(iii) The forces are shown to be negative because the forces being applied to the building are in the negative y direction.

```
Forces - Direction Vector (0 1 0)
                          Forces (n)
Zone                      Pressure        Viscous         Total
wall-fluid-body           -1410.7133      -0.49003351     -1411.2033
------------------------- --------------- --------------- ---------------
Net                       -1410.7133      -0.49003351     -1411.2033
```

Total Force = 1411.2 Newtons

Forces cause by Pressure = 1410.71 Newtons

Forces Caused by Viscous = .49 Newtons

These force values were taken at the last point of the run. It was oscillating and therefore, an estimate of the middle of the wave was stopped and data was taken.

The drag in task 3A was smaller than the drag in task 3b. One hypothesis on why this is the result is because in task 3b, the cross-sectional area was shown more to the flow of the fluid. The fluid flow in task 3b was being distributed along both sides of the building and not just one surface. Another hypothesis is that the fluid flow was hitting the vertex of the pentagon, causing a bigger pressure difference. The cross-sectional area of the vertex is very small; causing a bigger pressure compared to task 3a because P=F/A. If the forces are still the same, then a smaller area will cause a bigger pressure difference. This orientation would have decreased the drag due to the viscosity of the fluid but increased the drag caused by the pressure.