

## MAE 494/598 Applied CFD, Fall 2019, Project 4 – External flow (23 points)

**Revised.** Changes made to Task 1 and 4 are highlighted.

Hard copy of report is due 1:30 PM on the due date. Please follow the rules for collaboration as described in the cover page of the document for Project 1. **A statement on collaboration is mandatory for all.** All tasks in this project are for both MAE598 and MAE494, **except Task 4 which is for MAE598 only.**

### Task 1

(a) Consider the system of a 2-D flow passing a cylinder as illustrated in Fig. 1. The 2-D cylinder, essentially a circular disk, is centered at  $(x, y) = (0, 0)$  with a radius of 5 cm. To perform the simulation using Ansys-Fluent, the cylinder is placed in a 2-D computational domain with its dimension (in cm) given in Fig 1. The left boundary is a *velocity inlet* with a constant  $x$ -velocity of 3.5 cm/s. (The  $x$ -direction is as indicated in Fig. 1.) The right boundary is a *pressure outlet* with zero gauge pressure. Top and bottom are walls. The system is filled with *water* with constant density and viscosity (using the default values from Fluent database).

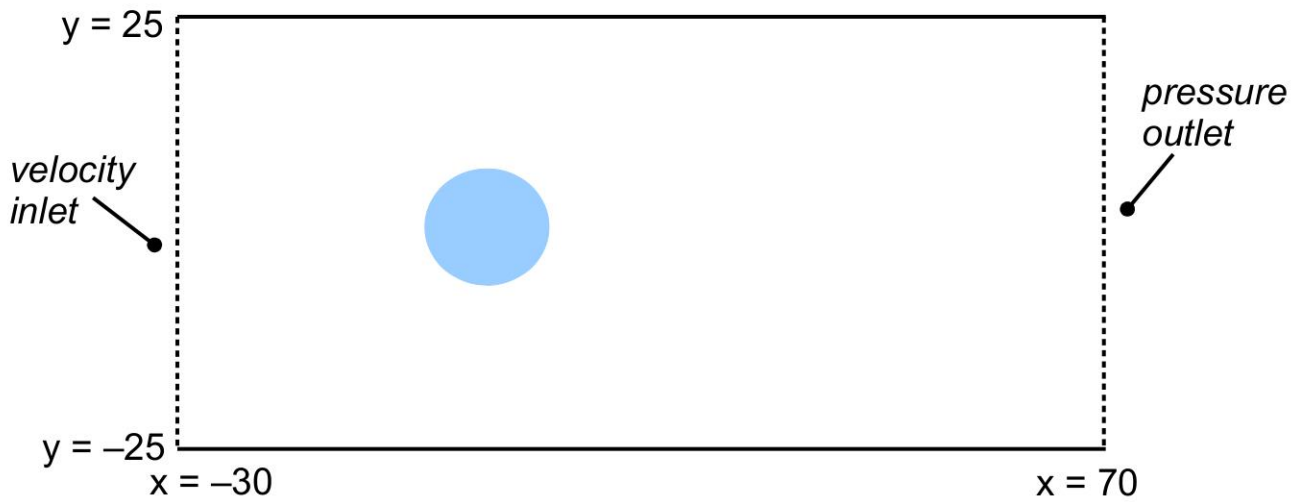
At  $t = 0$ , the system is initialized with  $x$ -velocity = 3.5 cm/s,  $y$ -velocity = 0, and gauge pressure = 0 within the domain. This task uses *laminar* model to run a transient simulation to at least  $t = 5$  minutes. The deliverables are

- (i) An estimate of Reynolds number for this system
- (ii) Contour plots of *static pressure*, *y-velocity*, and *stream function* at the end of simulation
- (iii) A line plot of the **lift force** as a function of time over the last 3 minutes of the simulation. For example, if the run is performed to  $t = 5$  min, the plot will be from  $t = 2$  min to  $t = 5$  min. Using this plot, estimate the amplitude and period of oscillation of the **lift force**. Amplitude can be defined as (maximum – minimum)/2. If the lift coefficient is not oscillatory but steady or chaotic in time, please state so and there would be no need to estimate the amplitude and period.

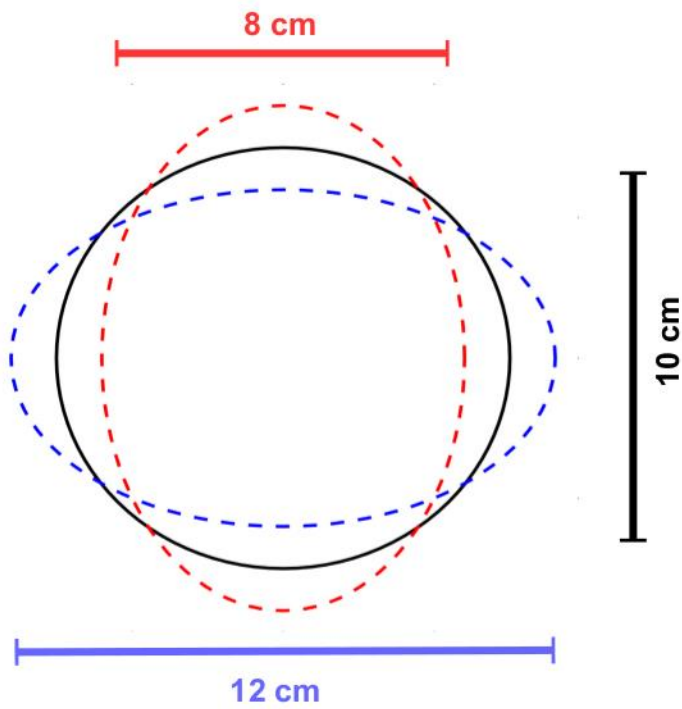
(b) Use the same setting as (a) to perform two additional simulations with elliptical instead of circular cylinders. **Run 1** uses an ellipse elongated in  $y$ -direction, with the lengths of its major and minor axes 12 cm and 8 cm (the major axis is parallel to  $y$ -axis), as illustrated by the red dashed curve in Fig. 2. **Run 2** uses an ellipse elongated in  $x$ -direction, with the lengths of its major and minor axes 12 cm and 8 cm, as illustrated by the blue dashed curve in Fig. 2. The deliverables are

- (i) For each run, a line plot of **lift force** as a function of time over the last 3 minutes of the simulation and estimates of the amplitude and period of oscillation of the lift coefficient. Please also provide a summary and comparison of the (amplitude, period) of oscillation for the 3 cases in (a) and (b).

Please see **Addendum to Project 4** for further explanations on the revision, and for other options to complete related tasks in Task 1 and 4.



**Fig. 1** The geometry of the 2D cylinder and computational domain for Task 1a. (Not drawn to scale.) All numbers are in cm.



**Fig. 2** The shape of the elliptical object used in Task 1b. Black: circle from Task 1a. Red: ellipse elongated in  $y$ -direction; Blue: ellipse elongated in  $x$ -direction.

## Task 2

In this task, we test the aerodynamics of a 3-D “flying saucer” in a cylindrical virtual wind tunnel. The 3-D object is shown in Fig. 3. It can be created by setting unit to “cm”, reading the profile data in the file, "flyingsaucer2DH.txt", to create a curve (as shown in Fig. 4), then revolving that curve around the  $y$ -axis to build the full 3-D surface. The setup of the virtual wind tunnel is as illustrated in Fig. 5. It has two openings which will be set up as *velocity inlet* and *pressure outlet*, as indicated in Fig. 5. All other surfaces are walls. The detailed dimension (in cm) of the system is given in Fig. 6. (For simplicity, we show only the cross section along the plane of symmetry. Remember that the system is fully 3-D.) The radius and length of the cylindrical tube are 60 cm and 200 cm, respectively. The precise  $x$  and  $y$  coordinates of the corner points are given in Fig. 6. Note that the flying saucer is placed closer to the inlet than the outlet. A tilt angle,  $\theta$ , is also defined in Fig. 6. It is the angle of rotation of the 3-D object with respect to the  $z$ -axis. The configuration shown in Fig. 5 corresponds to the case with  $\theta = 0^\circ$ .

The system is filled with air with a constant density of  $0.4 \text{ kg/m}^3$  and constant viscosity of  $1.45 \times 10^{-5} \text{ N s m}^{-2}$ . [These values represent the condition of the atmosphere at approximately the cruising altitude ( $\sim 10 \text{ km}$ ) for commercial airliners.] For all simulations in this task, the inlet velocity is set to a parabolic profile for the  $x$ -velocity given by

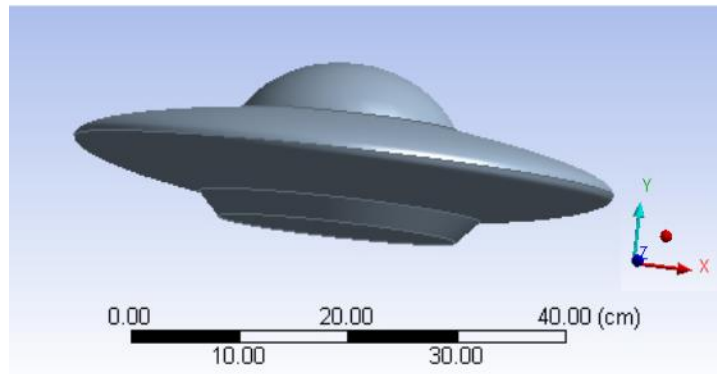
$$u = 50 \left( 1 - \left( \frac{r}{R} \right)^2 \right) \text{ m s}^{-1} ,$$

where  $r = \sqrt{y^2 + z^2}$  is the radial distance from the center of the circular opening of the inlet, and  $R = 60 \text{ cm}$  is the radius of that circular opening. With this setup,  $u$  reaches the maximum of  $50 \text{ m/s}$  at  $r = 0$ , and minimum of  $0$  at the wall where  $r = R$ .

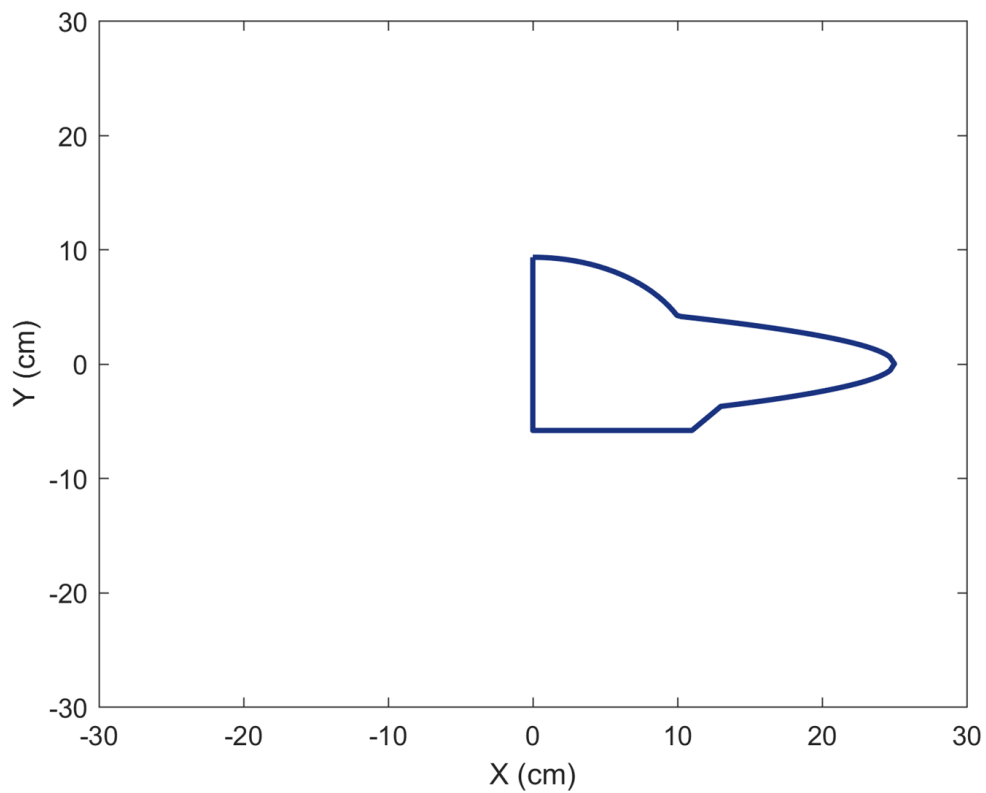
All simulations will use turbulence k-epsilon model, and will seek *steady solution*. Perform four runs with the tilt angle set to  $\theta = 0^\circ, 15^\circ, 30^\circ$  and  $45^\circ$ . The deliverables are:

- (i) Contour plots of  $x$ -velocity on the plane of symmetry for the two cases with  $\theta = 0^\circ$  and  $\theta = 45^\circ$ .
- (ii) A line plot of the *lift force* and *drag force* (that the fluid exerts on the flying saucer) as a function of the tilt angle. This is a simple plot with two curves, each represents just four data points. (For this plot, please use the total force, i.e., the sum of the contributions from pressure and viscous terms. No need to split the two terms.) For this system, lift and drag are the forces in the  $y$ -direction and  $x$ -direction, respectively. See Fig. 6 for the definitions of those directions.

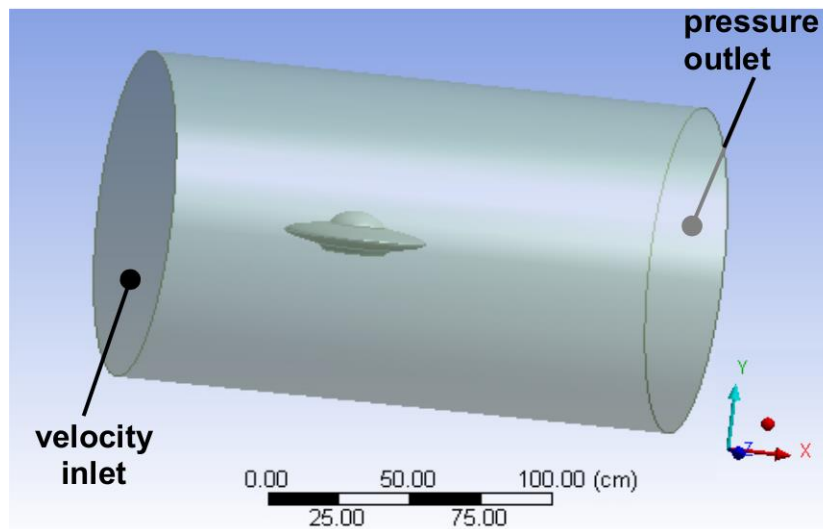
[For each run, please perform a sufficient number of iterations to ensure that lift and drag no longer trend upward or downward. This can be checked by monitoring the lift or drag as a function of the number of iterations. If the value of lift or drag continues to fluctuate with the number of iterations (which could happen if the actual solution is oscillatory or chaotic in time), take the average over a range of iterations. Alternatively, in that case we allow the option of performing a transient simulation and take the time average of lift or drag.]



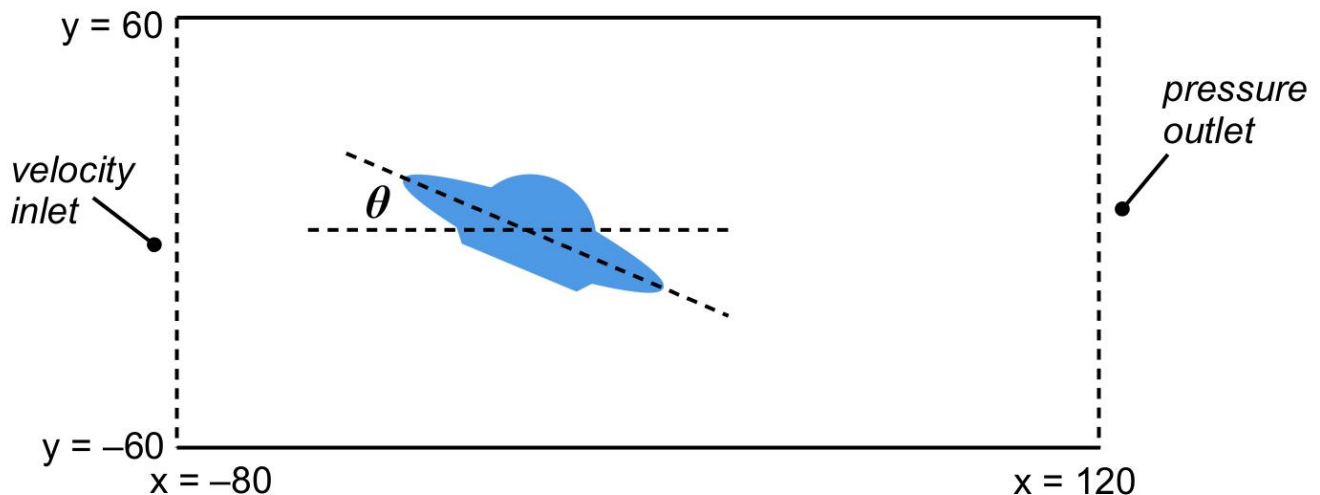
**Fig. 3** The geometry of the “flying saucer”.



**Fig. 4** The profile of the "half 2D flying saucer". The detailed data is in `flyingsaucer2DH.txt`.



**Fig. 5** The setup of the 3-D virtual wind tunnel.



**Fig. 6** The dimension (in cm) of the cylindrical virtual wind tunnel along its plane of symmetry, and the definition of the tile angle.

### Task 3

In this task, we simulate a flow over a pentagon-shaped building in a virtual wind tunnel. The building is shown in Fig. 7a. It is 2 m tall and each side of the pentagon is 1 m long. The building is placed at the center of the bottom plate (i.e., the “ground”) of the tunnel, as shown in Fig. 8. The dimension of the computational domain is given in Fig. 8. The left and right openings are set as *velocity inlet* and *pressure outlet*. All other surfaces are walls. The tunnel is filled with air with constant density and viscosity (using the default values from Fluent database). All simulations in this task will use turbulence k-epsilon model and seek *steady solution*. The inlet velocity is set to a uniform 50 m/s.

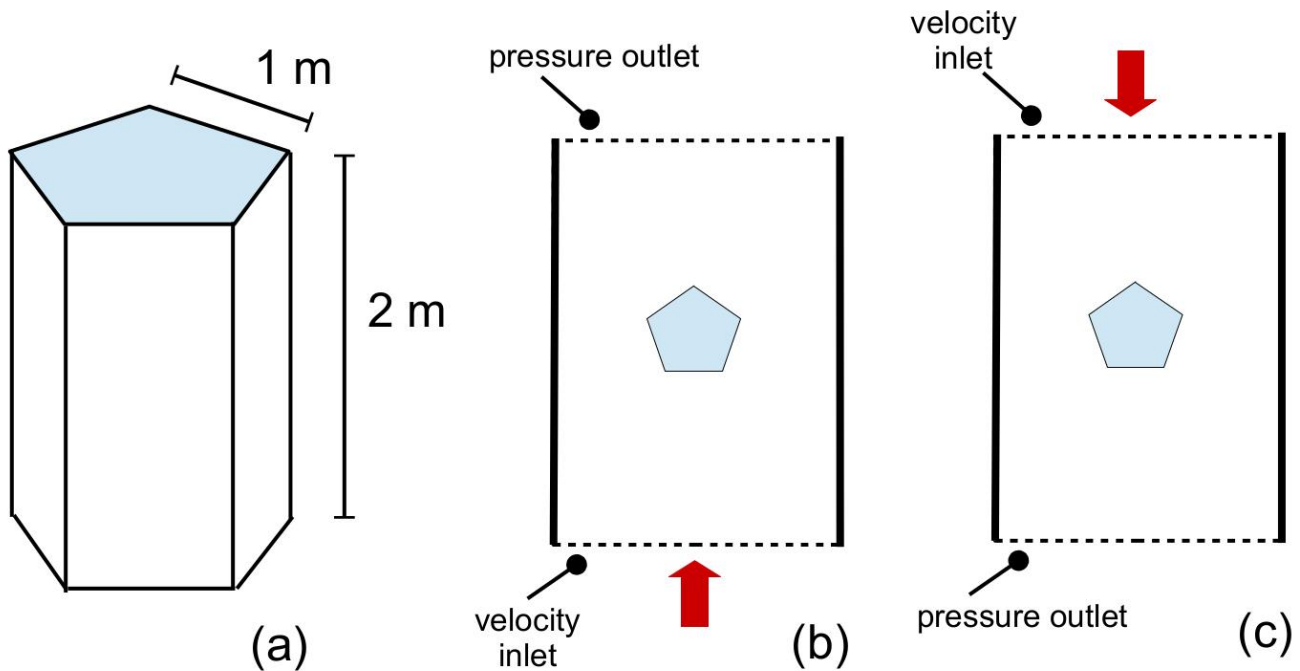
(a) Perform the simulation with the setup shown in Fig. 7b. Namely, the wind coming from the inlet is attacking a flat edge of the pentagon. The deliverables are:

(i) Contour plots of *static pressure* and *y-velocity* on the horizontal plane with  $z = 1$  m. (This is the plane that cuts through the middle of the building. The  $y$ -direction runs parallel to the virtual wind tunnel; See Fig. 8 for the definition.)

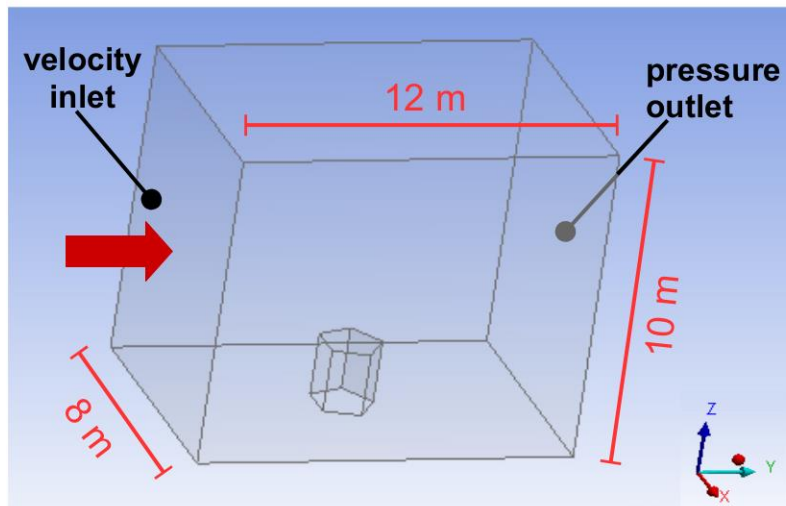
(ii) A contour plot of the *y-velocity* on the plane of symmetry. (This is the vertical plane that runs through the center of the virtual wind tunnel.)

(iii) The values of total drag force (that the fluid exerts on the building) and the individual contributions to the drag force from the pressure and viscous terms.

(b) Perform an additional simulation using the setup in Fig. 7c. Namely, the wind is blowing towards one of the vertices of the pentagon. Produce the same deliverables as in Part (a). In addition, based on the results from (a) and (b), briefly discuss how the structures of pressure and velocity in the two sets of simulations affect the drag forces in those two cases.



**Fig. 7** The geometry of the building is shown in (a). The setups of the two runs for Task 3a and 3b are shown in (b) and (c) as the top view of the system.



**Fig. 8** The setup of the 3-D virtual wind tunnel for Task 3.

**Task 4** (for MAE 598 only)

In the 3rd video shown in Lecture 24 (related to the background for Task 1), it was mentioned that “some asymmetric forms of cylinder are unstable”, in that a flow passing the cylinder induces a large-amplitude response so as to damage the structure. Using the setup in Task 1, try to run a case with a cylinder that has an asymmetric shape but otherwise a cross-sectional area comparable to that of the “circular cylinder” in Task 1a. The goal is to examine whether a large-amplitude response in the lift can be produced by the numerical simulation with a certain choice of an asymmetrically-shaped cylinder. The deliverables are (i) A plot of the geometry of the cylinder in the fashion of Fig. 1. (ii) A line plot of **lift force** vs. time, similar to deliverable (iii) of Task 1a. (For this deliverable, the plot needs not necessarily be over the last 3 minutes of the run. Please feel free to arrange the plot as long as it clearly illustrates the temporal behavior and the amplitude of the lift coefficient.)

Note: The catastrophic collapse of the solid structure as shown in the video is due in part to resonance that involves the vibration of the structure. Since in the numerical simulation the cylinder is fixed in space, the behavior of the lift force might not be as dramatic as in the video. If after some trials you do not find a dramatic amplification in the lift associated to an asymmetric cylinder, please submit whatever you find.