

MAE 494/598 Applied CFD, Fall 2018, Project 4 – External flow (25 points)

Hard copy of report is due at 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 3(b) which is for MAE598 only.**

General note: For the 3-D simulations in this project, even a mesh with moderate resolution can easily exceed the 500K-element limit. As a compromise, we don't demand that those 3-D cases be run with very high resolution. Further guidelines on this point will be provided in class.

Task 1

(a) Consider the system of a 2-D flow passing a cylinder as illustrated in Fig. 1. **The unit in Fig. 1 is "cm".** 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 "virtual wind tunnel" with its dimension given in Fig. 1. The system has an inlet at left and an outlet at right. The boundary conditions for the two openings are set to *velocity inlet* and *outflow*, respectively. Top and bottom are walls. The system is filled with *water* with constant density and viscosity (using the default values from Fluent database). All simulations in this task will use *Laminar model* and will seek *transient solution*. At $t = 0$, each run is initialized with zero velocity and zero gauge pressure inside the domain. Perform a transient simulation with *inlet velocity* = 0.3 cm/s and run it to $t = 1$ hour. The deliverables are (i) An estimate of Reynolds number for this case; (ii) Contour plots of *x-velocity* and *static pressure* at $t = 1$ hour. Here, the direction of x is as indicated in Fig. 1.

(b) Use the same setting as (a) but change the inlet velocity to 2 cm/s. Perform a transient simulation to $t = 1$ hour. The deliverables are (i) An estimate of Reynolds number for this case; (ii) Contour plots of *x-velocity* and *static pressure* at $t = 1$ hour; (iii) A plot of *lift coefficient* as a function of time over the last 10 minutes of the simulation (i.e., for $t = 50$ min – 1 hour). From the plot, estimate the period and amplitude of oscillation for the lift coefficient. (The lift coefficient is the non-dimensionalized lift force. The latter is the force that the fluid exerts on the cylinder in the y -direction. See Fig. 1 for the definition of the y -direction.)

(c) Use the same setting as (b) (i.e., inlet velocity = 2 cm/s) 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. For each case, plot *lift coefficient* as a function of time over the last 10 minutes of the simulation. Determine the period and amplitude of oscillation for the lift coefficient for those two cases and compare them to the case with a circular cylinder in Task 1b. Otherwise, no need to show any contour plots.

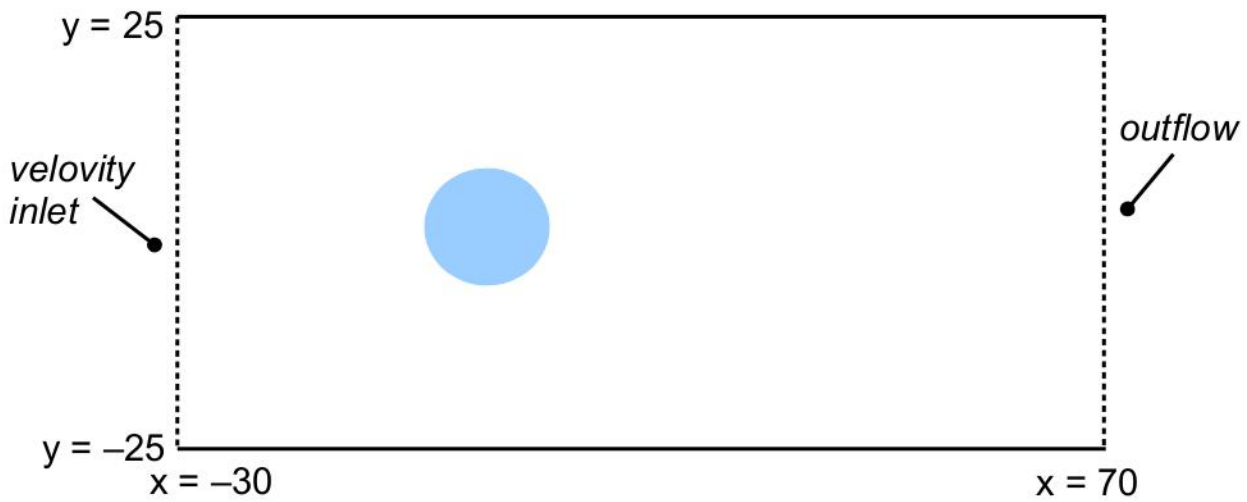


Fig. 1 The geometry of the 2D cylinder and "virtual wind tunnel" for Task 1. (Not drawn to scale.) All numbers are in cm.

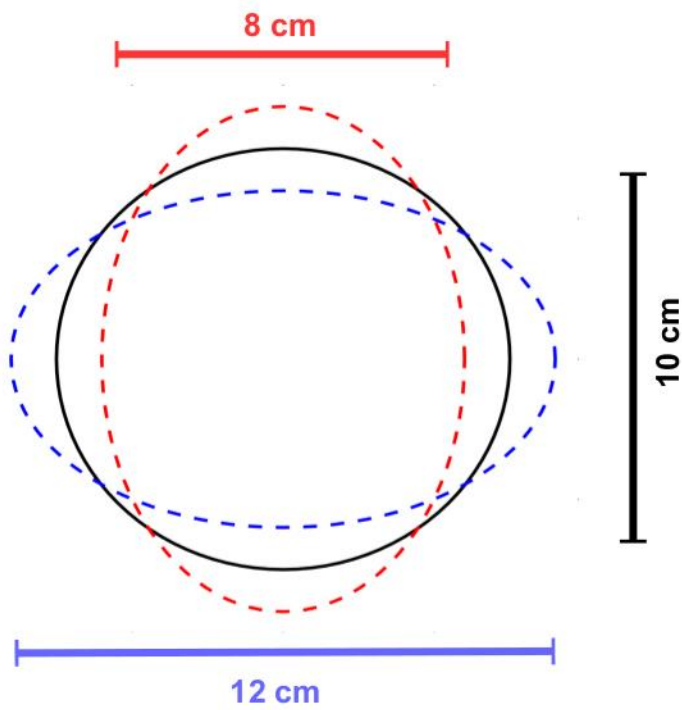


Fig. 2 The shape of the elliptical object used in Task 1c. Black: circle from Task 1b. Red: ellipse elongated in y -direction; Blue: ellipse elongated in x -direction.

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. The tunnel is filled with air with constant density and viscosity (using default properties from Fluent database). It has two openings which will be set up as *velocity inlet* and *outflow*, as indicated in Fig. 5. All other surfaces are walls. The detailed dimension (in cm) of the system is given in Fig. 6. (It suffices to show the cross section along the plane of symmetry, but 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 $\theta = 0^\circ$.

For all simulations, the *inlet velocity* is set to a uniform 50 m/s, in the direction normal to the surface of the inlet. (This is always the case, regardless of the value of the tilt angle.) All simulations will use turbulence k -epsilon model, and will seek *steady solution*. Please perform a sufficient number of iterations to ensure that the key deliverables no longer change significantly with further iterations. This can be checked by (for example) monitoring the lift or drag as a function of the number of iterations.

Perform four simulations with the tilt angle set to $\theta = 0^\circ, 15^\circ, 30^\circ$ and 45° . The deliverables are:

- (i) A plot of the mesh along the plane of symmetry, for the case with $\theta = 45^\circ$.
- (ii) Contour plots of the x -velocity along the plane of symmetry for the two cases with $\theta = 0^\circ$ and $\theta = 45^\circ$.
- (iii) 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; See Fig. 6 for the definitions of those directions.

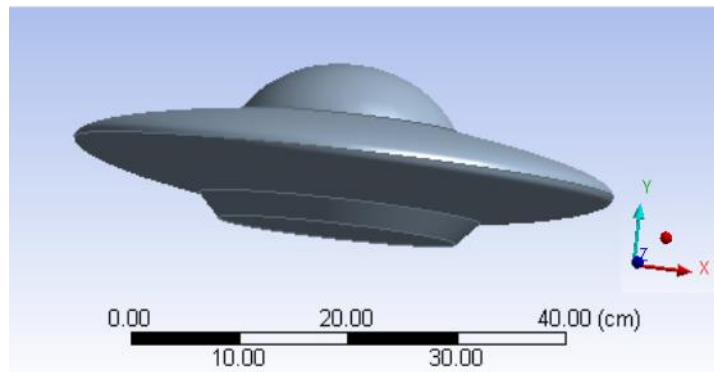


Fig. 3 The geometry of the “flying saucer”.

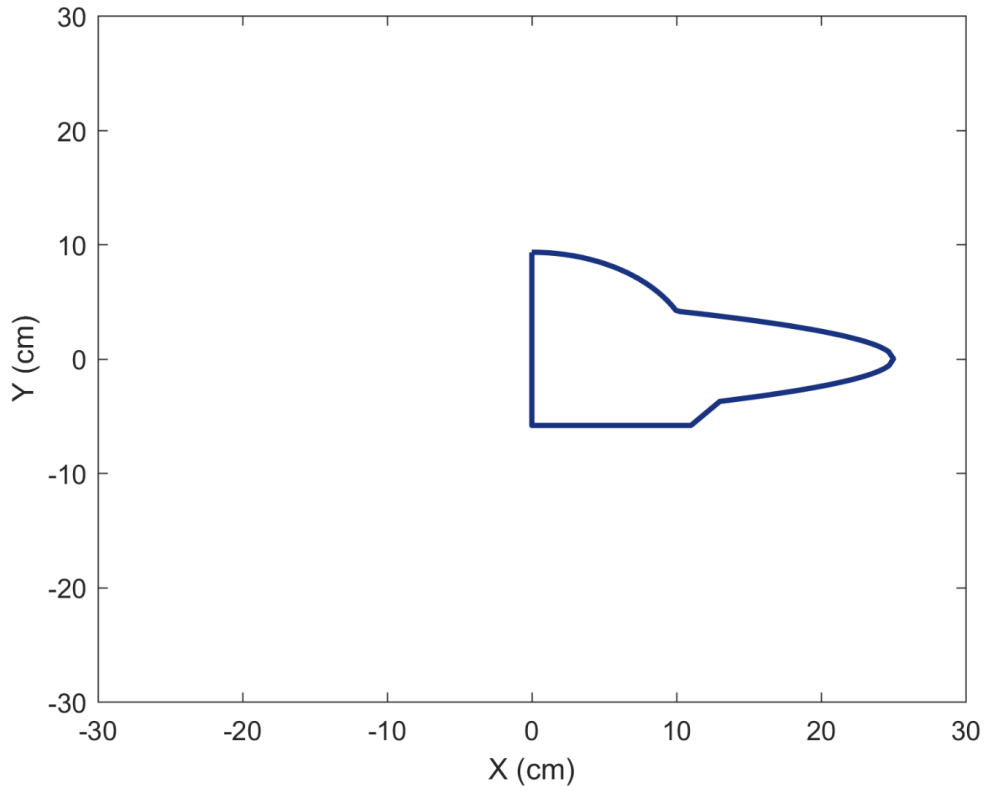


Fig. 4 The profile of the "half 2D flying saucer". The detailed data is in `flyingsaucer2DH.txt`.

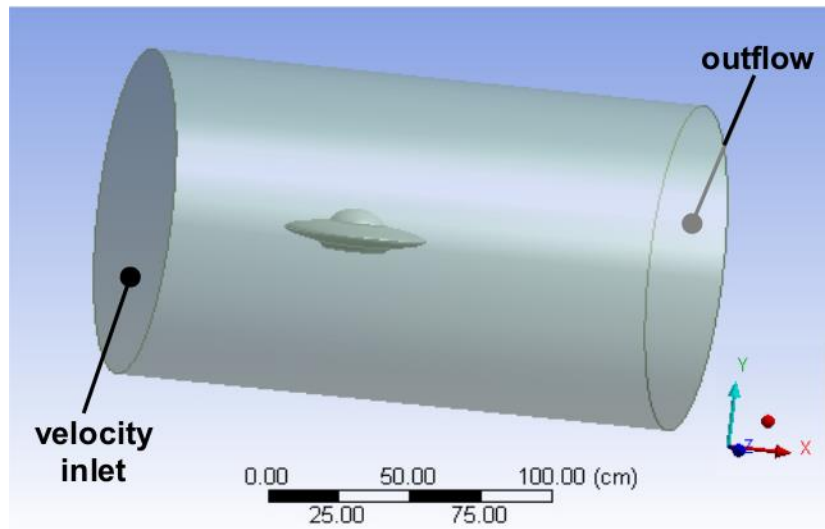


Fig. 5 The setup of the 3-D virtual wind tunnel.

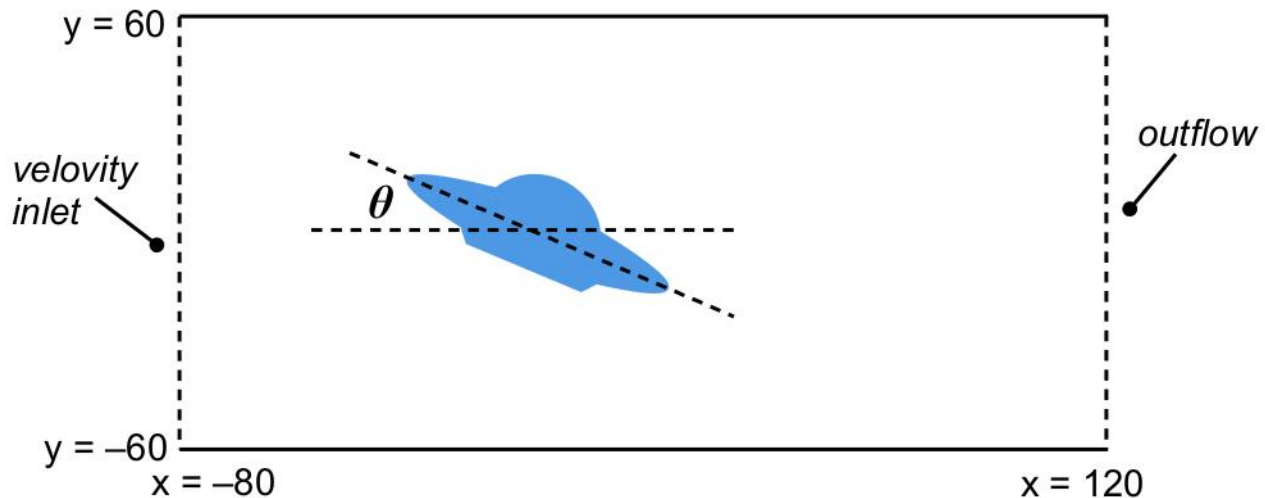


Fig. 6 The dimension (in cm) of the cylindrical virtual wind tunnel along its plane of symmetry, and the definition of the tile angle.

Task 3

In this task, we test the aerodynamics of a pentagon shaped building in a virtual wind tunnel. The building is shown in Fig. 7a. It is 1 m tall and each side of the pentagon is 0.5 m long. This building is placed at the center of the bottom plate (i.e., the “ground”) of the wind tunnel, as detailed in Fig. 8. The dimension of the wind tunnel is given in Fig. 8. The left and right openings are set as *velocity inlet* and *outflow*. All other surfaces are walls. The tunnel is filled with air with constant density and viscosity (using the properties 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, in the direction normal to the surface of the inlet.

(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** along the horizontal plane with $z = 0.5$ m. (This is the plane that cuts through the middle of the building.)

(ii) A contour plot of the **y-velocity** along the plane of symmetry. (This is the vertical plane that runs through the center of the wind tunnel.) (See Fig. 8 for the definition of **y-direction**.)

(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) (For MAE598 only) 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 different structures in pressure and velocity (for example, how pressure is distributed around the building) in the two sets of simulations affect the drag forces in those two cases.

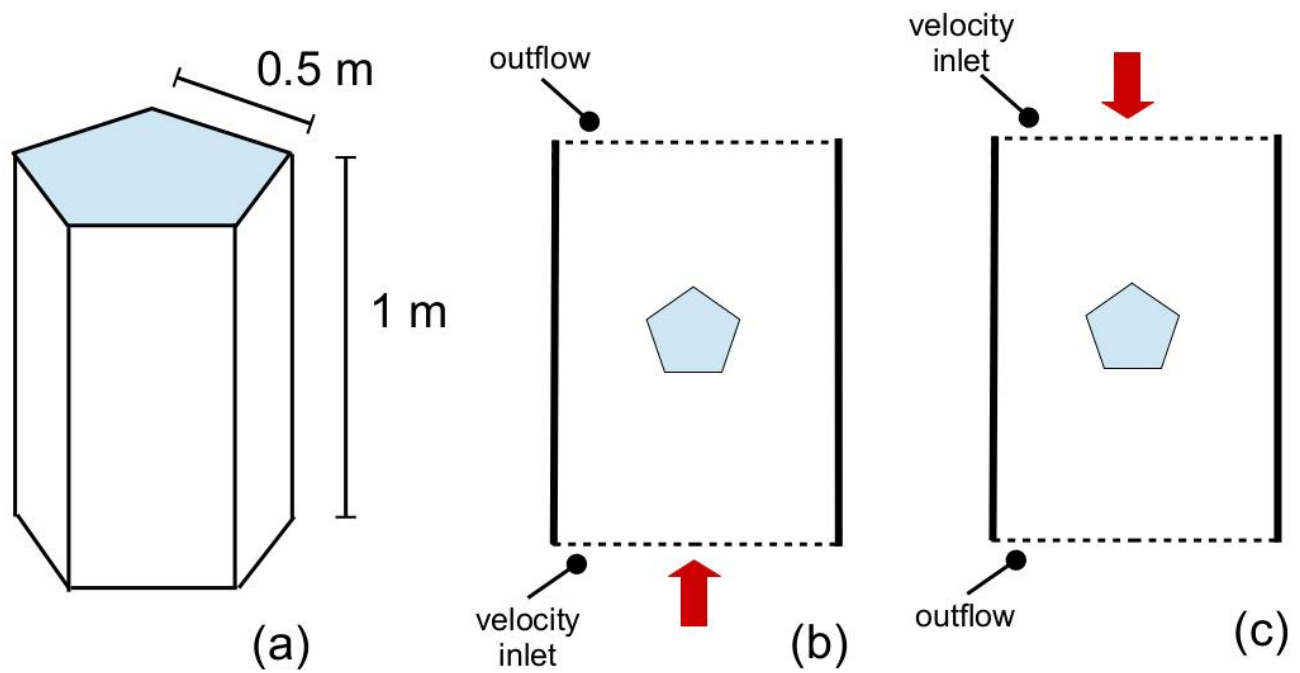


Fig. 7 The geometry of the building is shown in (a). The setups of the two runs for Task 3a and 3b are shown in (b) and (c) as the top view of the system.

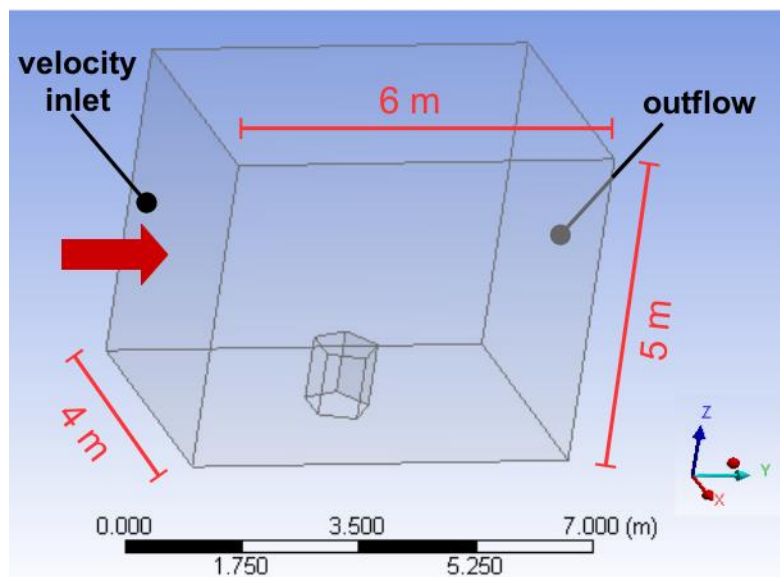


Fig. 8 The setup of the 3-D virtual wind tunnel for Task 3.