

MAE 560/460 Applied CFD, Fall 2020, Project 3 – External flow (25 points)

Deadline at Canvas is (Arizona time) 11:59 PM, Monday, November 30th. Please follow the rules for collaboration as described in the cover page of the document for Project 1. A statement on collaboration is required for all submissions. Task 1 and 2 are for both MAE560 and MAE460. Task 3 is for MAE560 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 diameter of 8 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 x -velocity set to 4 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 = 4 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 $t = 3$ 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 $t = 3$ min.
- (iii) A line plot of the *lift force* (that fluid exerts on the cylinder) as a function of time from $t = 0$ to $t = 3$ min. Using this plot, estimate the amplitude and period of oscillation of the lift force. (The estimate should use only the later portion of the simulation, excluding the initial “spin up” when the oscillation is not yet fully established.) Amplitude can be defined as $(\text{maximum} - \text{minimum})/2$. If the lift force 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 10 cm and 6 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 10 cm and 6 cm, as illustrated by the blue dashed curve in Fig. 2. The deliverables are

- (iv) For each run, a line plot of *lift force* as a function of time from $t = 0$ to $t = 3$ min. From the plot, estimates of the amplitude and period of oscillation of the lift force. Summarize the amplitude and period of the lift force from the 3 runs in Task 1 in the following table:

	Amplitude (in Newton)	Period (in second)
Circular cylinder	?	?
Elliptical cylinder, Run 1	?	?
Elliptical cylinder, Run 2	?	?

Otherwise, no need to make any contour plots for Part (b).

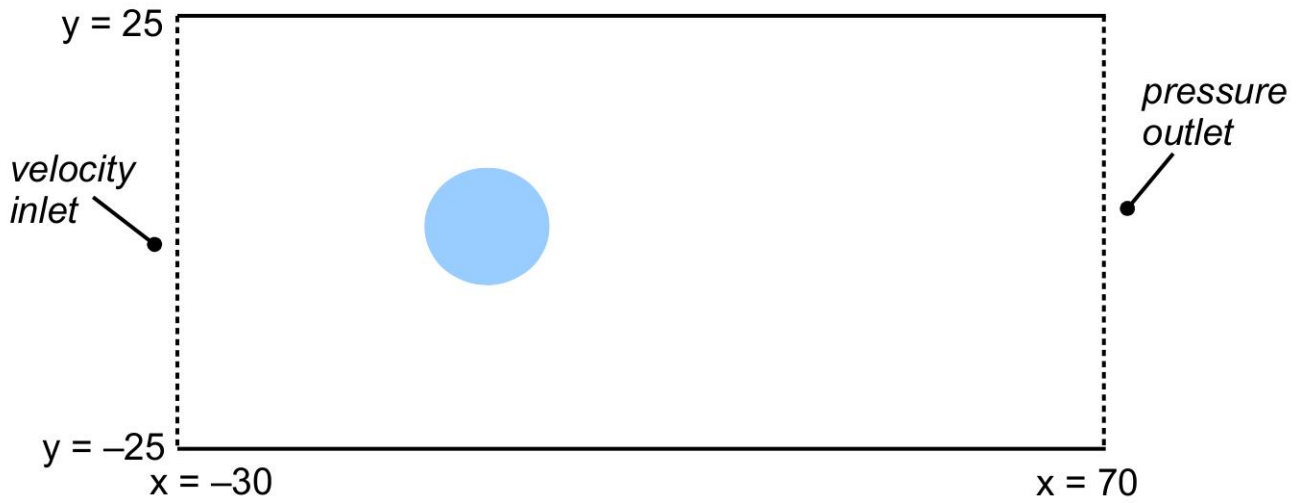


Fig. 1 The geometry of the 2D cylinder and computational domain for Task 1a. (Not drawn to scale.) All numbers are in cm.

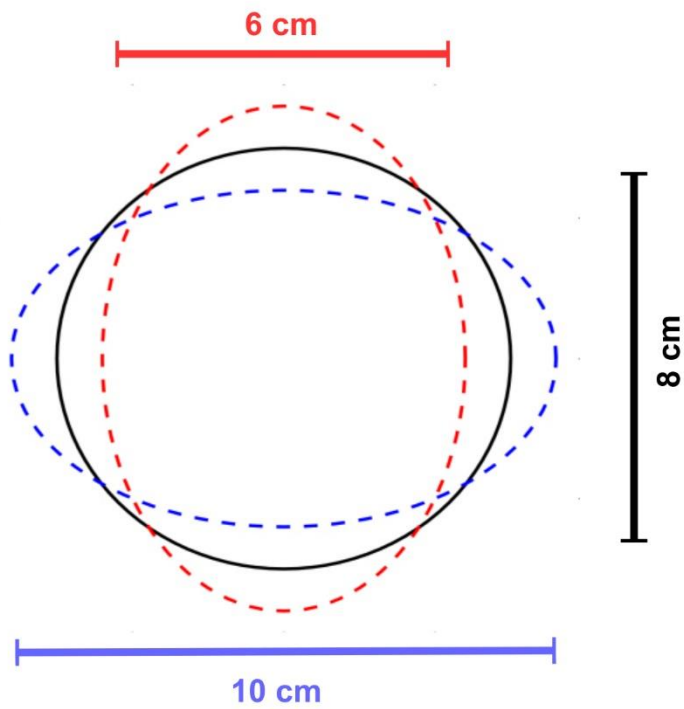


Fig. 2 The shape of the elliptical object used in Task 1b. Black: circle from Task 1a. Red: ellipse elongated in y -direction; Blue: ellipse elongated in x -direction.

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, θ , 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 0.3 kg/m^3 and constant viscosity of $1.44 \times 10^{-5} \text{ N s m}^{-2}$. [These values represent the condition of the atmosphere at approximately the cruising altitude ($\sim 12 \text{ km}$) for commercial airliners.] For all simulations in this task, the inlet velocity is set to a parabolic profile for the x -velocity given by

$$u = 75 \left(1 - \left(\frac{r}{R} \right)^2 \right) \text{ m s}^{-1} ,$$

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 75 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 three runs with the tilt angle set to $\theta = 0^\circ$, 20° , and 40° . The deliverables are:

- (i) A plot of the mesh along the plane of symmetry for the case with $\theta = 40^\circ$.
- (ii) Contour plots of x -velocity on the plane of symmetry for the three cases with $\theta = 0^\circ$, 20° , and 40° .
- (iii) The values of *lift force* and *drag force* (that fluid exerts on the flying saucer) as a function of the tilt angle. (For this deliverable, please present only the total force, i.e., the sum of the contributions from pressure and viscous terms. No need to further separate the two terms.) This should be summarized in the following table:

	Lift force (in Newton)	Drag force (in Newton)
$\theta = 0^\circ$?	?
$\theta = 20^\circ$?	?
$\theta = 40^\circ$?	?

[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 done by monitoring lift or drag as a function of the number of iterations. If lift or drag continues to fluctuate with 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.]

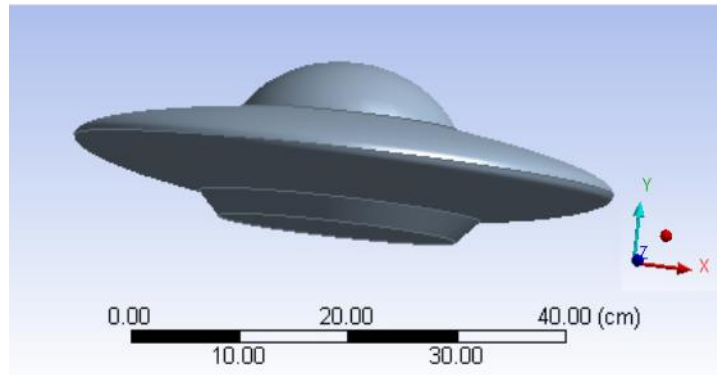


Fig. 3 The geometry of the “flying saucer”.

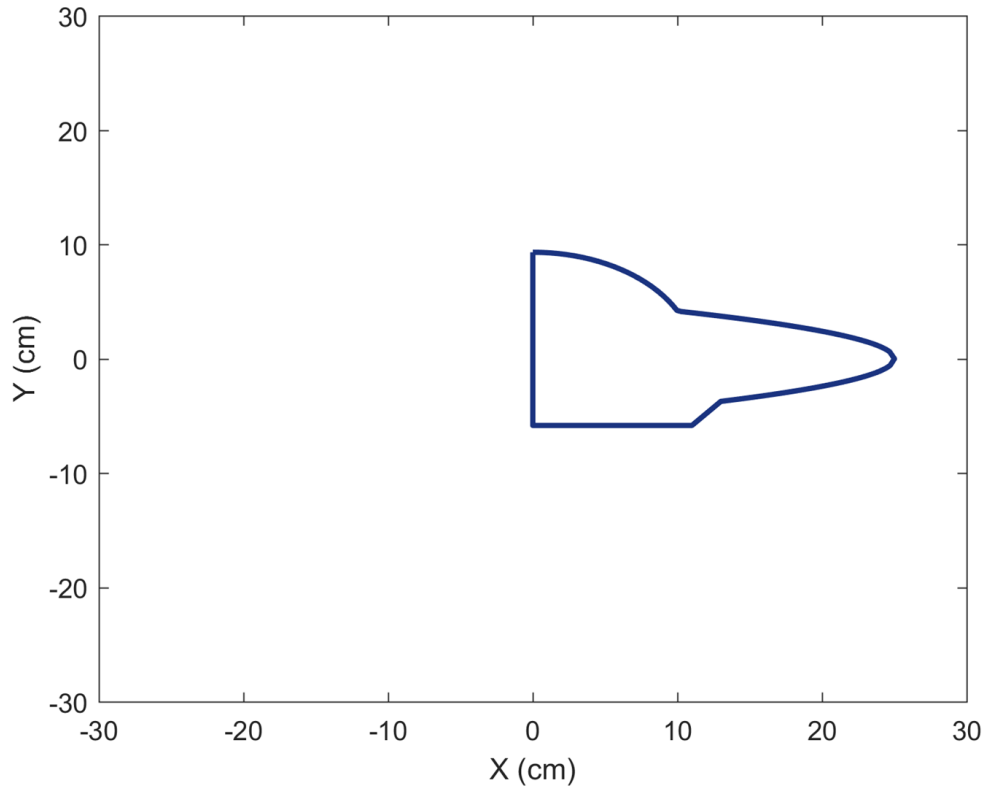


Fig. 4 The profile of the "half 2D flying saucer". The detailed data is in `flyingsaucer2DH.txt`.

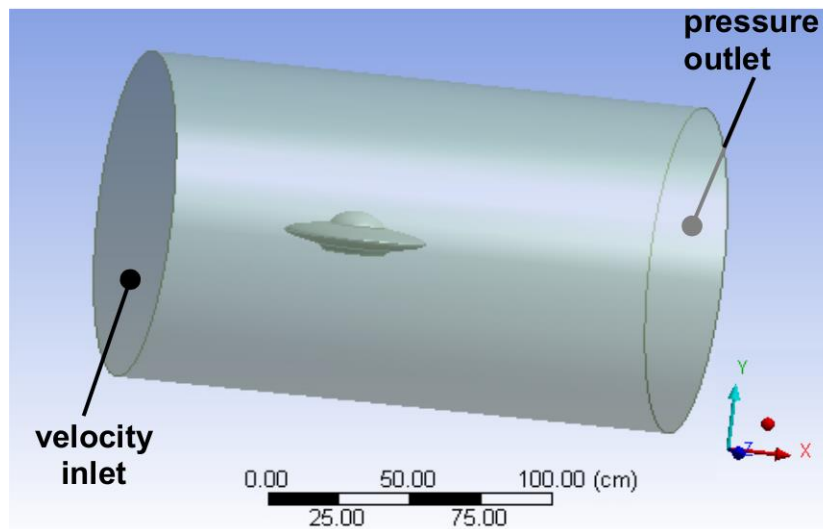


Fig. 5 The setup of the 3-D virtual wind tunnel.

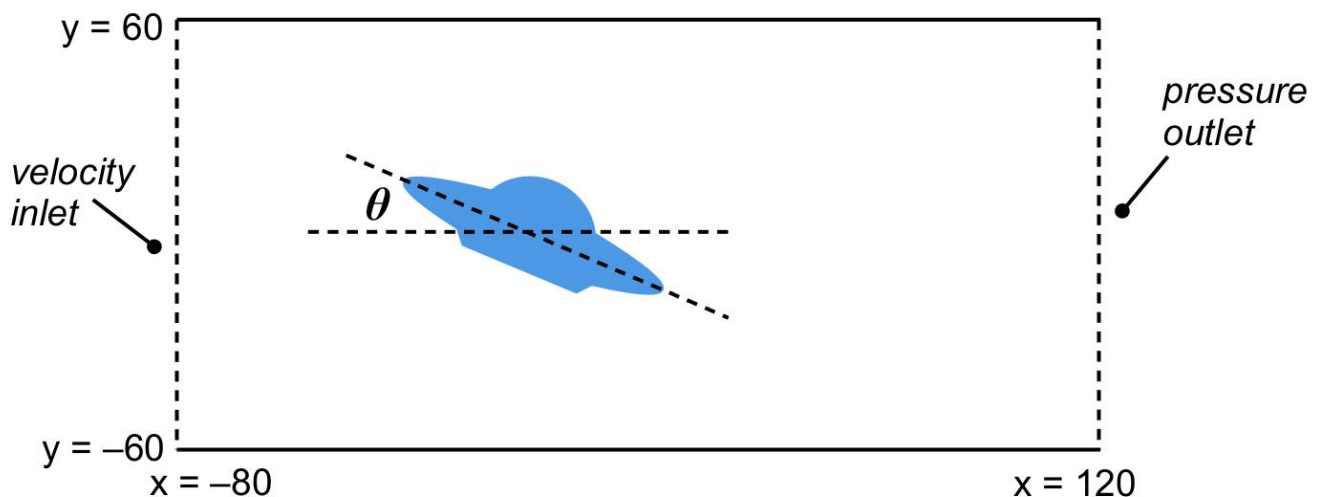


Fig. 6 The dimension (in cm) of the cylindrical virtual wind tunnel along its plane of symmetry, and the definition of the tile angle.

Task 3 (MAE560 only)

In this task, we simulate a 3-D flow over a pentagon-shaped building in a virtual wind tunnel. The building is shown in Fig. 7a. It is 1.5 m tall and each side of the pentagon is 0.8 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 30 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 $z = 0.75$ m. (This is the horizontal plane that cuts through the middle of the building. The *y*-direction runs parallel to the direction of inlet velocity; See Fig. 8 for the definition.)

(ii) The values of total drag force (that fluid exerts on the building) and the individual contributions to the drag force from the pressure and viscous terms. Note that, by the definition given in Fig. 8, the drag force is the force in *y*-direction.

(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. For the presentation of drag force, we recommend filling the following table:

	Total drag (N)	Pressure term of drag (N)	Viscous term of drag (N)
Case (a) (based on Fig. 7b)	?	?	?
Case (b) (based on Fig. 7c)	?	?	?

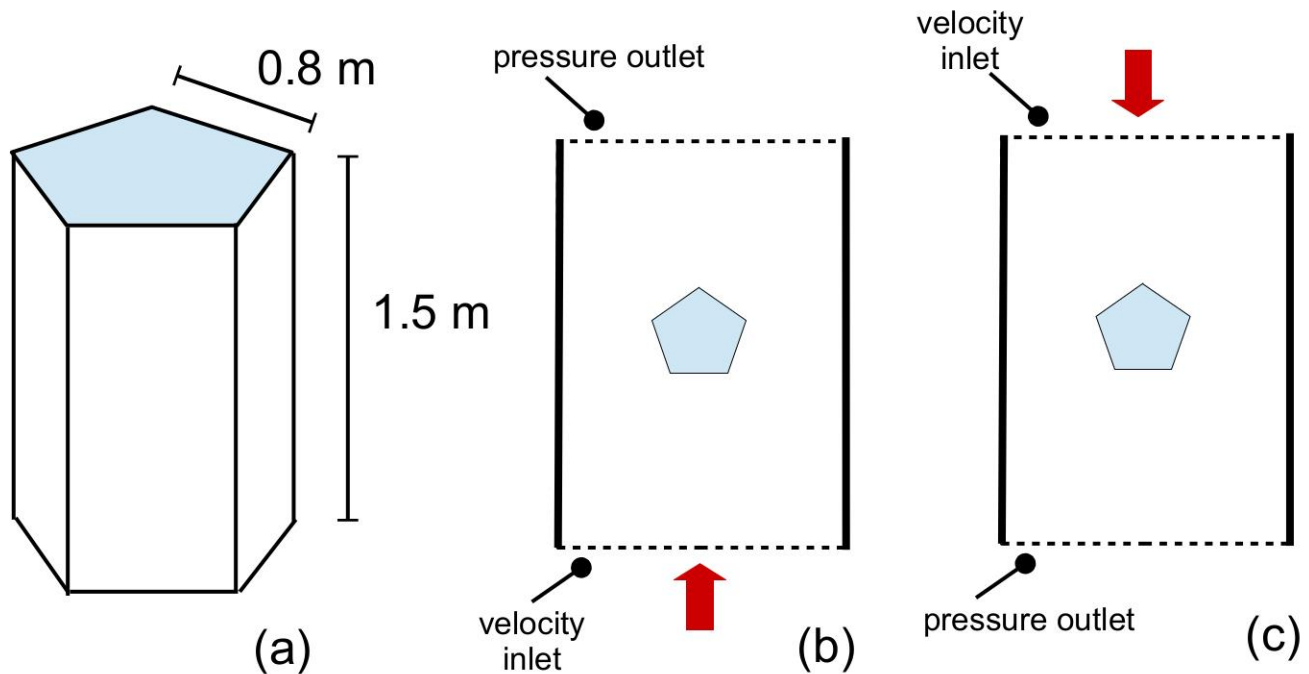


Fig. 7 The geometry of the building is shown in (a). The setups of the two runs for Task 3a and 3b are shown in (b) and (c) as the top view of the system.

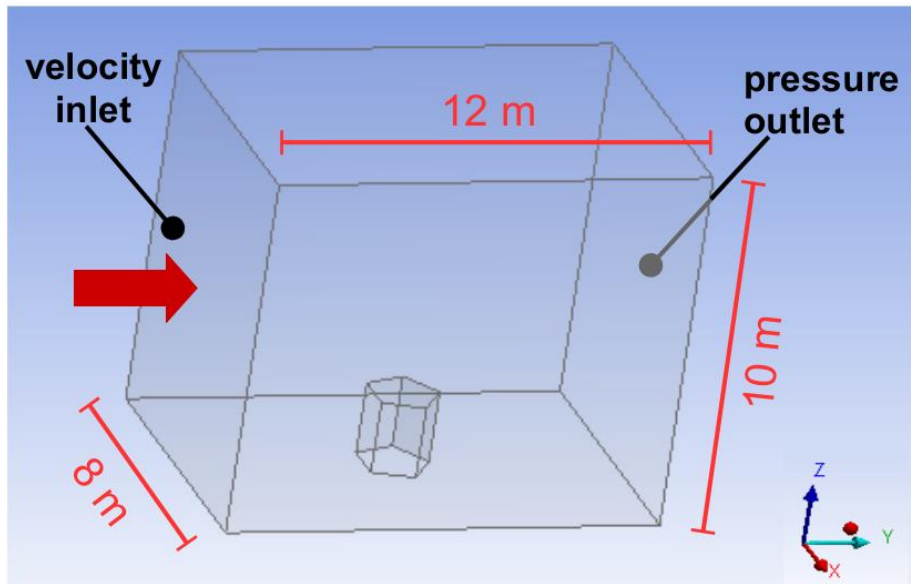


Fig. 8 The setup of the 3-D virtual wind tunnel for Task 3a. (Not drawn to scale.)