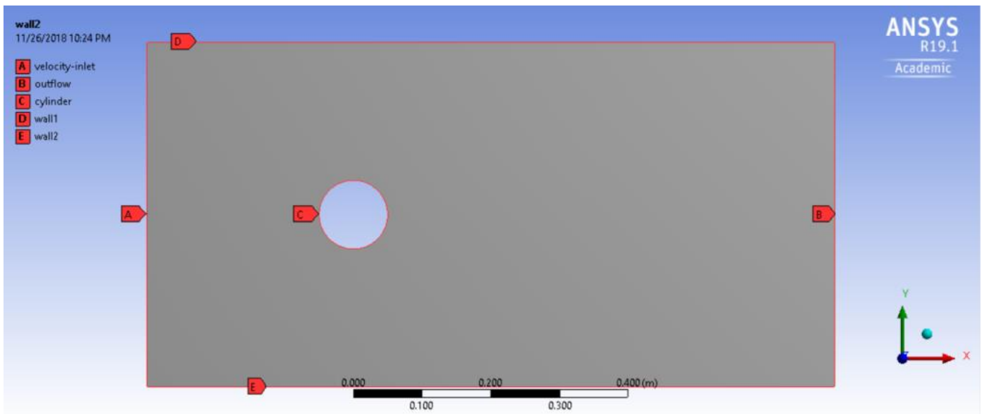
Ramteja Reddy Kondakindi

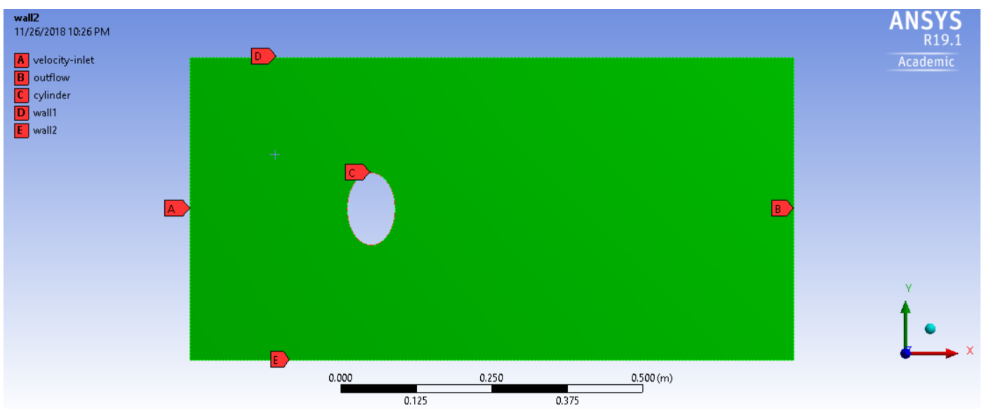
MAE 598
Applied Computational Fluid Dynamics
Project 4

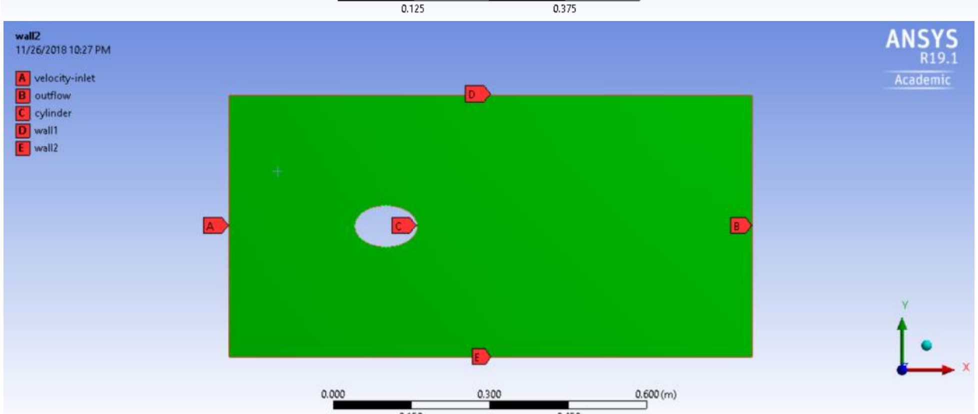
NO COLLABORATION

External Flow Simulations
Task 1:

Background:
This task involves a 2-D water flow passing over a cylinder (circular and elliptical). The geometries are created in ANSYS design modeler. The circular cylinder has a radius of 5cm and the elliptical cylinders have 12cm for major axes and 8cm for minor axes. The cylinder is placed in a virtual wind tunnel (100cmx50cm) to perform simulations in Fluent. The boundary conditions are velocity inlet for the inlet, and outflow on the opposite side. The other two sides are considered as walls. The remaining boundary will be created as cylinder which we will use for calculations. In addition, the inlet velocity is 0.3 cm/s for part a and 2 cm/s for part b & c, in the direction normal to the surface of the inlet and going into the tunnel. The below figures show the geometries and boundary conditions. A laminar model is used, and transient solution is solved for 1 hour.



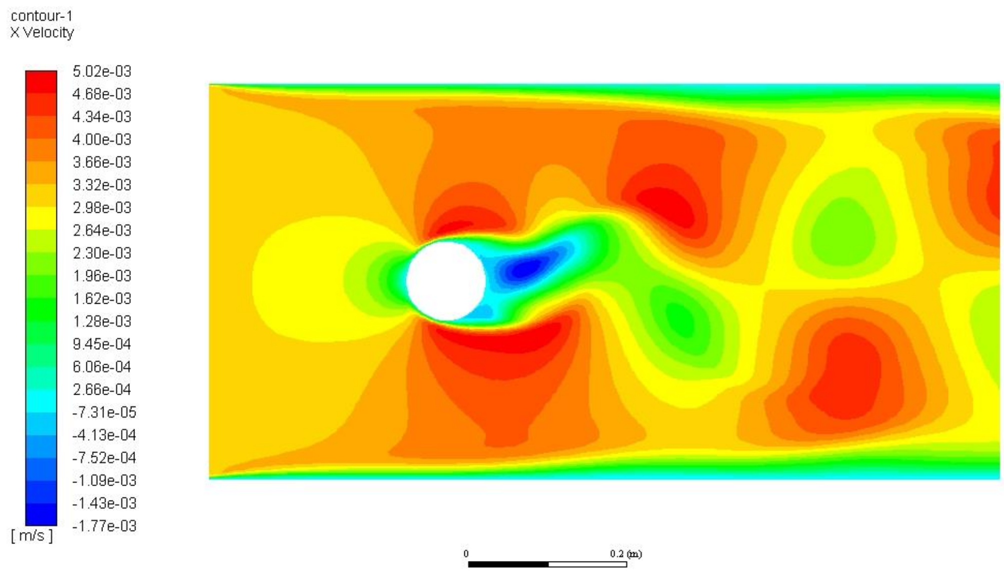




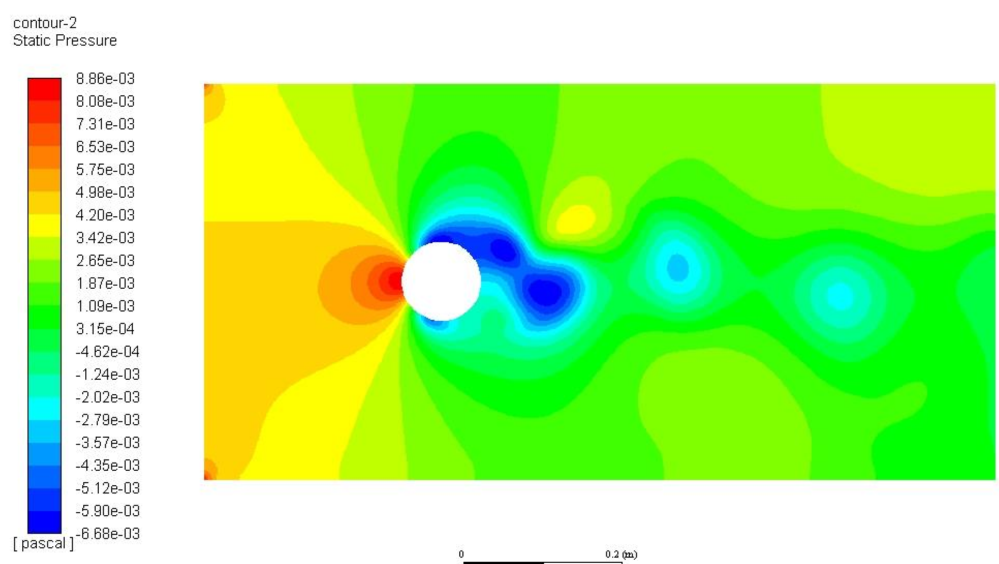
a) The Reynold's number is calculated using the equation $Re = \frac{\rho * u * D}{\mu}$

The density value is taken from Fluent database and found to be 998.2 kg/m^3, the viscosity is 0.001003 kg/(m*s). The estimated Reynold's number is 298.564.

Contour Plot of x-velocity at t = 1hr.



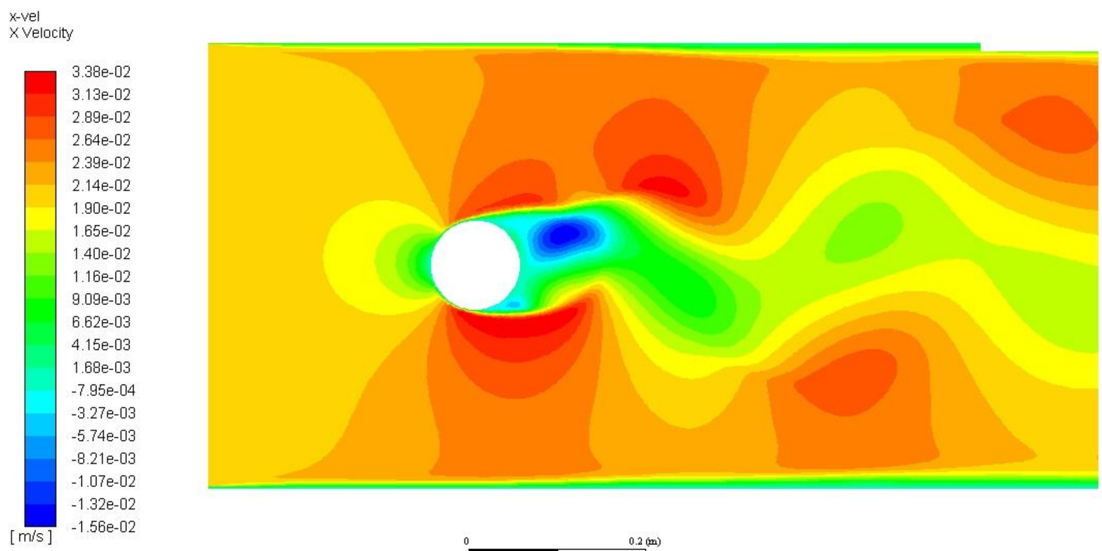
Contour Plot of Static Pressure at t = 1hr.



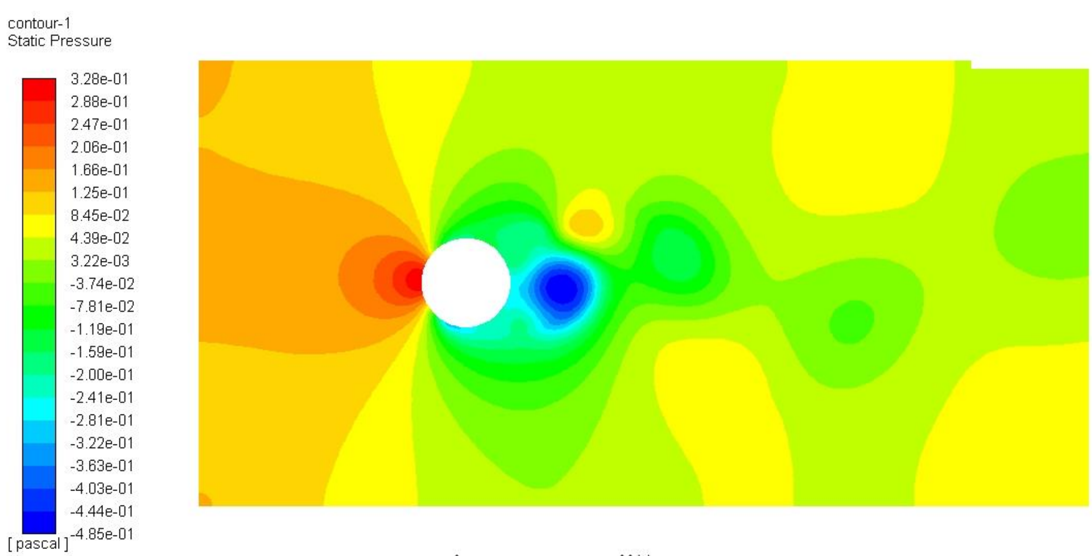
b) The Reynold's number is calculated using the equation $Re = \frac{\rho * u * D}{\mu}$

The density value is taken from Fluent database and found to be 998.2 kg/m^3, the viscosity is 0.001003 kg/(m*s). The estimated Reynold's number is 1990.43.

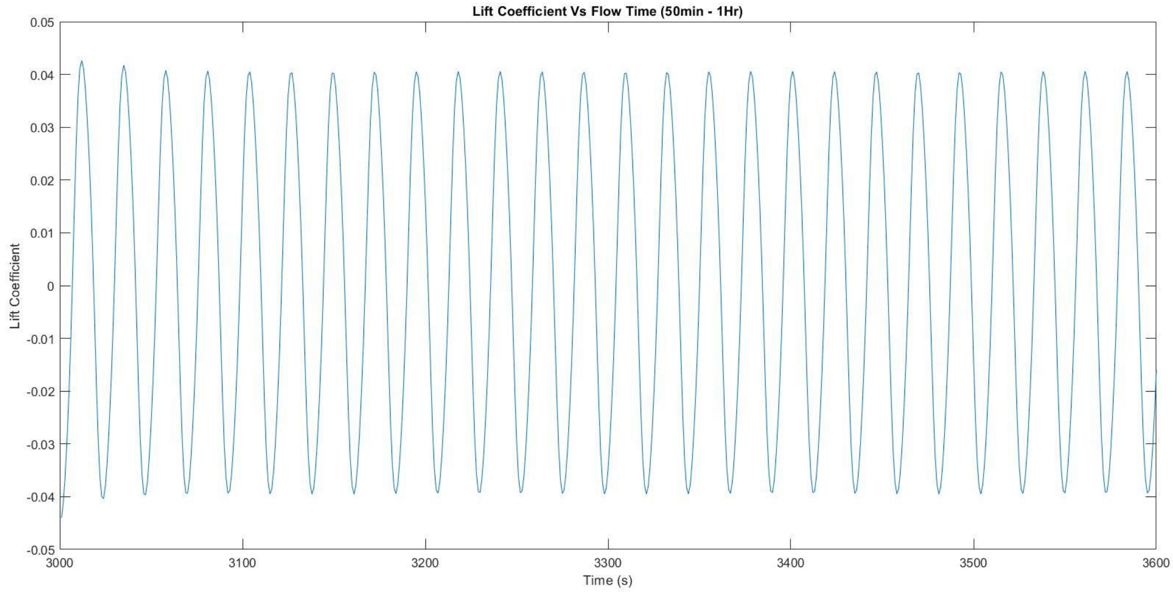
Contour Plot of x-velocity at t = 1hr.



Contour Plot of Static Pressure at t = 1hr.

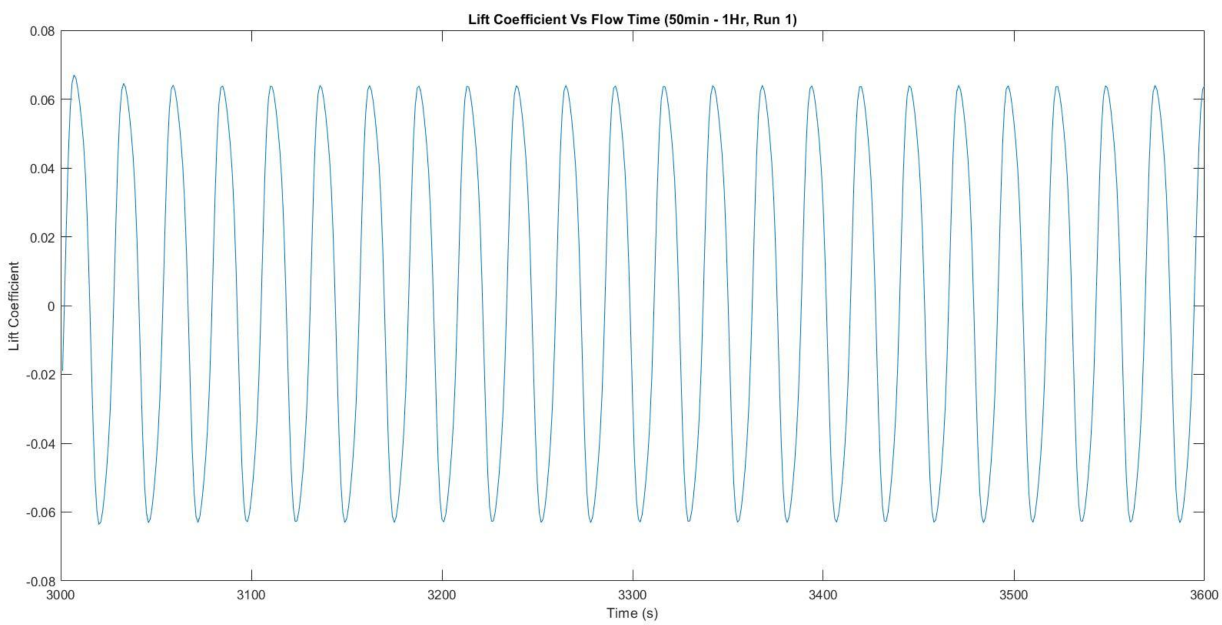


Plot of Lift Co-efficient as a function of time at t = 50min - 1hr.



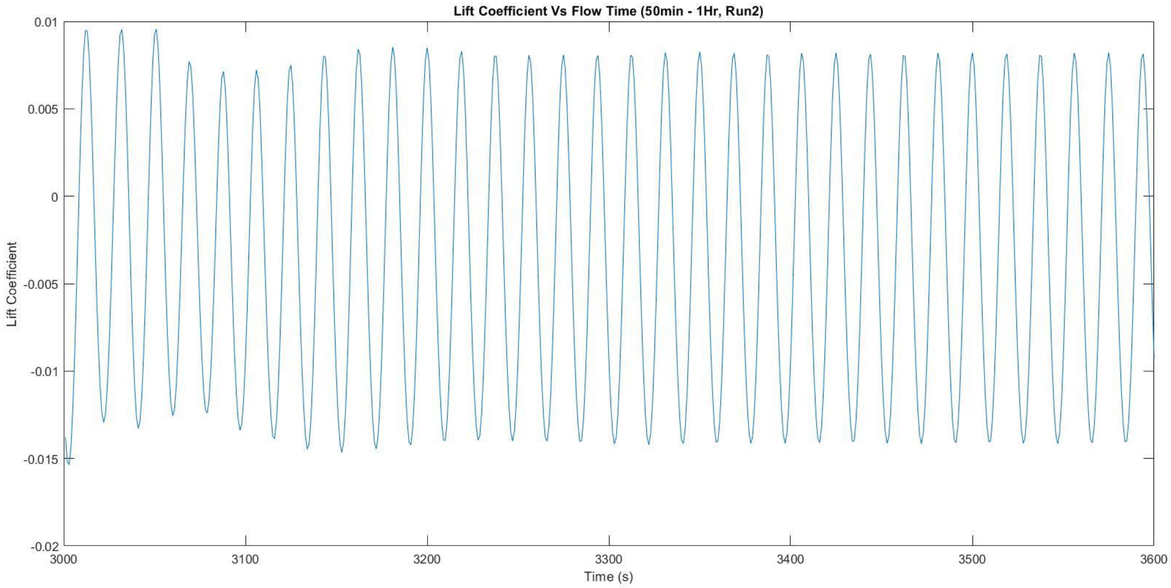
From the plot we can see the amplitude is 0.04067 and the period is 23 seconds.

c) Plot of Lift Co-efficient as a function of time at t = 50min - 1hr for Run1.



From the above plot we can see the amplitude is 0.06382 and the period is 25 seconds for Run1.

Plot of Lift Co-efficient as a function of time at t = 50min - 1hr for Run2.



From the above plot we can see the amplitude is 0.008528 and the period is 19 seconds for Run2. The amplitude when the flow is over a circular cylinder is in between the amplitudes of Run 1 and Run 2. Similarly, the period is in between run1 and run 2.

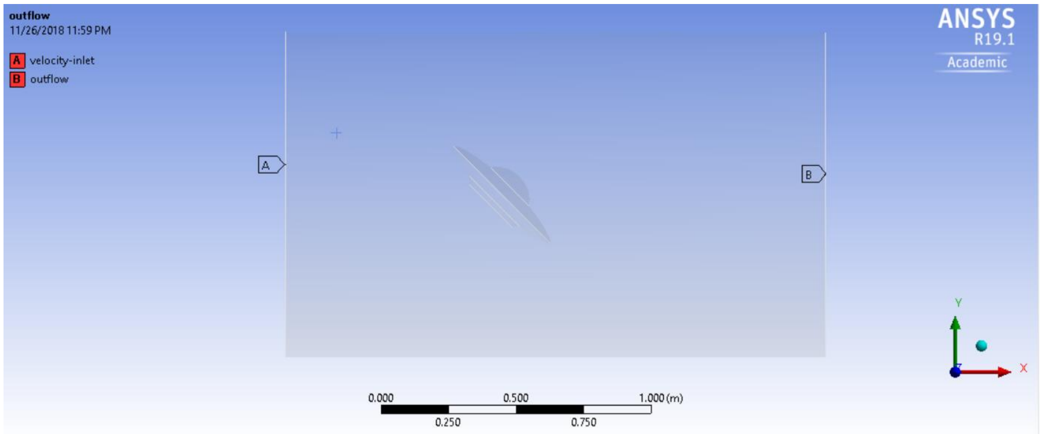
Run1 Amplitude > Amplitude of lift coefficient over circular cylinder > Run 2 Amplitude.

Run1 Period > Period of lift coefficient over circular cylinder > Run 2 Period.

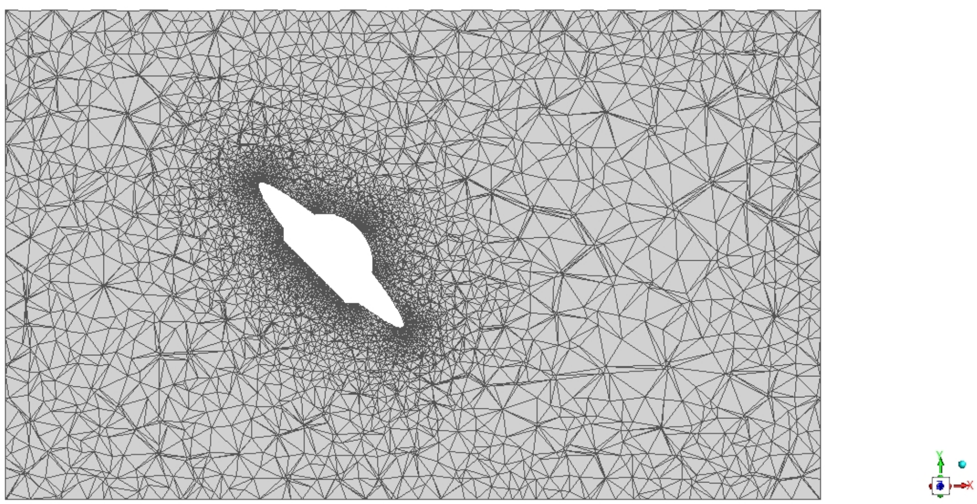
Task 2:

Background:

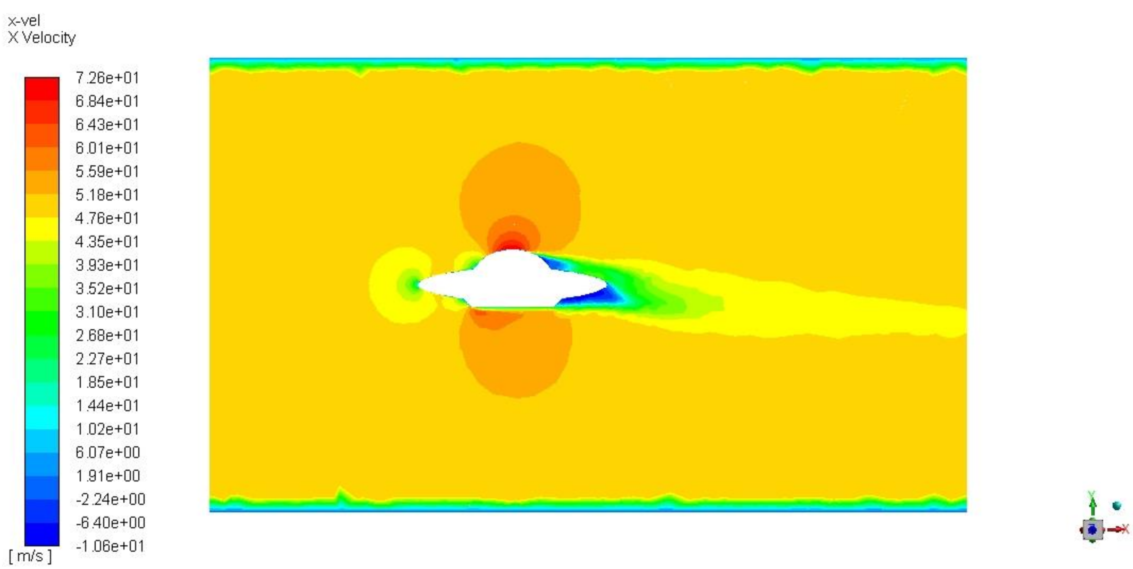
This task involves the aerodynamics of a 3-D "flying saucer" in a cylindrical virtual wind tunnel. The geometry is created using data coordinates file in design modeler. The radius and length of the cylindrical tube are 60 cm and 200 cm, respectively. The boundary conditions are velocity inlet for the inlet, and outflow on the opposite side. The cylinder is given a named selection as wall-cylinder such that the remaining boundary is created as wall-fluid by ANSYS which is used for calculating lift and drag force. The angle between the z-axis and the angle of rotation of flying saucer w.r.t z-axis is defined as tilt angle. We performed simulations for 4 different tilt angles (0°, 15°, 30°, and 45°). The tunnel is filled with air in this task with constant density and viscosity. In addition, the inlet velocity is 50 m/s, in the direction normal to the surface of the inlet and going into the tunnel. The below figures show the geometry at 45° and boundary conditions. A turbulence k-epsilon model, and steady solution is simulated.



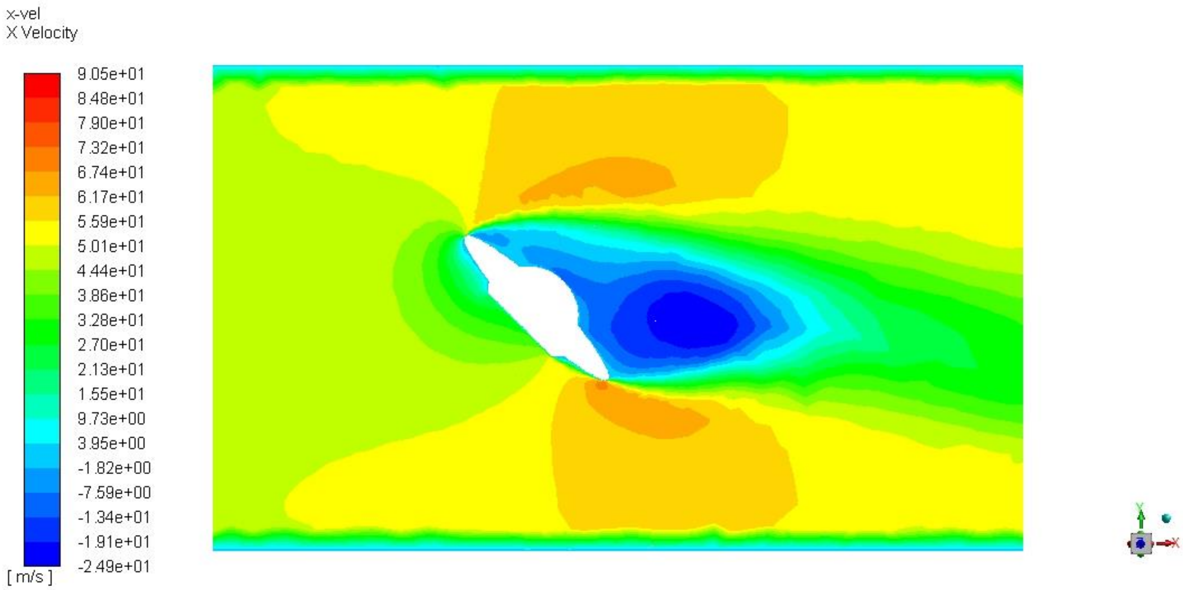
Plot of the mesh along the plane of symmetry when tilt angle is 45°



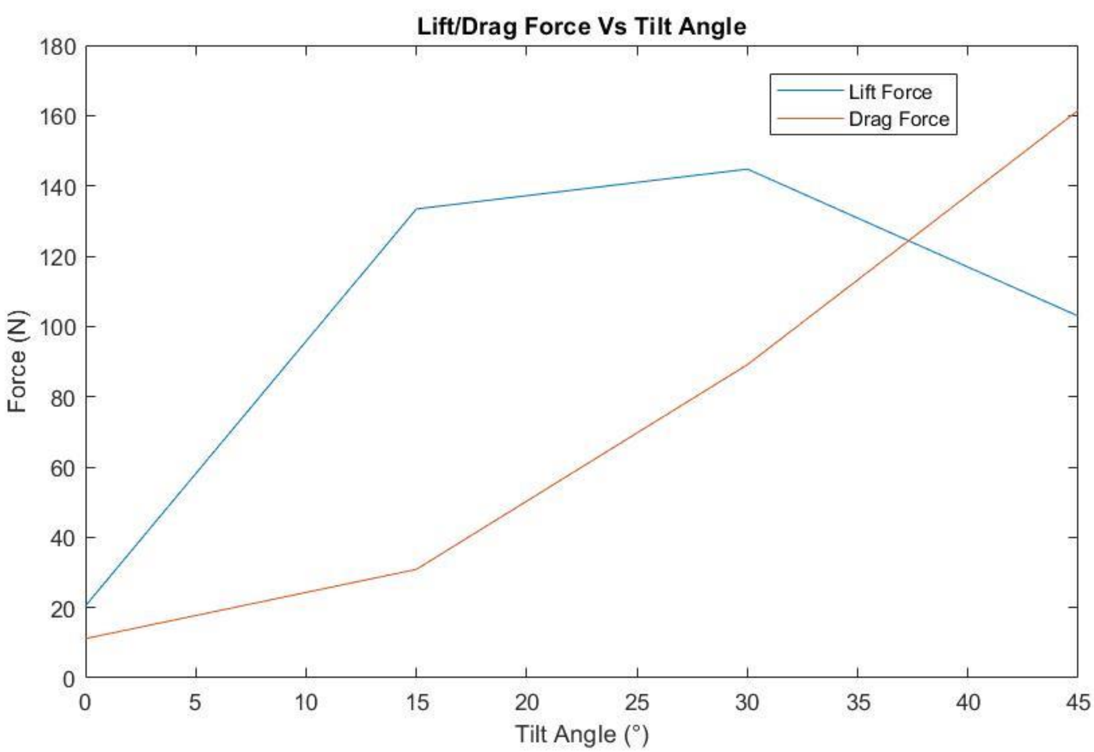
Contour plot of x-velocity along plane of symmetry when tilt angle is 0°



Contour plot of x-velocity along plane of symmetry when tilt angle is 45°



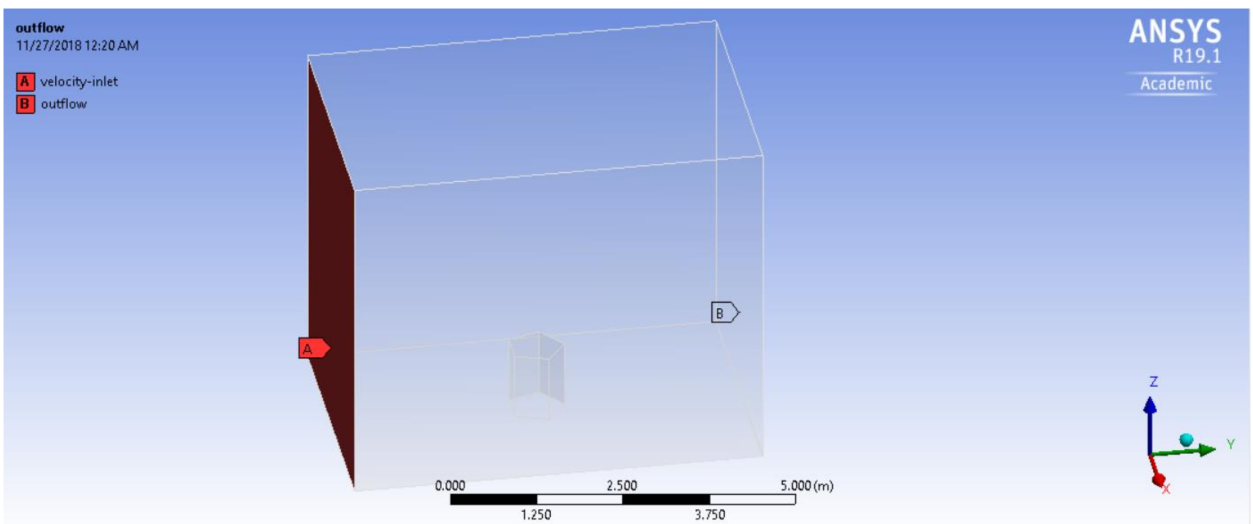
Line plot of lift and drag force as a function of tilt angle.



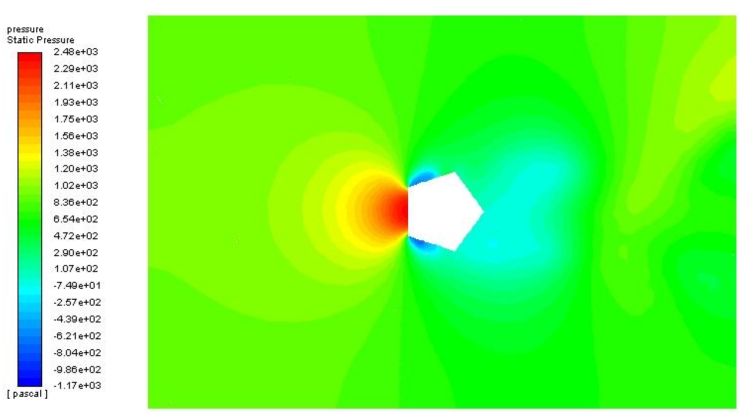
Task 3:

Background:

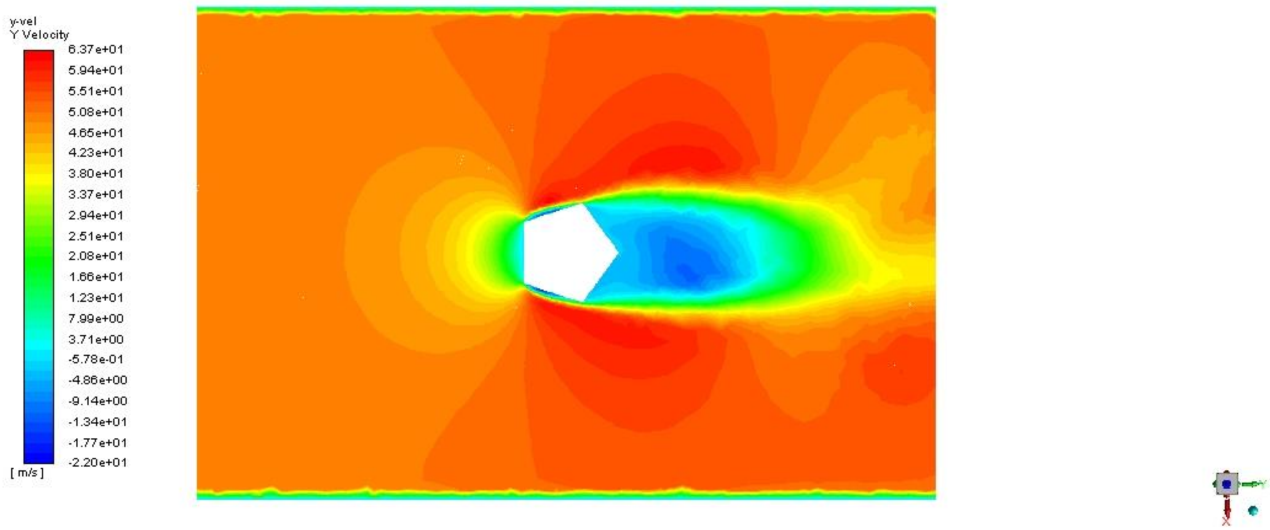
This task involves the aerodynamics of a pentagon shaped building in a virtual wind tunnel. The geometry is created using design modeler. The height of the building is 1 m and each side of the pentagon is 0.5 m. The dimensions of the wind tunnel are 4mx6mx5m. The building is placed at the bottom of the tunnel and at the center. The boundary conditions are velocity inlet for the left side, and outflow on the opposite side. All other sides of the tunnel are walls. The building is considered as wall-fluid by ANSYS which is used for calculations. The tunnel is filled with air in this task with constant density and viscosity. In addition, the inlet velocity is 50 m/s, in the direction normal to the surface of the inlet and going into the tunnel. The below figures show the geometry and boundary conditions. A turbulence k-epsilon model, and steady solution is simulated. In part A one side of the building faces the velocity inlet boundary and in part B the vertex of the building faces the velocity inlet boundary.



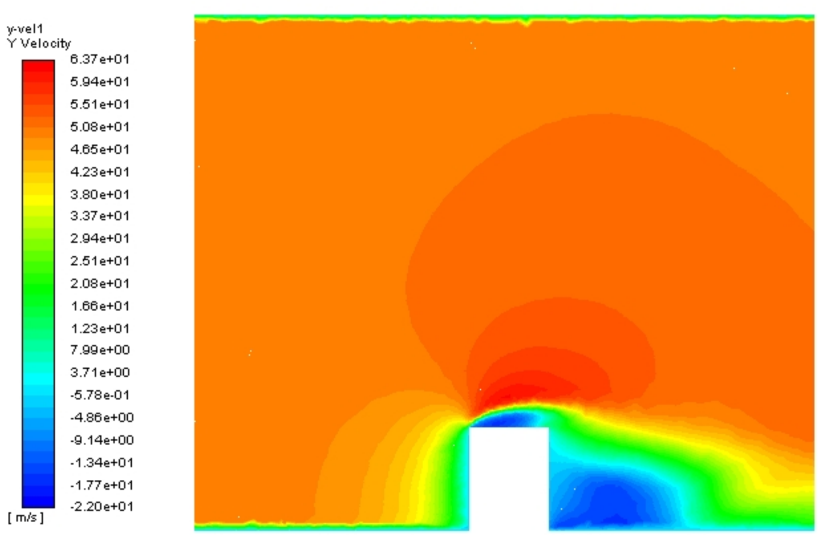
a) Contour plot of steady pressure along the horizontal plane.



Contour plot of y-velocity along the horizontal plane.

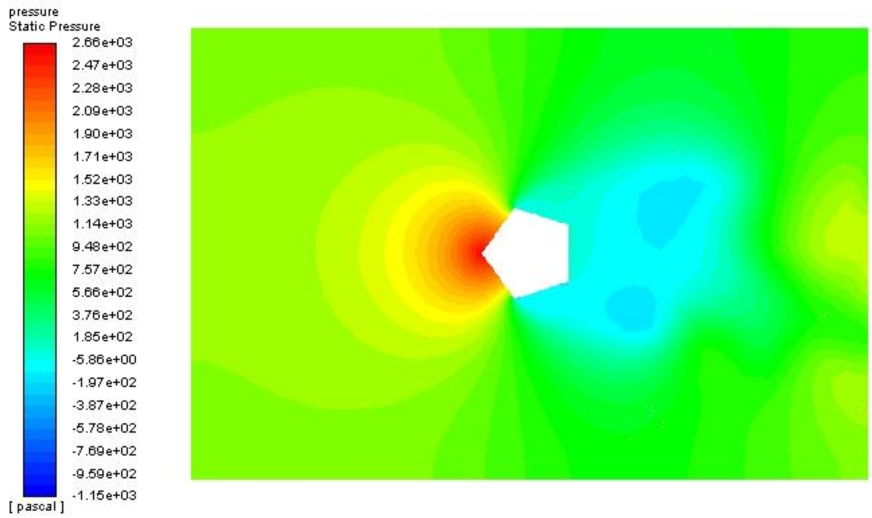


Contour plot of y-velocity along the plane of symmetry.

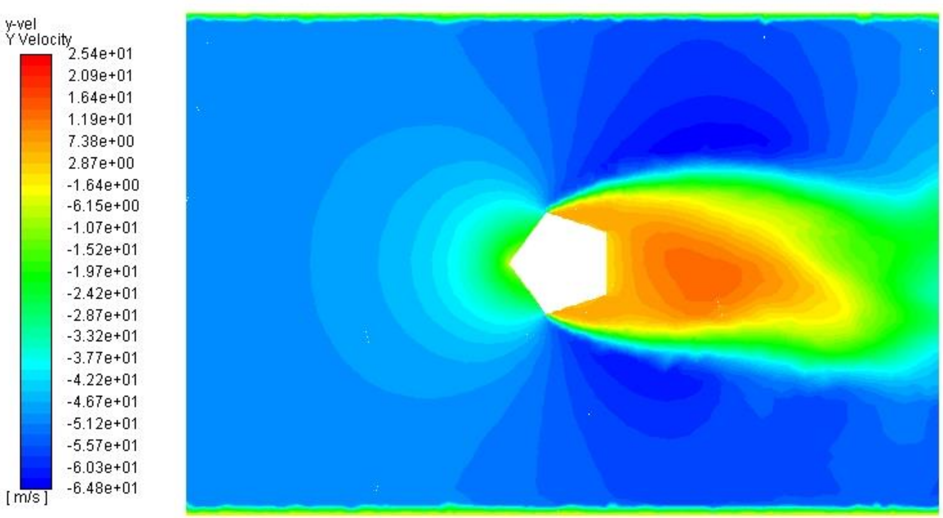


The total drag force in the case when wind blows towards a side of the building is 866.78064 N and the pressure force is 867.83484 N and the viscous force is -1.0541964 N.

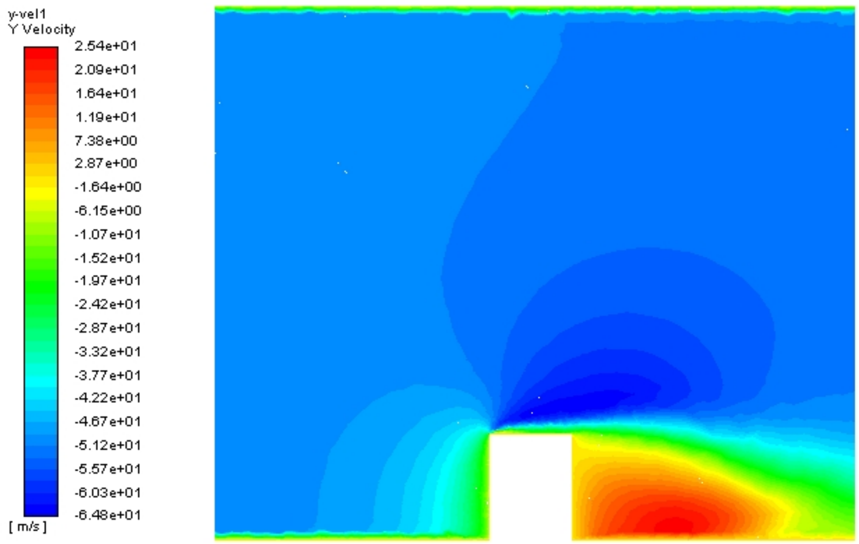
b) **Contour plot of steady pressure along the horizontal plane.**



Contour plot of y-velocity along the horizontal plane.



Contour plot of y-velocity along the plane of symmetry.



The total drag force in the case when the wind blows towards vertex of the building is 1441.1706 N and the pressure force is 1439.6096 N and the viscous force is 1.5609715 N.

From the contour plots of pressure, we can see, when the wind blows over the flat edge of the building the pressure is uniformly acting on the face. In the case where the vertex of the building is facing the inlet the vertex edge is splitting the flow and the wind blows on to the two sides of the vertex and thus increasing the pressure force and drag force.

Similarly, from the contour plots of y-velocity we can see, when the wind blows over the flat edge the maximum flow velocity is around 63.7 m/s when compared to part b where the maximum flow velocity is around 25.4 m/s. This is because when the vertex is facing the wind flow direction, the flow is obstructed more than the flat edge scenario.

As it is a pentagon the vertex edge is making an angle of 120° with the two sides. If the angle is decreased the maximum flow velocity will increase and pressure force decreases.