

## MAE 560 PROJECT 2 : APPLIED CFD

COLLABORATION STATEMENT : – *No Collaborations*

Note : – *Several mesh figures might seem distorted due to high number of mesh elements stuffed in the domain to get better accuracy, however, upon zooming in, the mesh displays no distortion and displays the actual shape of each elements. The solution contours might have the same kind of problem, this is due to the rather blur and less bright impressions of contours being produced by ANSYS solution section as compared to CFD-Post toolbox. For this reason, several colormaps have been customized to make the contours more distinct and readable especially in task 3 for static pressure.*

### TASK 1 : – 2-D FLOW PASSING A CYLINDER

Analysis type of system used in project – **2D** analysis

Following figure displays the dimensions of the system geometry: –

#### CASE 1: - CYLINDER WITH DIAMETER 8 CM

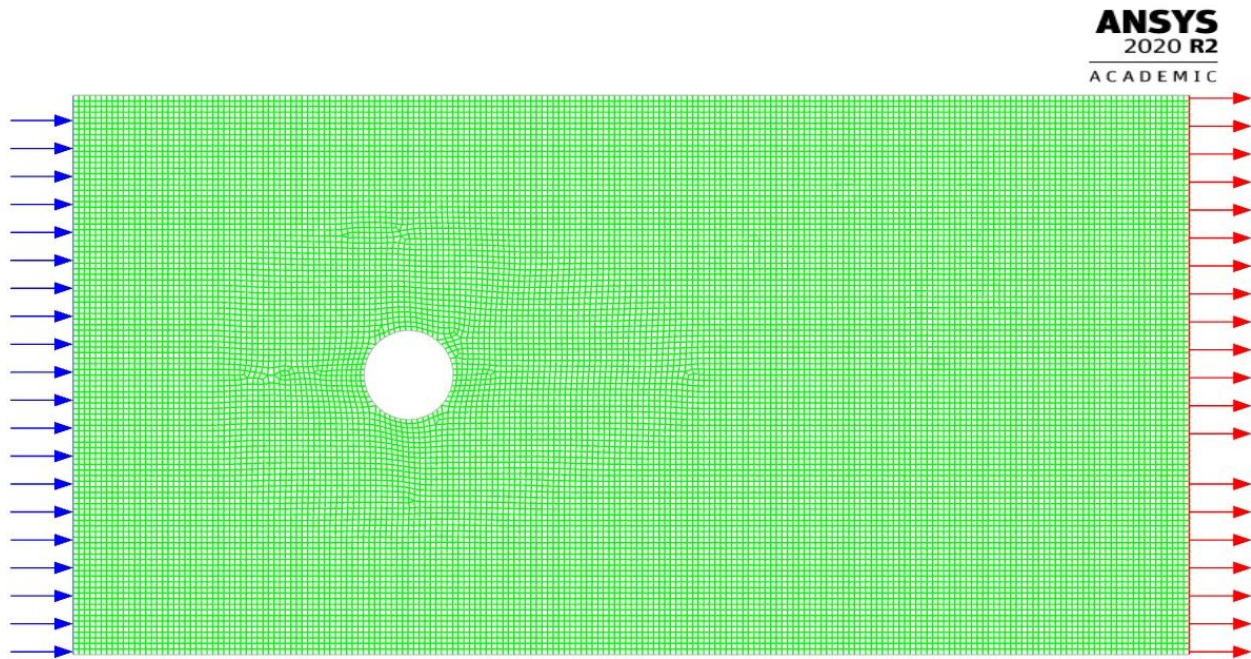


Fig. 1 Setup of the given cylinder geometry as viewed in ANSYS Fluent Solver with linear element size of 5 mm

Fluids used for CFD analysis = *Water - liquid*  
Solver type used = *Pressure-based Solver*  
Solver used = *Transient Solver*

#### MODELS USED: –

1. Viscous Model is turned on for Laminar model.
2. Gravity is turned off, since it was not mentioned.

**BOUNDARY CONDITIONS USED: –**

1. Velocity at inlet has *x-velocity* and *y-velocity* set at 4 cm/s and 0 cm/s respectively for inlet and *gauge pressure* at 0 Pa for same opening.
2. The other opening was set as pressure outlet with zero gauge pressure.
3. The other sides of the domain were by default set as stationary wall.

**METHODS USED: –**

1. Pressure Velocity Coupling Scheme => SIMPLE Scheme.
2. Spatial Discretization Methods used are
  - a. Gradient => Least Squares Cell Based
  - b. Pressure => Second Order
  - c. Momentum => Second Order Upwind
  - d. Transient Formulation => First Order Implicit

**INITIALIZATION USED: –**

1. Initialization Method = Standard Initialization
2. Gauge Pressure = 0 Pa
3. X Velocity = 0.04 m/s
4. Y Velocity = 0.00 m/s

**TIME ADVANCEMENT CALCULATIONS SETTINGS USED: –**

Time step size for this case was selected as *0.005 seconds* with total number of time steps as *3600* that would result in a total simulation of *3 minutes* with *20 iterations per time step*. The time step size selected was sufficiently enough to generate the results without much numerical error that would hinder the final results of the system under this specific flow.

**(i) Reynolds number estimation for the above given system**

$$\text{Re} = \frac{\rho v D}{\mu}$$

Here for this system,

$$\rho = 998.2 \text{ kg/m}^3$$

$$v = 0.04 \text{ m/s}$$

$$D = 0.08 \text{ m}$$

$$\mu = 0.001003 \text{ kg/m-s}$$

$$\text{Re} = \frac{998.2 * 0.04 * 0.08}{0.001003} = 3184.685$$

**(ii) Contour Plots**

Contour plot of static pressure as viewed from plane of computation is

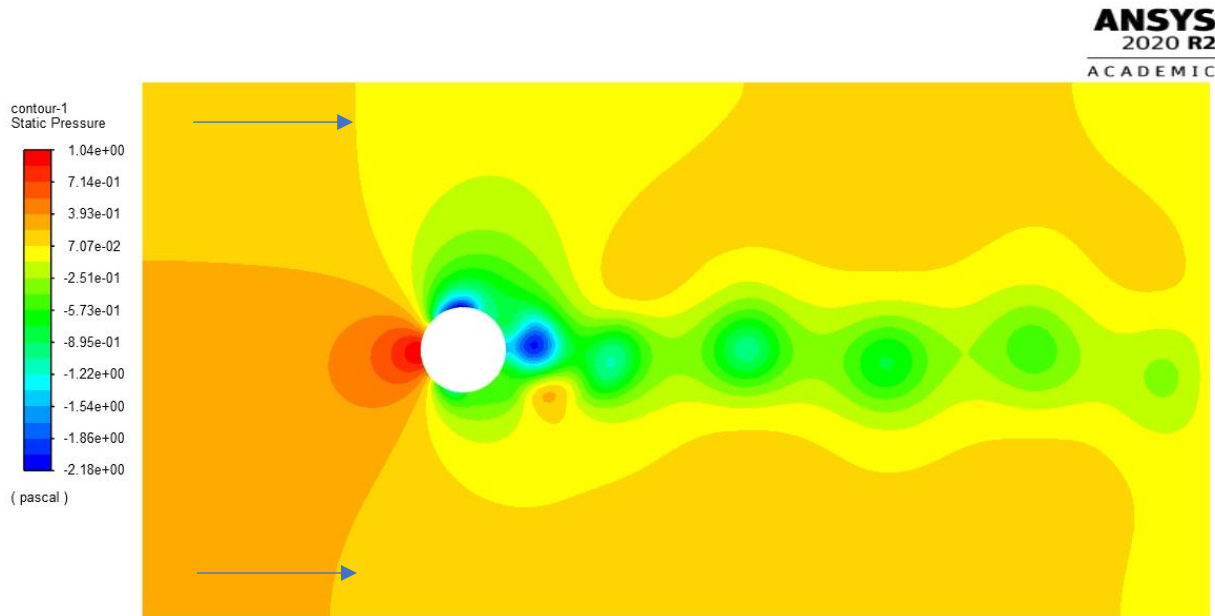


Fig. 2 Contour plot of static pressure of fluid in domain with external cylindrical geometry obstructing the free flow of water

Contour plot of Y - velocity as viewed from plane of computation is

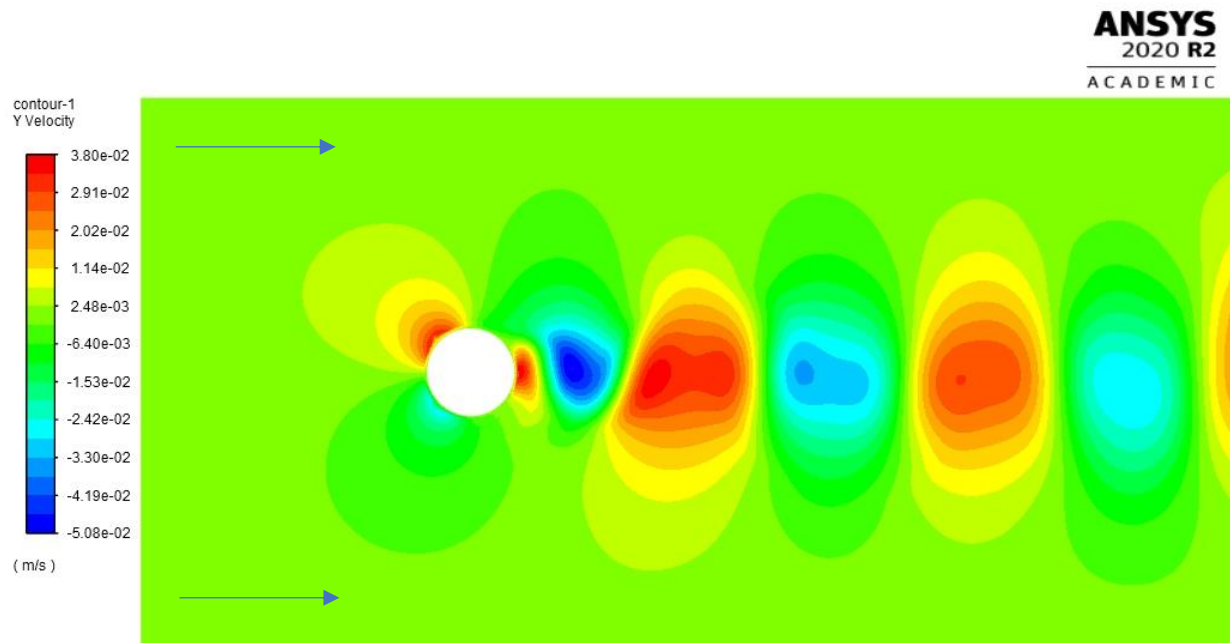


Fig. 3 Contour plot of Y-velocity of fluid in domain with external cylindrical geometry obstructing the free flow of water

Contour plot of stream function as viewed from plane of computation is

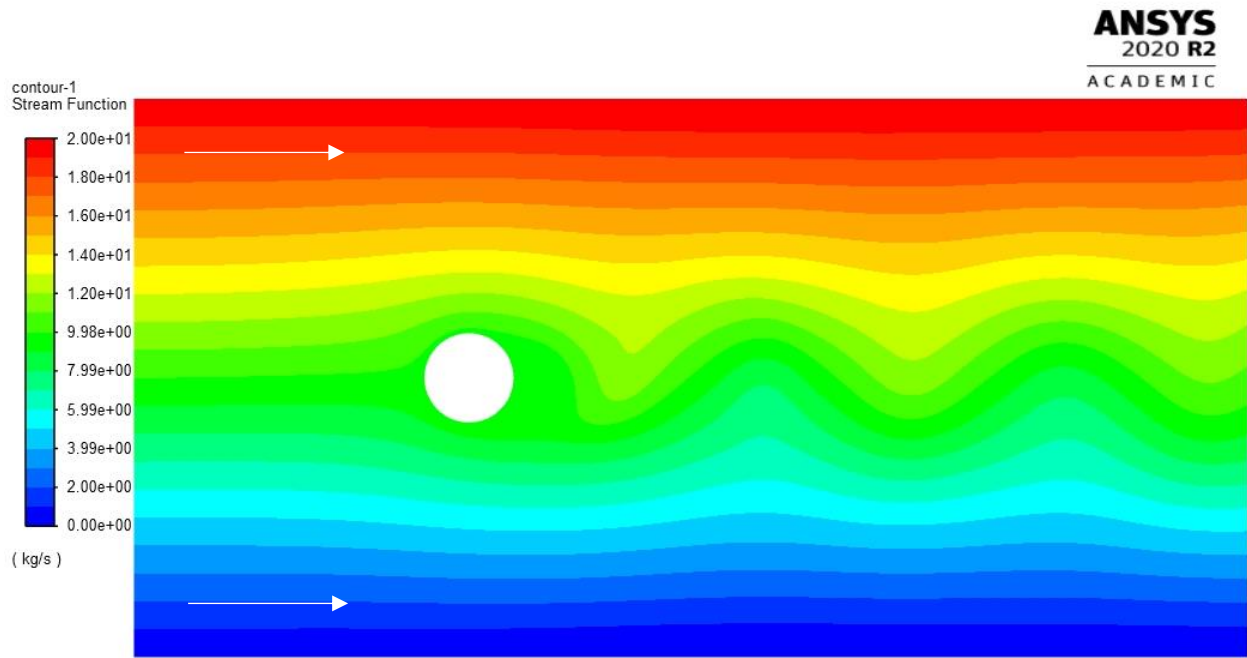


Fig. 4 Contour plot of stream function of fluid in domain with external cylindrical geometry obstructing the free flow of water

**(iii) Line Plot of the lift force**

The line plot of the lift force with respect to time flow is as shown below: -

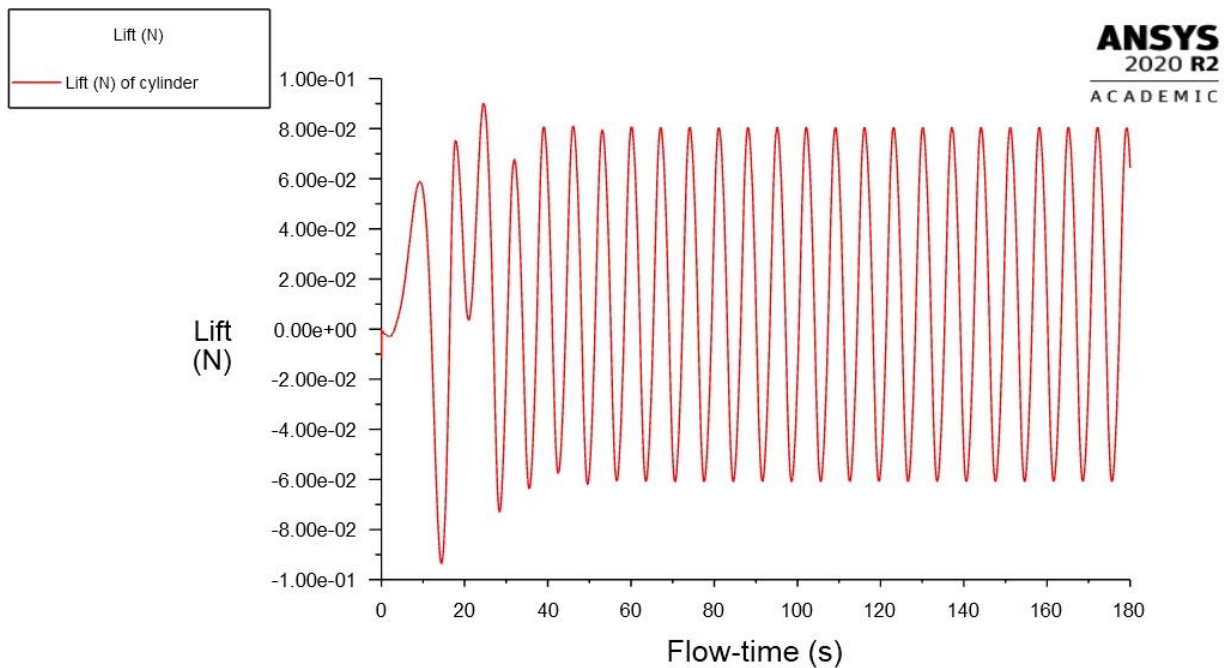


Fig. 5 Line plot of lift force from time = 0 seconds to time = 180 seconds (3 minutes) for given geometry

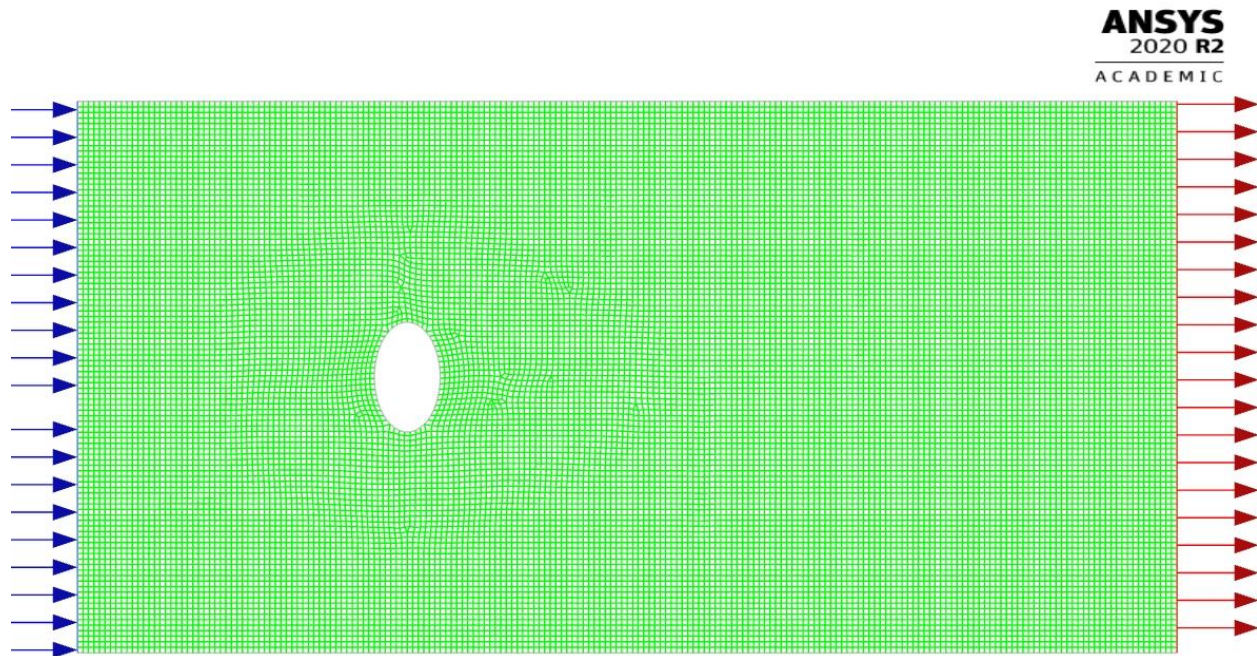
**CASE 2: – VERTICAL ELLIPSE WITH MAJOR & MINOR AXIS OF 10 CM & 6 CM RESPECTIVELY**

Fig. 6 Setup of the given vertical ellipse geometry as viewed in ANSYS Fluent Solver with linear element size of 5 mm

Fluids used for CFD analysis = *Water - liquid*  
Solver type used = *Pressure-based Solver*  
Solver used = *Transient Solver*

**MODELS USED: –**

1. Viscous Model is turned on for Laminar model.
2. Gravity is turned off, since it was not mentioned.

**BOUNDARY CONDITIONS USED: –**

1. Velocity at inlet has *x-velocity* and *y-velocity* set at 4 cm/s and 0 cm/s respectively for inlet and *gauge pressure* at 0 Pa for same opening.
2. The other opening was set as pressure outlet with zero gauge pressure.
3. The other sides of the domain were by default set as stationary wall.

**METHODS USED: –**

1. Pressure Velocity Coupling Scheme => SIMPLE Scheme.
2. Spatial Discretization Methods used are
  - a. Gradient => Least Squares Cell Based
  - b. Pressure => Second Order
  - c. Momentum => Second Order Upwind
  - d. Transient Formulation => First Order Implicit

**INITIALIZATION USED: –**

1. Initialization Method = Standard Initialization
2. Gauge Pressure = 0 Pa
3. X Velocity = 0.04 m/s
4. Y Velocity = 0.00 m/s

**TIME ADVANCEMENT CALCULATIONS SETTINGS USED: –**

Time step size for this case was selected as *0.005 seconds* with total number of time steps as *3600* that would result in a total simulation of *3 minutes* with *20 iterations per time step*. The time step size selected was sufficiently enough to generate the results without much numerical error that would hinder the final results of the system under this specific flow.

**(i) Line Plot of the lift force**

The line plot of the lift force with respect to time flow is as shown below: -

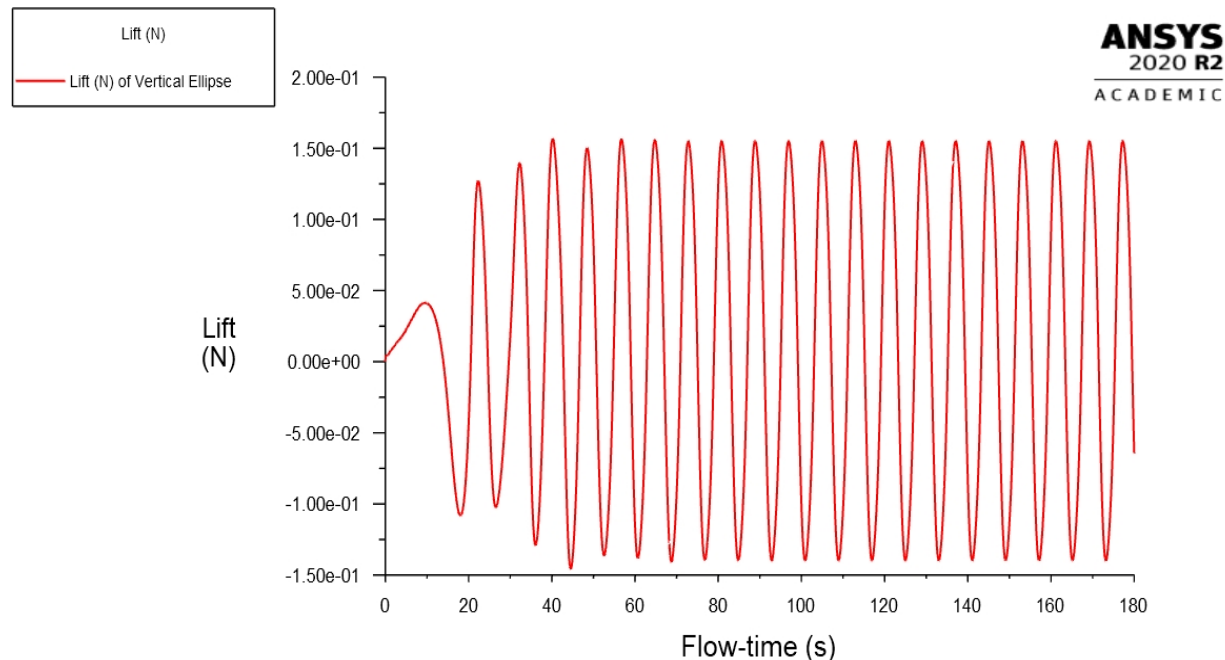


Fig. 7 Line plot of lift force from time = 0 seconds to time = 180 seconds (3 minutes) for given geometry

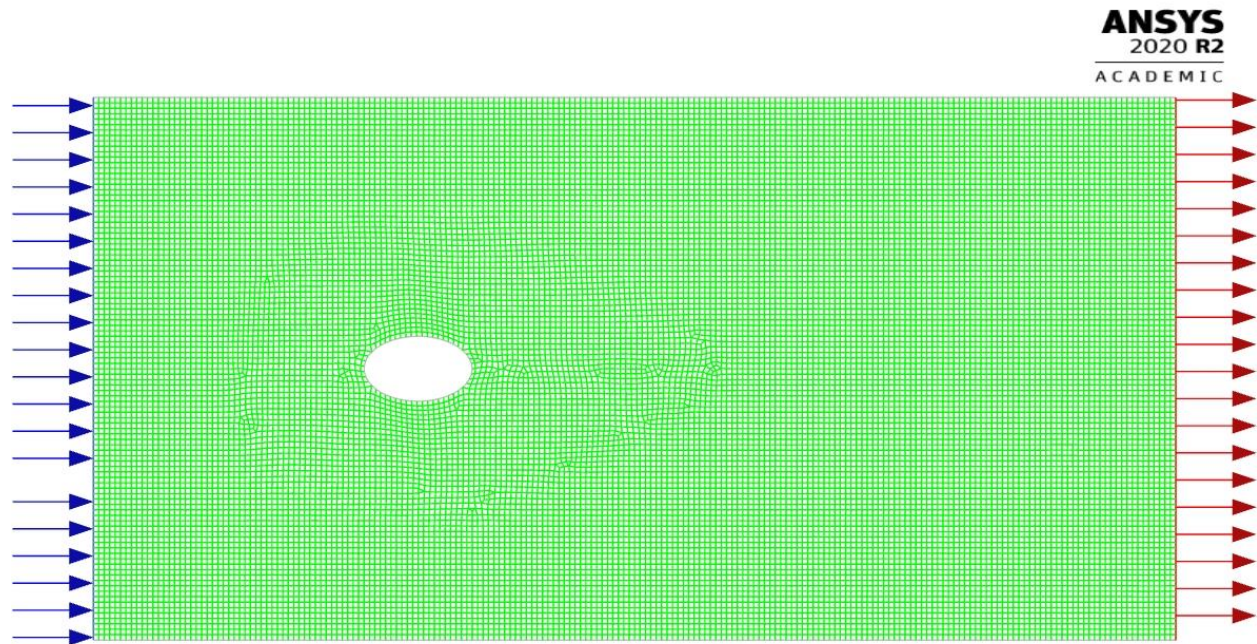
**CASE 3: – HORIZONTAL ELLIPSE WITH MAJOR & MINOR AXIS OF 10 CM & 6 CM RESPECTIVELY**

Fig. 8 Setup of the given horizontal ellipse geometry as viewed in ANSYS Fluent Solver with linear element size of 5 mm

Fluids used for CFD analysis = *Water - liquid*  
Solver type used = *Pressure-based Solver*  
Solver used = *Transient Solver*

**MODELS USED: –**

1. Viscous Model is turned on for Laminar model.
2. Gravity is turned off, since it was not mentioned.

**BOUNDARY CONDITIONS USED: –**

1. Velocity at inlet has *x-velocity* and *y-velocity* set at 4 cm/s and 0 cm/s respectively for inlet and *gauge pressure* at 0 Pa for same opening.
2. The other opening was set as pressure outlet with zero gauge pressure.
3. The other sides of the domain were by default set as stationary wall.

**METHODS USED: –**

1. Pressure Velocity Coupling Scheme => SIMPLE Scheme.
2. Spatial Discretization Methods used are
  - a. Gradient => Least Squares Cell Based
  - b. Pressure => Second Order
  - c. Momentum => Second Order Upwind
  - d. Transient Formulation => First Order Implicit

**INITIALIZATION USED: –**

1. Initialization Method = Standard Initialization
2. Gauge Pressure = 0 Pa
3. X Velocity = 0.04 m/s
4. Y Velocity = 0.00 m/s

**TIME ADVANCEMENT CALCULATIONS SETTINGS USED: –**

Time step size for this case was selected as *0.005 seconds* with total number of time steps as *3600* that would result in a total simulation of *3 minutes* with *20 iterations per time step*. The time step size selected was sufficiently enough to generate the results without much numerical error that would hinder the final results of the system under this specific flow.

**(i) Line Plot of the lift force**

The line plot of the lift force with respect to time flow is as shown below: -

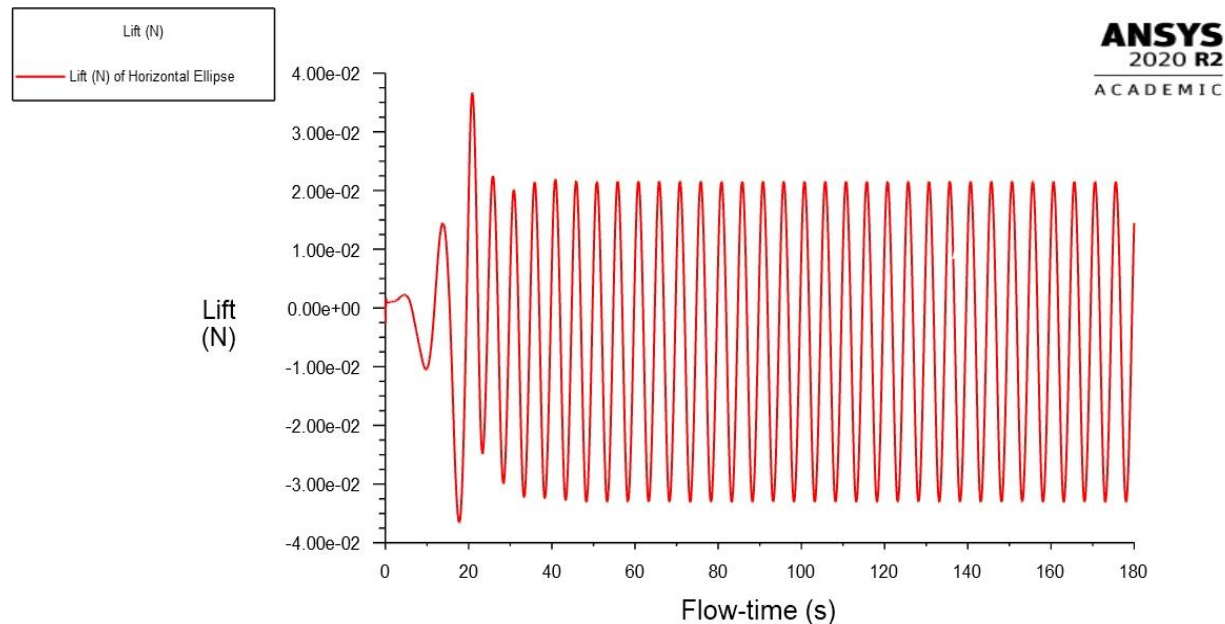


Fig. 9 Line plot of lift force from time = 0 seconds to time = 180 seconds (3 minutes) for given geometry



**(iv) Table of Amplitude and Time Period for different geometry**

	<b>Amplitude (in Newtons)</b>	<b>Time Period (in Seconds)</b>
<i>Circular Cylinder, Case 1</i>	0.07067	7.00
<i>Vertical Elliptical Cylinder, Case 2</i>	0.14720	7.95
<i>Horizontal Elliptical Cylinder, Case3</i>	0.02728	4.95

**TASK 2 : – FLYING SAUCER IN A CYLINDRICAL WIND TUNNEL**

Analysis type of system used in project – **3D** analysis

Following figure displays the setup of the system: –

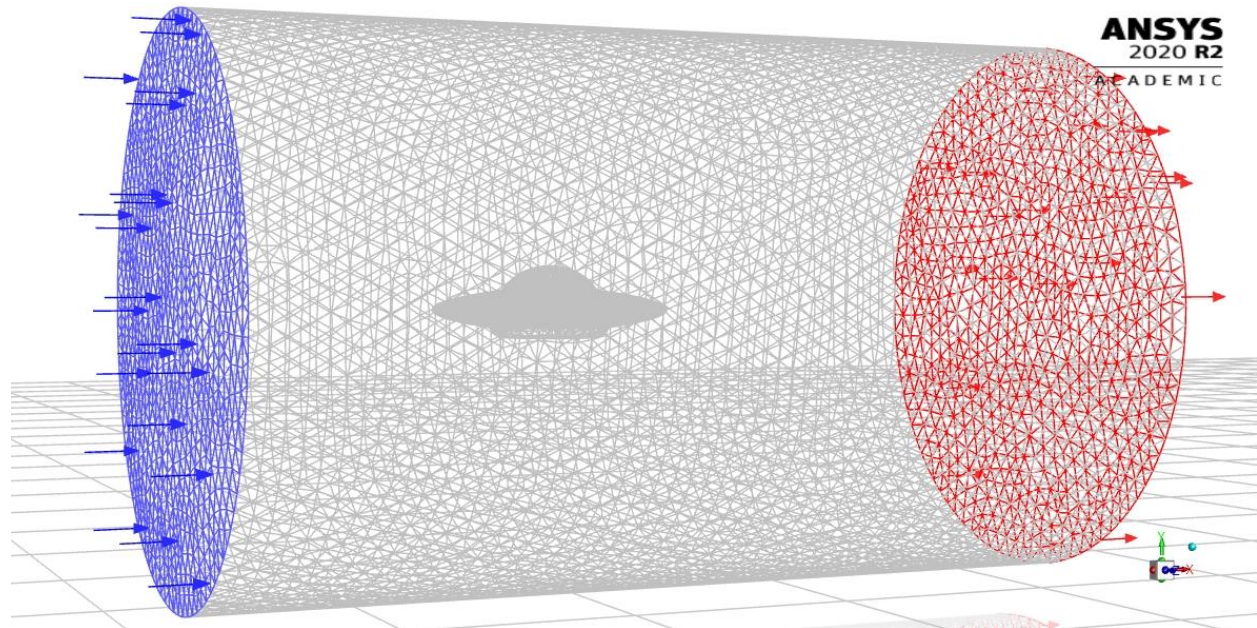


Fig. 10 Setup of the given cylinder geometry as viewed in ANSYS Fluent Solver with linear elements

Fluids used for CFD analysis = Air with constant density of  $0.3 \text{ kg/m}^3$  & viscosity as  $1.44e-05 \text{ kg/m-s}$

Solver type used = Pressure-based Solver

Solver used = Steady State Solver

**MODELS USED: –**

1. Viscous Model is turned on for k-epsilon model with standard settings.
2. Gravity is turned off, since it was not mentioned.

**BOUNDARY CONDITIONS USED: –**

1. Velocity at inlet has *x-velocity*, *y-velocity* and *z-velocity* set at  $75 * \left( 1 - \left( \frac{\sqrt{y^2+z^2}}{R} \right)^2 \right)$  m/s, 0 m/s, and 0 m/s respectively for inlet and *gauge pressure* at 0 Pa for same opening where R is the radius of the cylinder (60 cm) and y and z are the coordinates of the velocity points on inlet face.
2. The other opening was set as pressure outlet with zero gauge pressure.
3. The other sides of the cylindrical tunnel geometry were by default set as stationary wall.

**METHODS USED: –**

1. Pressure Velocity Coupling Scheme => Coupled Scheme.
2. Spatial Discretization Methods used are
  - a. Gradient => Least Squares Cell Based
  - b. Pressure => Second Order
  - c. Momentum => Second Order Upwind
  - d. Turbulent Kinetic Energy => First Order Upwind
  - e. Turbulent Dissipation Rate => First Order Upwind
  - f. Pseudo Transient option is turned on.

**INITIALIZATION USED: –**

1. Initialization Method = Standard Initialization
2. Gauge Pressure = 0 Pa
3. X Velocity = 0.00 m/s
4. Y Velocity = 0.00 m/s
5. Z Velocity = 0.00 m/s
6. Turbulent Kinetic Energy =  $1e-05 \text{ m}^2/\text{s}^2$
7. Turbulent Dissipation Rate =  $1e-06 \text{ m}^2/\text{s}^3$

**MESH SIZE USED: –**

<i>THETA ANGLE</i>	<i>ELEMENT MESH SIZE</i>
$0^\circ$	0.0425 m
$20^\circ$	0.0430 m
$40^\circ$	0.0450 m

**(i) Plot of the mesh along the plane of symmetry for the case with  $\theta = 40^\circ$**

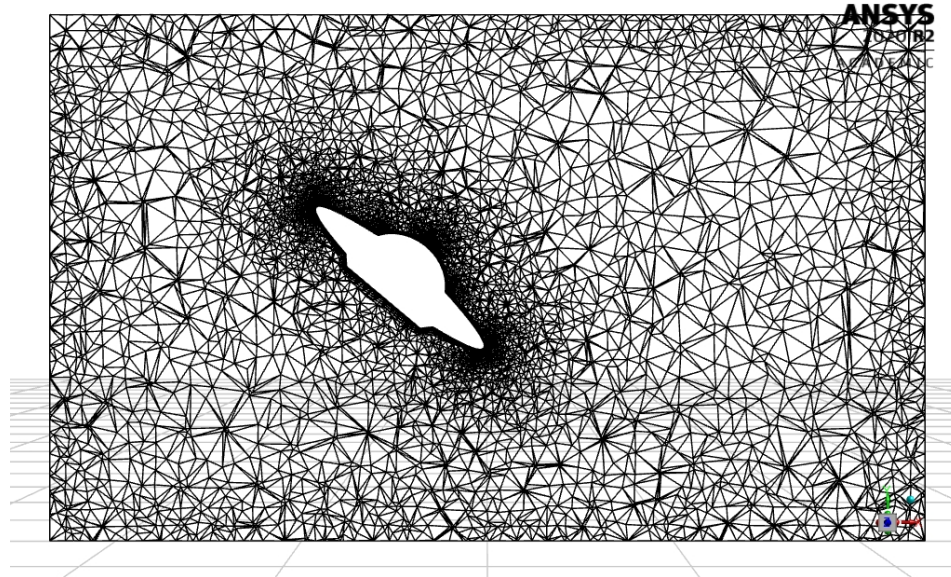


Fig. 11 Line plot of lift force from time = 0 seconds to time = 180 seconds (3 minutes) for given geometry

**(ii) Contour plots of  $x$ -velocity on the plane of symmetry for the three cases with  $\theta = 0^\circ, 20^\circ,$  and  $40^\circ$**

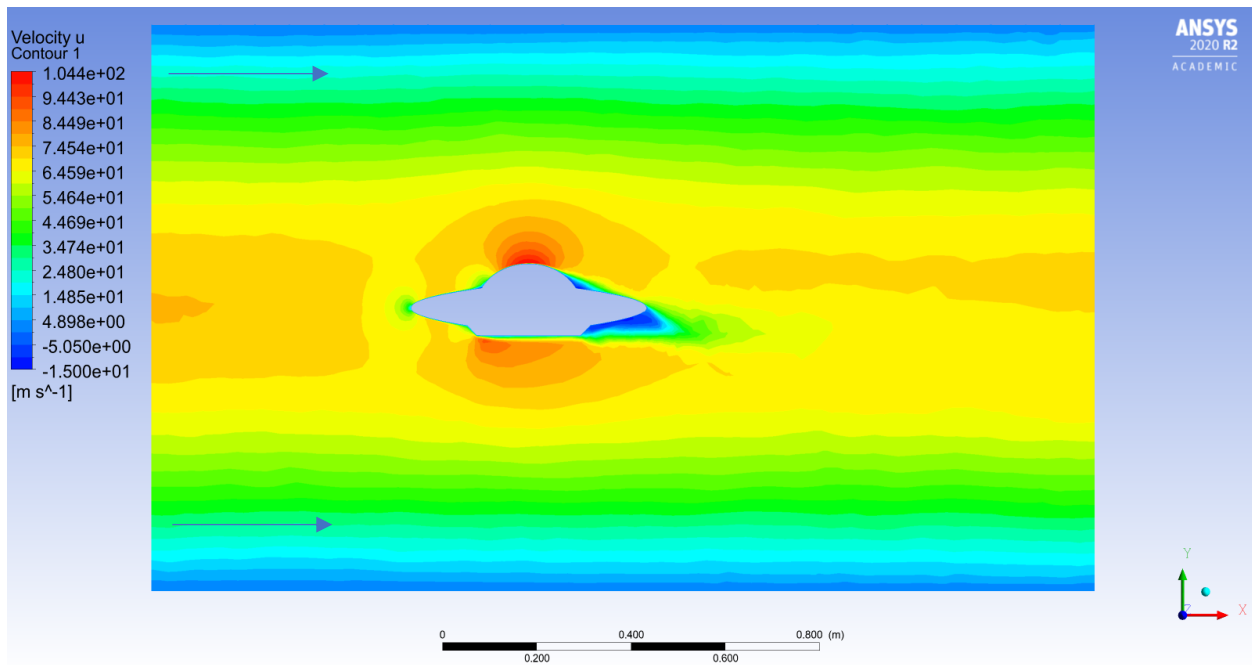


Fig. 12 Contour plot of  $x$ -velocity on the plane of symmetry for the case with  $\theta = 0^\circ$

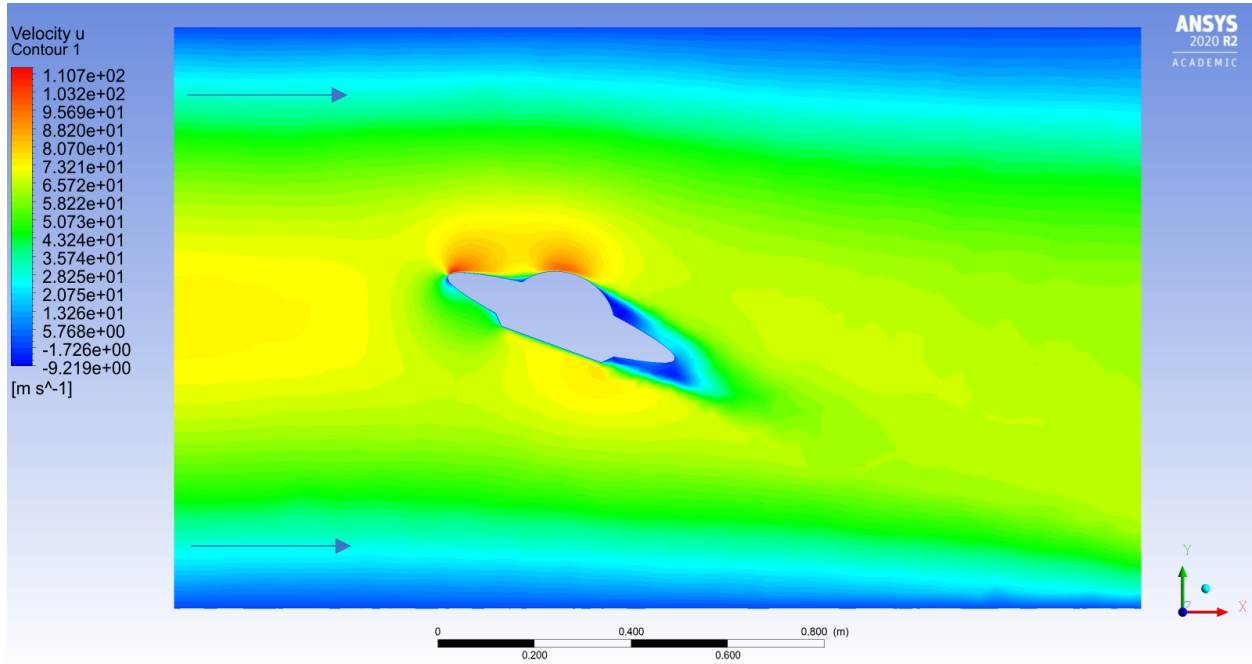


Fig. 13 Contour plot of x-velocity on the plane of symmetry for the case with  $\theta = 20^\circ$

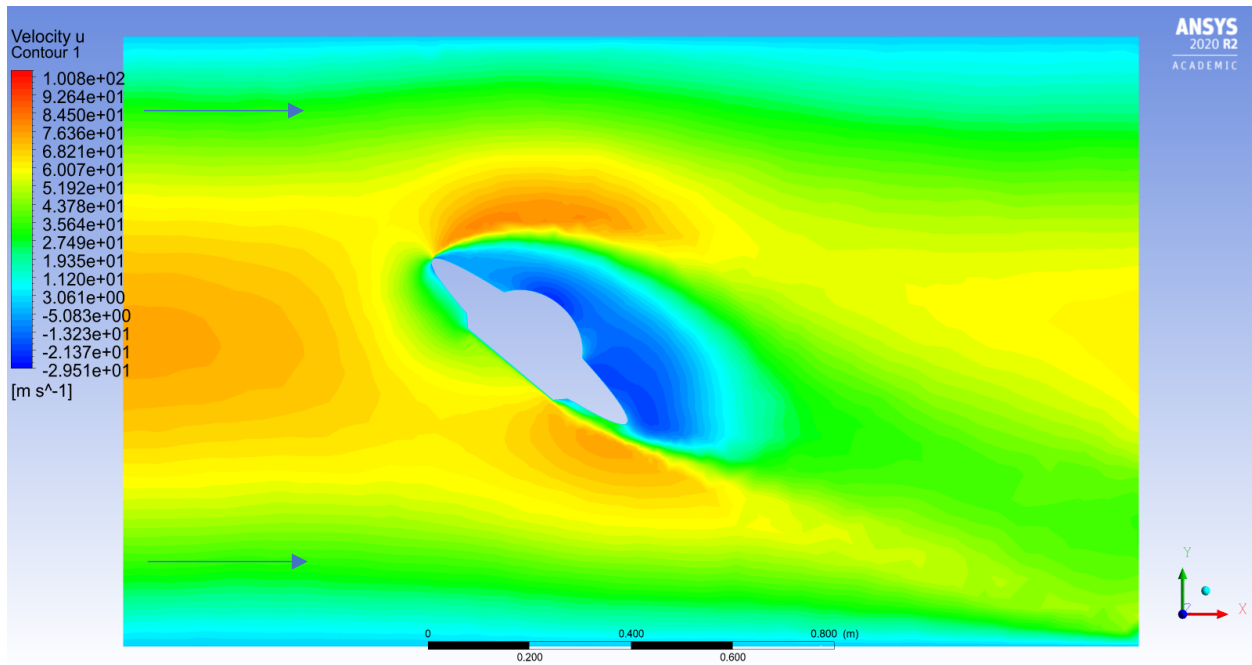


Fig. 14 Contour plot of x-velocity on the plane of symmetry for the case with  $\theta = 40^\circ$

**(iii) Lift force and Drag force for different orientations of flying saucer**

	<i>Lift Force (in Newton)</i>	<i>Drag Force (in Newton)</i>
$\theta = 0^\circ$	5.7357301	5.2038987
$\theta = 20^\circ$	65.880969	19.385001
$\theta = 40^\circ$	61.213903	68.845903

**TASK 3 : – FLOW OVER A PENTAGON-SHAPED BUILDING IN A WIND TUNNEL**

Analysis type of system used in project – **3D** analysis

Following figure displays the dimensions and setup of the system: –

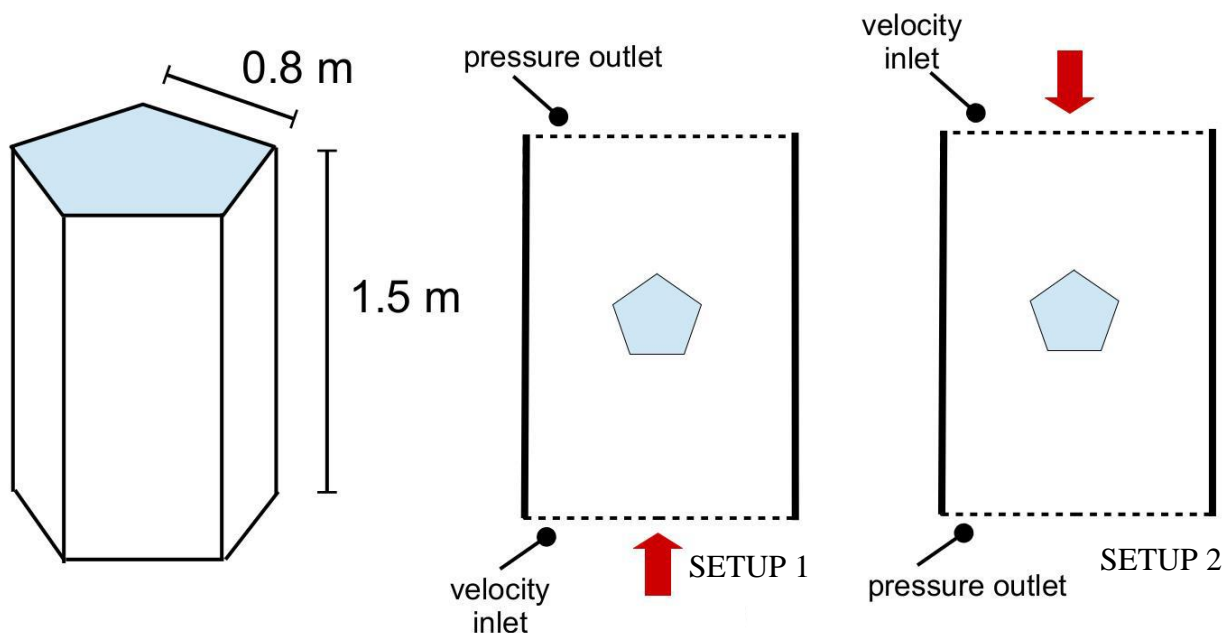


Fig. 15 Dimensions of building and Setup cases for the simulation

Fluids used for CFD analysis = *Air*  
 Solver type used = *Pressure-based Solver*  
 Solver used = *Steady State Solver*  
 Mesh Size used = 0.17 m

**MODELS USED: –**

1. Viscous Model is turned on for k-epsilon model with standard settings.
2. Gravity is turned off, since it was not mentioned.

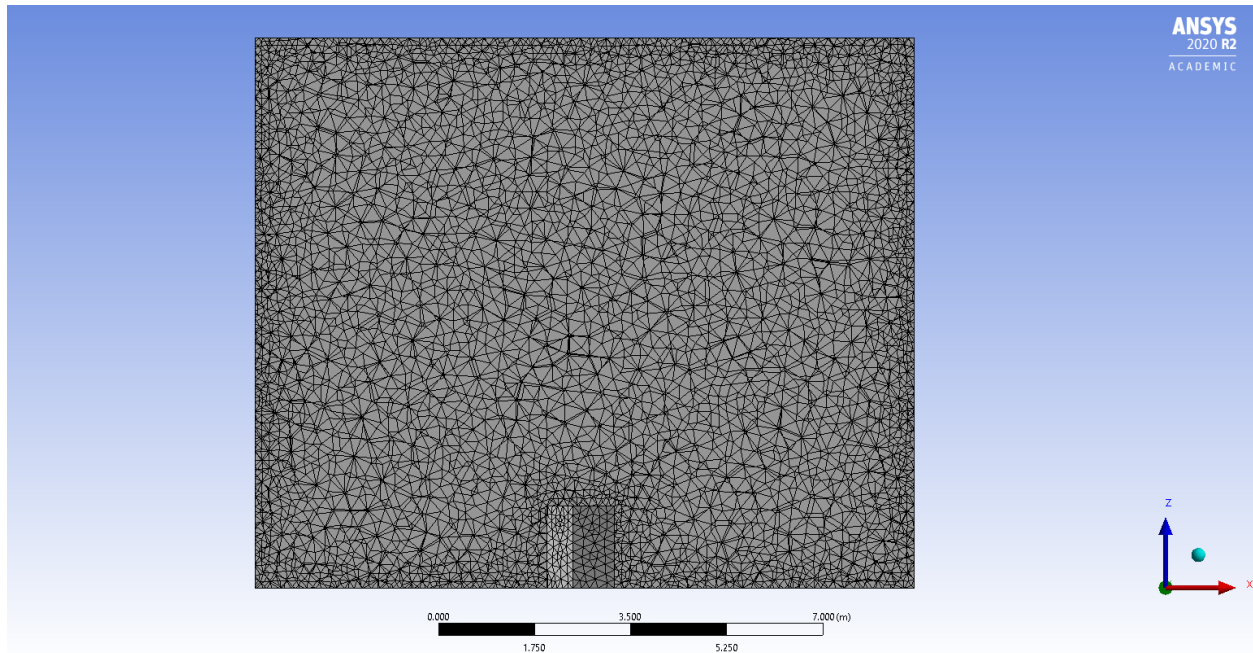


Fig. 16 Sectional view displaying the mesh for setup 2 with vertex facing fluid flow with element size of 0.17m

#### **BOUNDARY CONDITIONS USED: –**

1. Velocity at inlet has magnitude of 30 m/s.
2. The other opening was set as pressure outlet with zero gauge pressure.
3. The other sides of the cuboidal tunnel geometry were by default set as stationary wall.

#### **METHODS USED: –**

1. Pressure Velocity Coupling Scheme => Coupled Scheme.
2. Spatial Discretization Methods used are
  - a. Gradient => Least Squares Cell Based
  - b. Pressure => Second Order
  - c. Momentum => Second Order Upwind
  - d. Turbulent Kinetic Energy => First Order Upwind
  - e. Turbulent Dissipation Rate => First Order Upwind
  - f. Pseudo Transient option is turned on.

#### **INITIALIZATION USED: –**

1. Initialization Method = Standard Initialization
2. Gauge Pressure = 0 Pa
3. X Velocity = 0.00 m/s
4. Y Velocity = 0.00 m/s
5. Z Velocity = 0.00 m/s
6. Turbulent Kinetic Energy =  $1e-05 \text{ m}^2/\text{s}^2$
7. Turbulent Dissipation Rate =  $1e-06 \text{ m}^2/\text{s}^3$

**SETUP 1 – FLAT FACE FACING THE FLOW OF FLUID**

**(i) Contour plots of static pressure and x-velocity on the horizontal plane with  $z = 0.75$  m.**

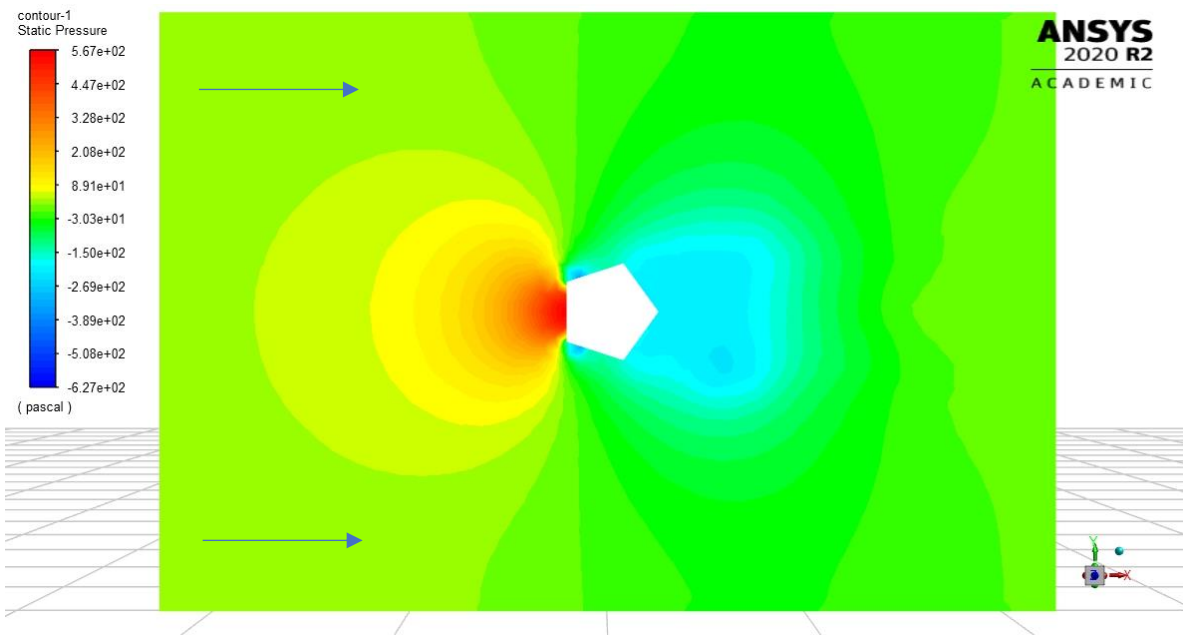


Fig. 17 Contour plot of static pressure on the horizontal plane for the case with flat face facing flow of fluid

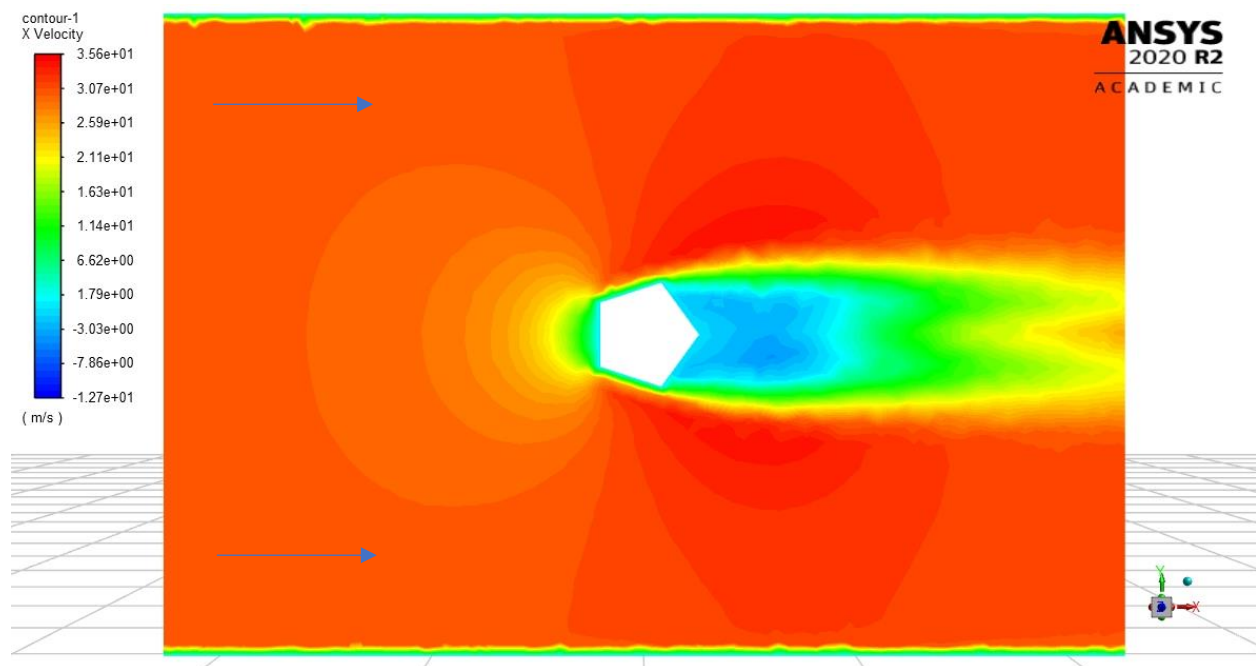


Fig. 18 Contour plot of X velocity on the horizontal plane for the case with flat face facing flow of fluid

**SETUP 2 – VERTICE FACING THE FLOW OF FLUID**

**(i) Contour plots of static pressure and x-velocity on the horizontal plane with  $z = 0.75$  m.**

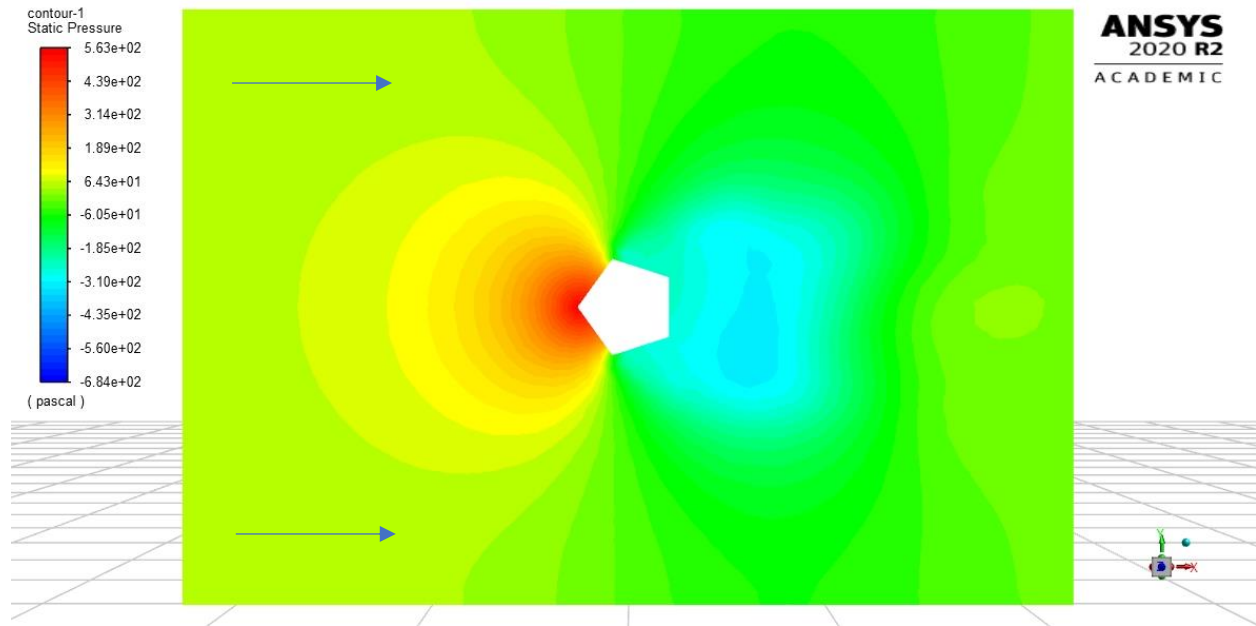


Fig. 19 Contour plot of static pressure on the horizontal plane for the case with either vertex facing flow of fluid

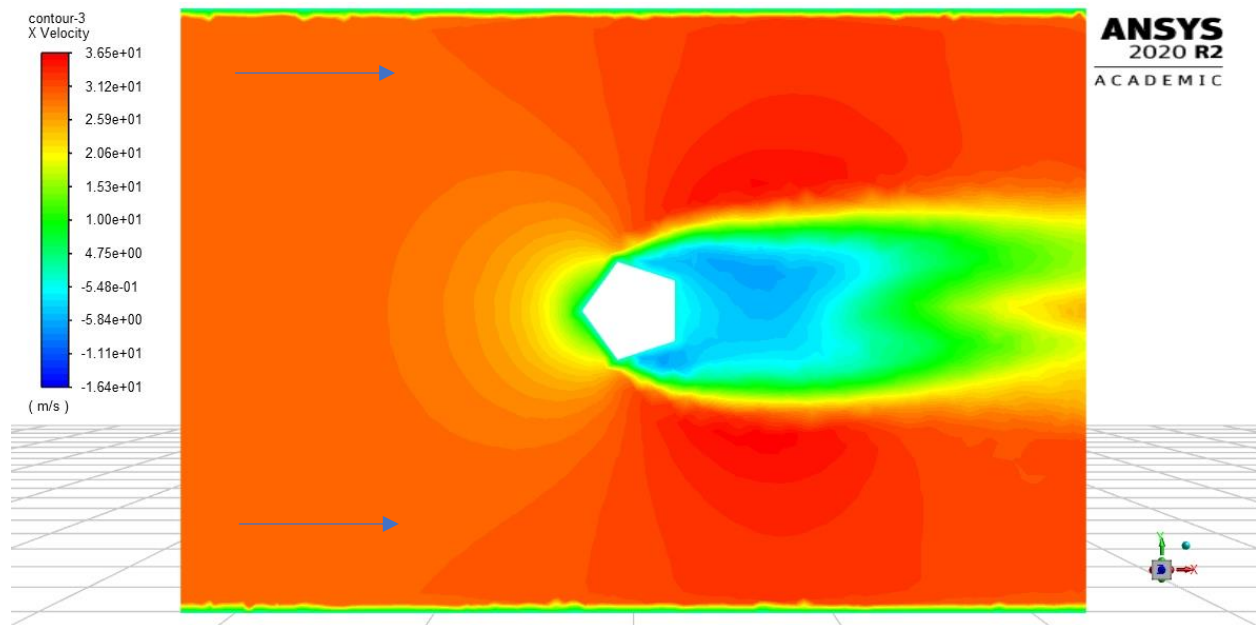


Fig. 20 Contour plot of X velocity on the horizontal plane for the case with either vertex facing flow of fluid



**(ii) Drag force estimation along with its contributing components for both setups**

	<i>Total Drag (N)</i>	<i>Pressure Term of Drag (N)</i>	<i>Viscous Term of Drag (N)</i>
<i>Setup - 1</i>	718.78041	715.78815	2.9922595
<i>Setup - 2</i>	1125.5706	1124.9666	0.60402393

Since we know that viscous drag is related to attached flow (friction between body and the moving fluid) and that it increases with delay in separation, it can be observed here in our simulation that setup 1 has three faces of the pentagon which are directly imparted by incoming fluid, hence it makes sense to have more viscous force in that case (i.e. flat face facing fluid) compared to the other one (i.e. vertex facing fluid) which only has only two faces of the pentagon directly facing the incoming fluid. This result is also supported by the velocity contours in the direction of the flow (in this case X-velocity, since geometry was designed with above specific orientation, hence x-velocity is parallel to the flow of fluid). The velocity contours portray the clear image of the fluid striking the walls of fluid as mentioned above.

Now for the pressure drag, since the pressure drag is rather more related with the formation of the wake and cross-sectional area of the body, it makes sense that the pressure drag is more for setup 2 compared to setup 1. Also, the cross-section area presented to fluid to attack directly in setup 2 is less compared to that in setup 1, which should result in more pressure force compared to its counterpart and from the static pressure contour plots is evident that for flat face setup, the fluid after striking the surface of the building doesn't have enough guidance from the shape of the building to move away from the building and as the fresh fluid keeps on striking the same part (i.e. the flat face of building), it creates a sense of high charge zone, which leads to more compression of the fluid and hence, an increase in static pressure in front of the fluid face.

Moreover, about the pressure drag, the more flow is attached to the body, the less is the drag generated due to the pressure difference due to the formation of the wake generated by detachment of the flow. Hence, since the flow is separated early in setup-2 compared to setup-1, it makes sense that the pressure drag is more in setup 2 compared to its counterpart.

The x-velocity contours illustrate the above mentioned fact about the pressure difference. As we know that the pressure difference is higher in setup-2, this leads to more rapid change in flow, which leads to increase in x-velocity and this can be seen through the contours as setup-2 has higher value of x-velocity in the direction of the flow (positive value of x-velocity) compared to setup-1.