

MAE 560/460 Applied CFD, Fall 2022, Project 3 – External flow (27 points)

Please follow the rules for collaboration as given in Project 1. A statement of collaboration is required. Task 1, 2, and 3 are for both MAE560 and MAE460. **Task 4 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 4 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 regular walls with no-slip boundary condition. The system is filled with *liquid diesel* (“diesel-liquid” in Fluent database) with constant density and viscosity, using the default values from Fluent database.

At $t = 0$, the system is initialized with x -velocity = 0, y -velocity = 0, and gauge pressure = 0 within the domain. This task uses *Laminar* model to run a transient simulation to at least $t = 1$ minute. The deliverables are

(D1) A description of the *mesh resolution*, *time step size*, and *maximum number of iterations per time step*, used for the simulation. In addition, an estimate of Reynolds number for the system. (This will be just one number for the whole system.)

(D2) Contour plots of *y-velocity*, *stream function*, and *vorticity magnitude* (not to be confused with *velocity magnitude*) at $t = 1$ min.

[Note: In (D2), for the 2-D flow *vorticity* is defined as $\omega = \partial v / \partial x - \partial u / \partial y$, where u and v are x - and y -velocity. *Vorticity magnitude* is its absolute value, $|\omega|$. While one could use Custom Field Function to define this quantity, it is not necessary as Fluent already has a built-in variable for it under “velocity”. For the 2-D flow, vorticity behaves like a dynamical tracer such that one could “visualize” the flow with the contour map of vorticity. For both stream function and vorticity magnitude, please adjust the setting of contour levels to ensure good quality of the plots. For vorticity magnitude, if the default setting for contour plot is used, too many contours will be concentrated near the cylinder and within the boundary layers near the top and bottom walls. This needs to be adjusted such that one can more clearly demonstrate the structure of the “vortex street” downstream of the cylinder (*cf.* the first video shown in Lecture 20).]

(D3) A plot of the *lift force* (that fluid exerts on the cylinder) as a function of time from $t = 0$ to $t = 1$ min. Using the plot, estimates of the amplitude and period of oscillation of the lift force. (This information can be merged into the table in the deliverable for D4.) The estimate should be based on the later portion of the simulation, excluding the initial “spin-up” stage when the oscillation is not yet fully established. Amplitude can be defined as (maximum – minimum)/2.

[Note: In (D3), if the lift force is not purely oscillatory but multiple-periodic or chaotic in time, please state so. In this case, amplitude and dominating period can be approximately estimated by averaging over multiple cycles of peaks and valleys. Please then briefly explain how you compute 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 5 cm and 3 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 5 cm and 3 cm, as illustrated by the blue dashed curve in Fig. 2. The deliverables are

(D4) For each of Run 1 and Run 2, a plot of the *lift force* as a function of time from $t = 0$ to $t = 1$ 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).

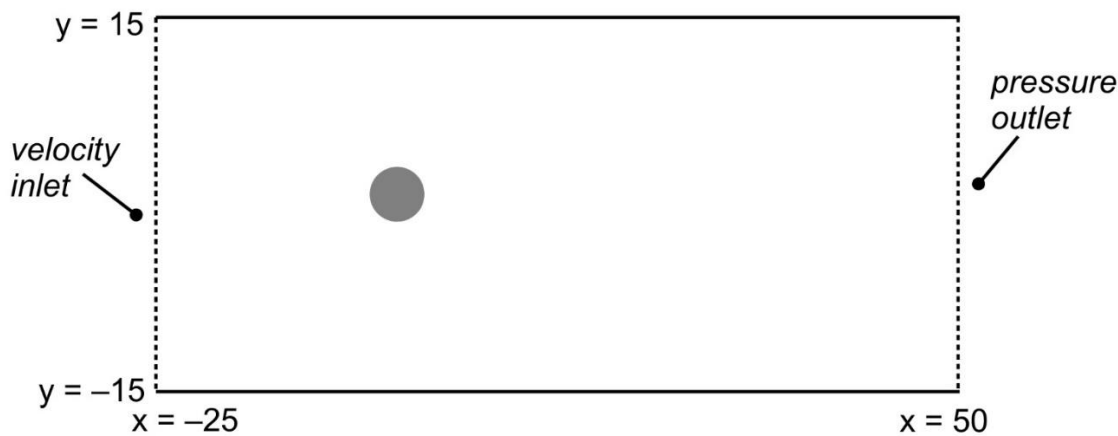


Fig. 1 The geometry of the 2D cylinder and computational domain for Task 1a. (Not drawn to scale.) All numbers indicated in the diagram are in cm. The cylinder is centered at (0, 0).

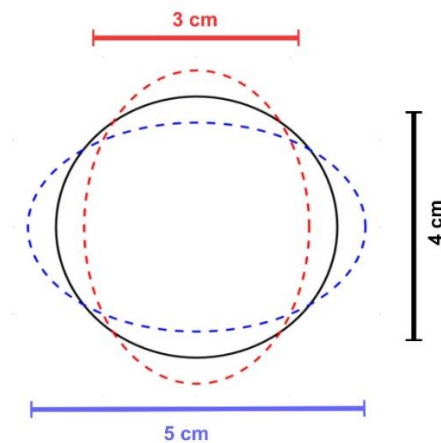


Fig. 2 The shape of the elliptical object used in Task 1b. Black: circle from Task 1a. Red: ellipse elongated in y -direction for Run 1; Blue: ellipse elongated in x -direction for Run 2.

[Remark 1: For the 2-D simulation in Task 1, Fluent internally assumes that all field variables are uniform in the “third direction”, and that the third direction has a depth of 1 m. This value is the default which can be changed in *Reference values*, but we will just use the default. Given this background, even though the simulation is 2-D, Fluent computes the lift force as if it is the force acting on a cylinder that is 1 m long in the third direction. Thus, the result has the unit of Newton.]

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 in DesignModeler by setting unit to “cm”, reading the profile data from "flyingsaucer2DH.txt" to generate a curve (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 shown in Fig. 5. It has a *velocity inlet* and a *pressure outlet*, as indicated in Fig. 5. The boundary condition for the wall of the cylinder is *wall with zero shear stress*. The surface of the flying saucer is regular wall with no-slip boundary condition. The detailed dimension (in cm) of the system is given in Fig. 6. (For clarity, we show only the cross section along the plane of symmetry. 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. (The flying saucer is placed closer to the inlet than the outlet.) A tilt angle, θ , is defined in Fig. 6. It is the angle of rotation of the 3-D object using the z -axis as the axis of rotation. 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 0.35 kg m^{-3} and constant dynamic viscosity of $1.5 \times 10^{-5} \text{ N s m}^{-2}$. [These values represent the condition of the atmosphere at approximately the top of troposphere ($\sim 10 \text{ km}$ altitude).] For all simulations in this task, the inlet velocity is set to an axially-symmetric parabolic profile for the x -velocity given by

$$u = 90 \left(1 - \left(\frac{r}{R} \right)^2 \right) \text{ m s}^{-1} ,$$

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 90 m/s at $r = 0$, and minimum of 0 at the wall where $r = R$. All simulations use turbulence k - ω model (with the default setting) and seek *steady solution*. Perform three runs with the tilt angle set to $\theta = 0^\circ$, 25° , and 50° . The deliverables are:

(D5) A plot of the mesh along the plane of symmetry for the case with $\theta = 25^\circ$.

(D6) Contour plots of x -velocity on the plane of symmetry for the three cases with $\theta = 0^\circ$, 25° , and 50° .

(D7) 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 with the numerical values of lift and drag:

	Lift force (in Newton)	Drag force (in Newton)
$\theta = 0^\circ$?	?
$\theta = 25^\circ$?	?
$\theta = 50^\circ$?	?

[**Remark 2:** 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, over a range of time in the late stage when the system has reached statistical equilibrium.]

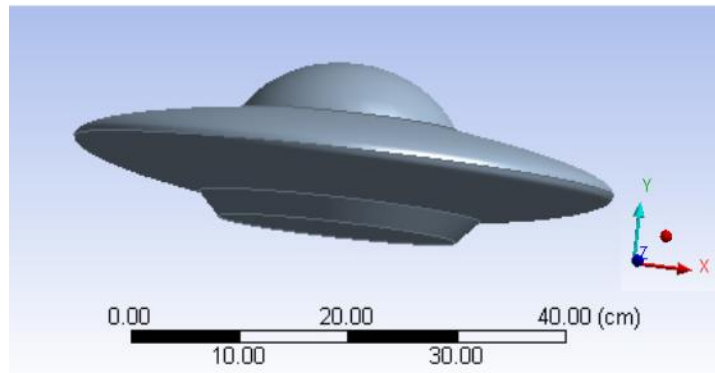


Fig. 3 The geometry of the “flying saucer”.

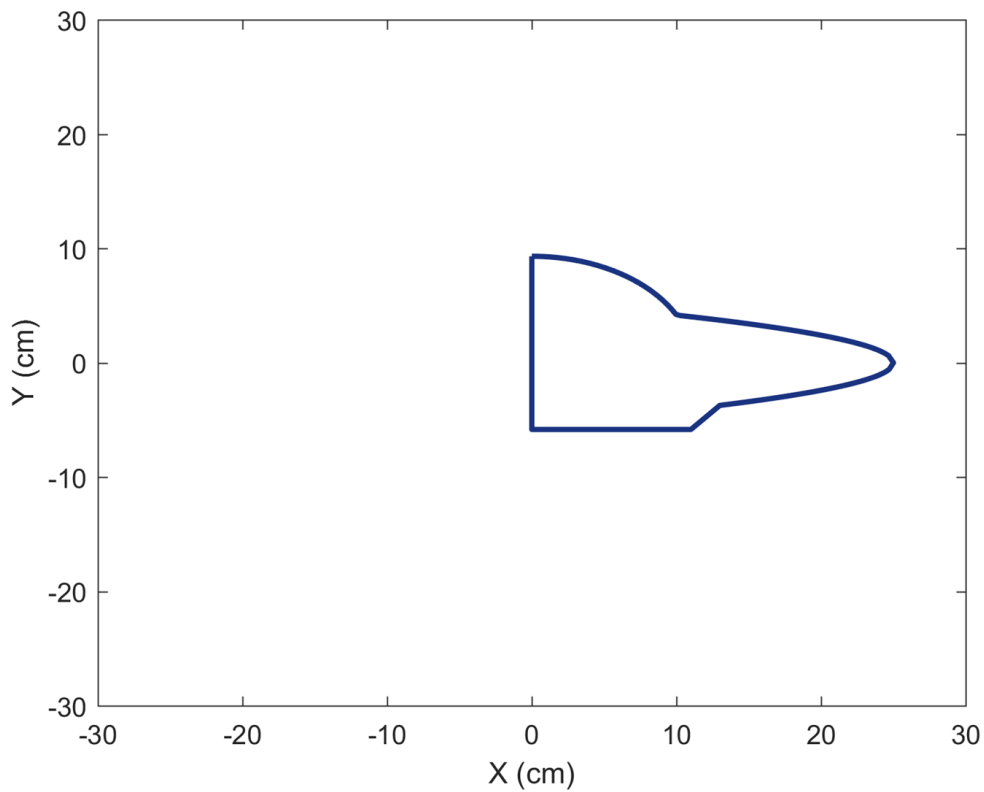


Fig. 4 The profile of the "half 2D flying saucer". The detailed data is in `flyingsaucer2DH.txt`.

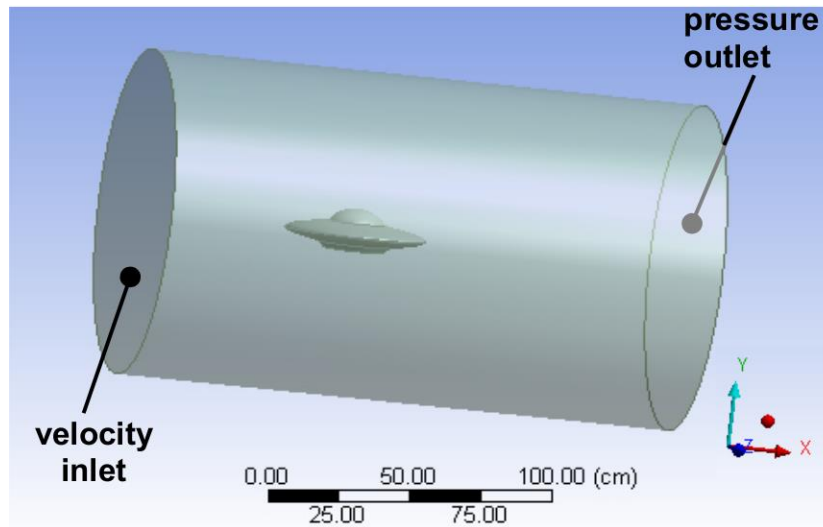


Fig. 5 The setup of the 3-D virtual wind tunnel.

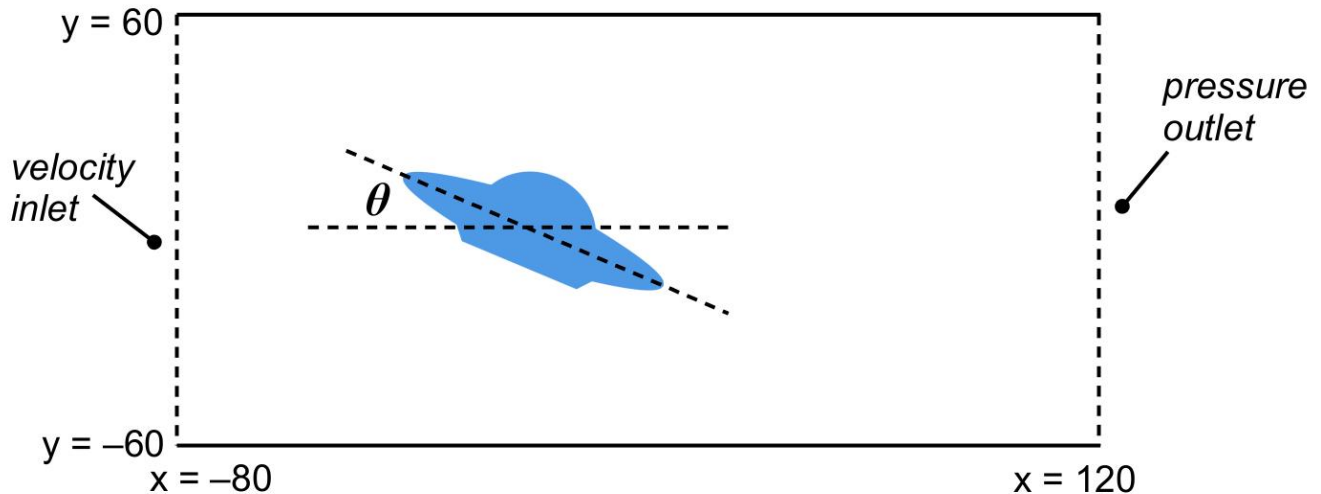


Fig. 6 The dimension (in cm) of the cylindrical virtual wind tunnel along its plane of symmetry, and the definition of the tilt angle.

Task 3

In this task, we simulate a 3-D flow passing a pentagon-shaped building in a virtual wind tunnel. The building is shown in Fig. 7a. It is 5 m tall and each side of the pentagon is 2 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* for Run 1 (Fig. 7b), and set as *pressure outlet* and *velocity inlet* for Run 2 (Fig. 7c). All other surfaces are walls. The bottom plate (i.e., the ground) and the surface of the pentagon building are regular walls with no-slip boundary condition. The two side walls and the top wall of the virtual wind tunnel are walls with *zero shear stress*.

The tunnel is filled with air with constant density and viscosity, using the default values from Fluent database. At the velocity inlet, we impose a profile of y -velocity as $v = 2z$, with z in meter and v in m/s. This gives $v = 0$ at the ground and $v = 50$ m/s at $z = 25$ m, i.e., the top of the domain. Otherwise, the imposed y -velocity is independent of x . (See Fig. 8 for the definitions of the x -, y -, and z -direction in this context.) All simulations in this task use turbulence k - ω model (with the default setting) and seek *steady solution*.

Perform two simulations with the setup based on Fig. 7b and 7c, respectively. For **Run 1** (Fig. 7b), the wind coming from the inlet is attacking a flat edge of the pentagon. For **Run 2** (Fig. 7c), a vertex of the pentagon is facing the incoming flow from the inlet. The deliverables are:

(D8) For each of **Run 1** and **Run 2**, contour plots of y -velocity on (i) the horizontal plane at $z = 2.5$ m. (This is the horizontal plane that cuts through the middle of the building), and (ii) the vertical plane that is the plane of symmetry for the system. There are total of 4 plots for this deliverable. [Please adjust contour levels to ensure clarity of the plots. A key feature of the flow field we would like to see is the “wind shadow” or wake immediately downstream of the building.]

(D9) For both runs, 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. This should be summarized in the following table:

	Total drag (N)	Pressure term of drag (N)	Viscous term of drag (N)
Run 1 (based on Fig. 7b)	?	?	?
Run 2 (based on Fig. 7c)	?	?	?

[**Remark 3:** Since Fluent might not have a built-in function for users to set the pressure term and viscous term of drag as two separate variables in *report definition*, for deliverable (D9) it is acceptable to simply use the values of the two quantities from the last iteration of the run. In other words, “averaging over a range of iterations” as suggested in *Remark 2* of Task 2 is not required. Nevertheless, one can still monitor the value of total drag, and that information can be used to determine whether the run has approximately reached equilibrium.]

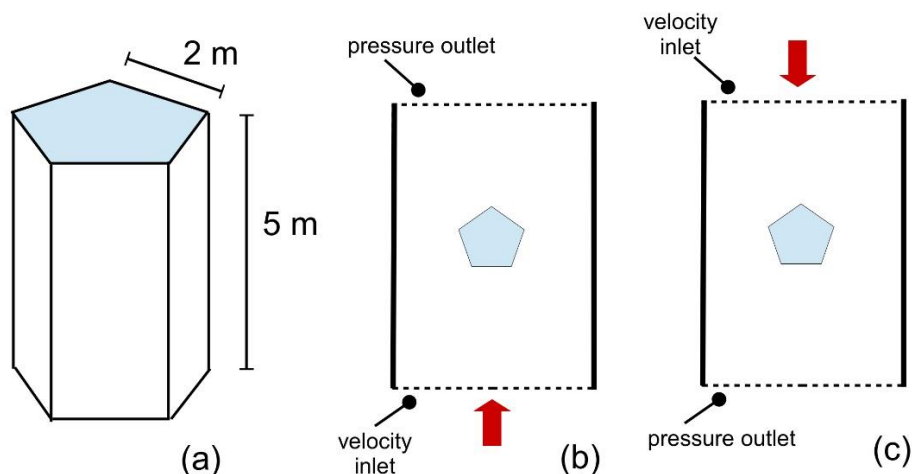


Fig. 7 The geometry of the building is shown in (a). The setups for Run 1 and Run 2 are shown in (b) and (c) as the top view of the system.

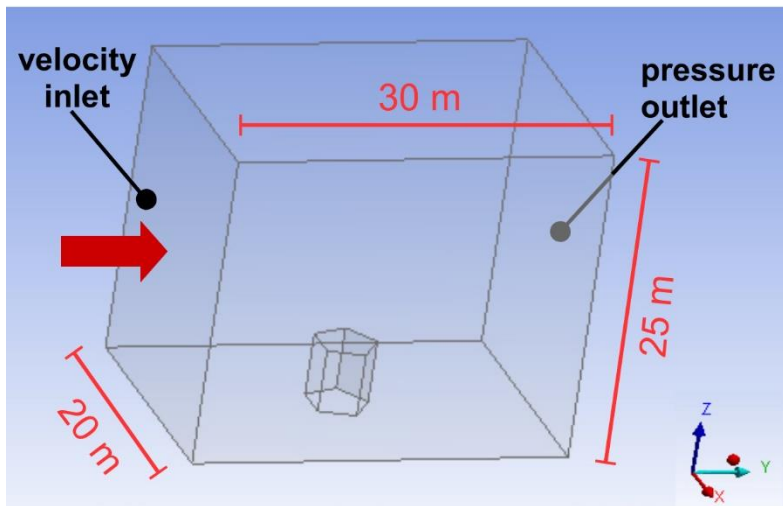


Fig. 8 The setup of the virtual wind tunnel for Task 3. Shown is the setting for Run 1. (Not drawn to scale)

Task 4 (for MAE 560 only)

This task is an extension of Task 1.

In the 3rd video shown in Lecture 20 (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 transient simulation with a cylinder that has an asymmetric shape but otherwise a cross-sectional area comparable to that of the “circular cylinder” in Task 1a. (Here, we ask that the cross-sectional area in both the x - and y -direction be comparable to the original circle.) The goal is to demonstrate that a large-amplitude response in the lift as shown in the video can be reproduced by numerical simulation. The transient simulation should be initialized in the same way as Task 1a, and be run to at least $t = 1$ min. The deliverables are

(D10) A contour plot of the *stream function* at $t = 1$ min. This plot also serves to illustrate your design of the geometry of the asymmetric cylinder.

(D11) A plot of *lift force vs. time* from the transient simulation, similar to deliverable (D3) of Task 1a. This plot should cover at least the range of $0 \leq t \leq 1$ min, but a greater range may be used if it helps to illustrate the temporal behavior of the system.

[**Remark 4:** 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.]