

MAE 598/494 ACFD, Fall 2016, Project 3 (regular tasks = 20 points)

Part I. Regular tasks for MAE 598 and MAE 494

Task 1

Consider the classic problem of flow passing a cylinder as illustrated in Fig. 1. The unit in Fig. 1 is "cm". Set up Fluent to perform a 2-D simulation by placing the 2-D cylinder (essentially a circular disk) within a "virtual wind tunnel" (which is just a simple channel in 2-D with two straight walls). The circular disk is centered at $(x, y) = (0, 0)$ and has a radius of 10 cm. The system has a simple velocity inlet at left and pressure outlet at right. For this task, consider a uniform inlet velocity.

(a) Consider the following setup for Fluent: Material: liquid kerosene with constant density and viscosity (use the default values from Fluent database); Inlet velocity: $u = 0.006$ m/s (uniform); Model: Viscous-laminar. Initialize the model with zero velocity inside the domain at $t = 0$. Perform a transient simulation to $t = 1$ hour. For this simulation, the basic requirements for meshing are (i) Setting "Sizing \rightarrow Relevance center" to "fine" to generate the initial mesh, and (ii) Further refining the mesh within the sub-domain indicated by the red dashed rectangle in Fig. 1 by "Adapt \rightarrow Region \rightarrow Adapt". (Unlike Project 2, here we need to "Adapt" instead of "Mark".) Using a mesh coarser than required will lead to a deduction. Based on the solution at $t = 1$ hour, produce the following deliverables:

- (1) An estimate of the Reynolds number of the system. Here, you can use the diameter of the cylinder as the length scale.
- (2) Contour plots of (i) velocity magnitude, (ii) y-component of velocity, and (iii) static pressure.
- (3) Line plots of the x-component of velocity along the vertical lines of $x = 50$ cm and $x = 150$ cm.

(b) Repeat (a) but now consider the modified setup: Material: water with constant density and viscosity; Inlet velocity: $u = 0.0003$ m/s (uniform); Model: Viscous-laminar. Produce deliverables (1)-(3) as described in Part (a).

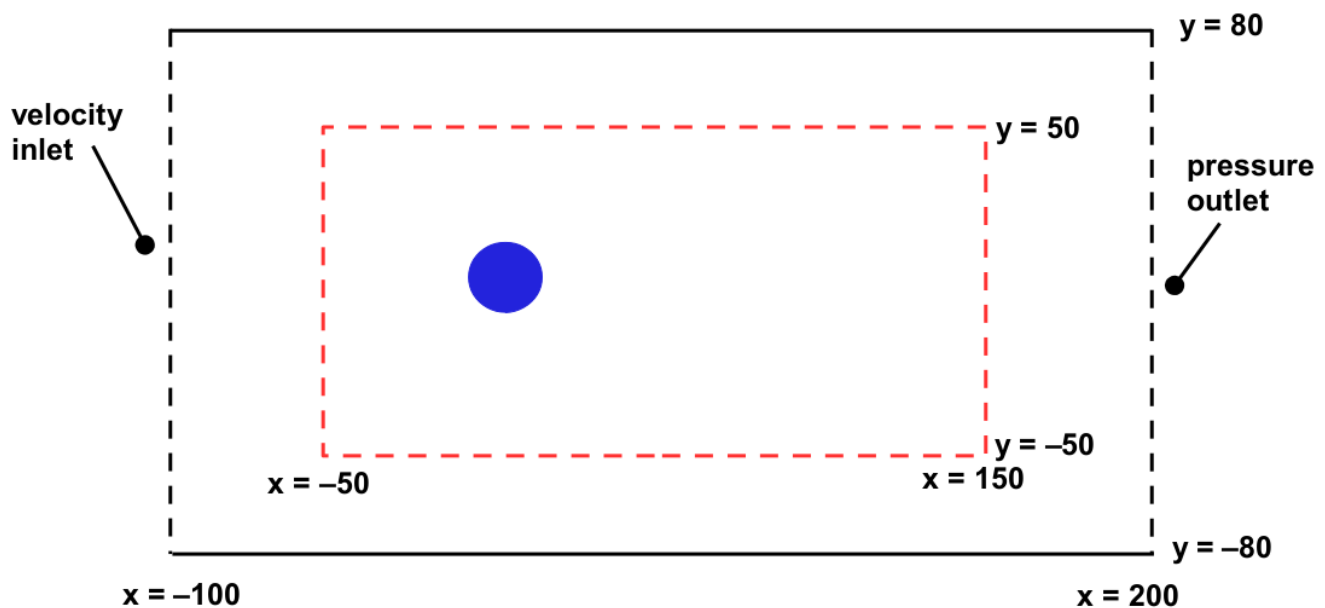


Fig. 1 The geometry of the system for Task 1.

Task 2

Consider a setup similar to Task 1 for a 2-D flow in a channel but replace the circular disk with a body with more complicated shape (called a "half fish"), as shown in Fig. 2. The unit in Fig. 2 is "cm". Be aware that the domains for Task 1 and 2 are different. The profile data for the "half fish", given in a text file named "half_fish_2016.txt", will be provided separately. Set the "unit" for the geometry modeler as "cm". With this setup, the numbers in the profile data will be regarded as in "cm". Read the profile data to Fluent to complete the generation of geometry. Note that we do not indicate the detailed dimension of the "half fish" in Fig. 2. It is not necessary because the detailed coordinates that define the outline of the body are already given by the profile data. As a quick reference, the detail of the profile of the "half fish" is shown in Fig. 3.

Consider the following setup for Fluent: Material: air with constant density and viscosity; Inlet velocity: $u = 10$ m/s (uniform); Model: Viscous-turbulence k-epsilon. Seek steady solution. Similar to Task 1, for this simulation the basic requirements for meshing are (i) Setting "Sizing \rightarrow Relevance center" to "fine" to generate the initial mesh, and (ii) Further refining the mesh within the sub-domain indicated by the red dashed rectangle in Fig. 2. Produce the following deliverables:

- (1) Contour plots of (i) velocity magnitude, (ii) stream function, and (iii) static pressure.
- (2) Calculation of the lift that the fluid exerts on the whole "half fish". Also, indicate the separate contributions by pressure and viscosity (stress) to the lift. [The lift here is the net force perpendicular to the direction of inlet velocity exerted on the "half fish". A positive lift means the net force is pointing in the positive y-direction.]
- (3) Calculation of the drag that the fluid exerts on the whole "half fish". Also, indicate the separate contributions by pressure and viscosity (stress) to the drag. [The drag here is the net force parallel to the direction of inlet velocity exerted on the "half fish". A positive drag means the net force is pointing in the positive x-direction]

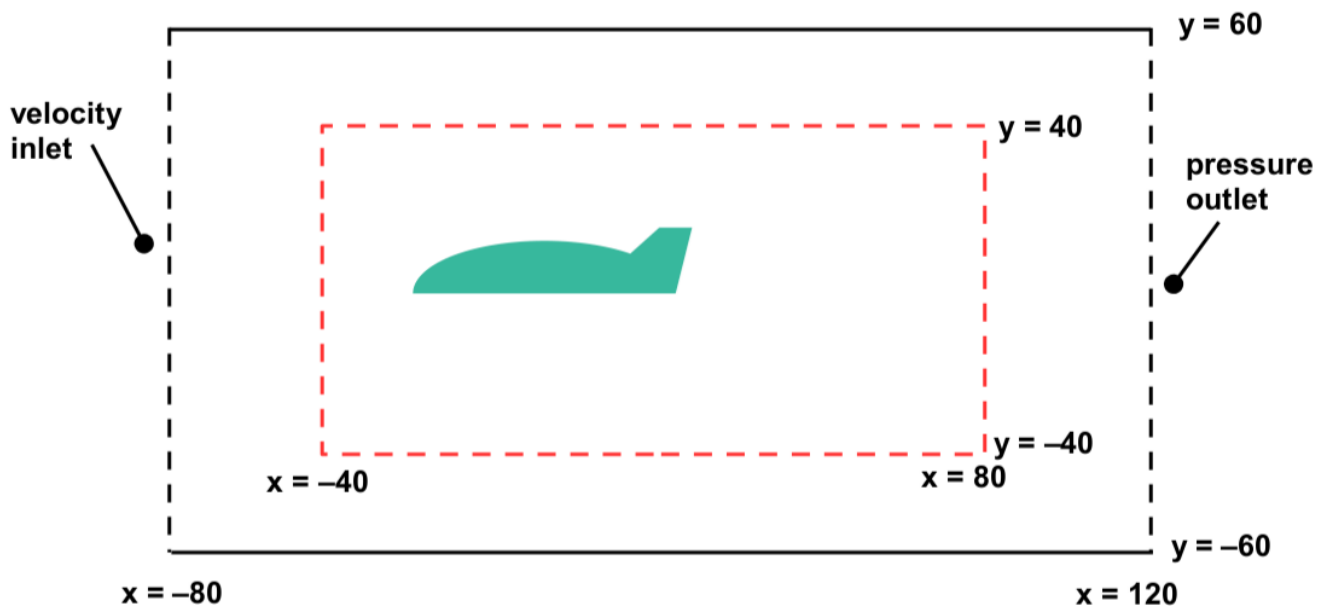


Fig. 2 The geometry of the system for Task 2. The "half fish" is not drawn to scale.

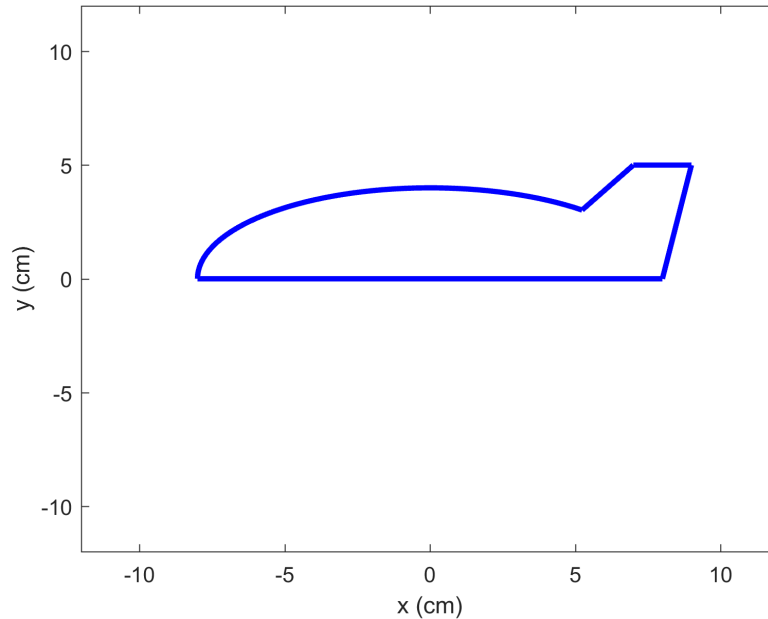


Fig. 3 The detail of the profile of the "half fish". Note that it consists of five segments.

Task 3

Consider a full 3-D version of the "virtual wind tunnel" used in Task 2 for simulating flow over a 3-D body. Here, we change the channel in Task 2 into a 3-D circular "tube" as shown in Fig. 4. The x-axis is the axis of symmetry of the tube. The tube has a velocity inlet at left and a pressure outlet at right. For this task, the 3-D body is a 3-D "full fish" which can be generated by first reading the same profile data used in Task 2, then revolving the 2-D "half fish" around the x-axis. The 3-D fish is placed in the center of the tube with the "head" of the fish facing the inlet. The cross section of the system along the x-y plane is illustrated in Fig. 5. The unit in Fig. 5 is "cm".

Fill the tube with air with constant density and viscosity. Set the inlet velocity to a uniform 10 m/s and choose the Viscous-turbulence k-epsilon model. Seek steady solution. For this simulation, the basic requirements for meshing are (i) Setting "Sizing → Relevance center" to "fine" to generate the initial mesh, and (ii) Further refining the mesh within the *sphere* as illustrated by the red dashed line in Fig. 5. This 3-D sphere is centered at $(x,y,z) = (0,0,0)$ with a radius of 15 cm. Produce the following deliverables:

- (1) A figure that shows the mesh of the system. This is to demonstrate that mesh refinement is done correctly. For example, consider showing the mesh along the plane of symmetry.
- (2) Contour plots of (i) velocity magnitude on the x-y plane, (ii) static pressure on the x-y plane, (iii) x-component of velocity on the vertical plane (parallel to the y-z plane and with a circular cross section) at $x = 25$ cm.
- (3) Calculation of the drag (i.e., force in x-direction) that the fluid exerts on the whole "fish".

Part II. Regular task for MAE 598 only

Task 4 (MAE 598 only)

Repeat Task 3 but change the uniform inlet velocity to a non-uniform, axially symmetric, parabolic profile with the same mass flow rate. This could be done by replacing the constant inlet velocity by a User Defined Function (UDF). For this task, you do not need to show the mesh but please show an additional contour plot of the x-component of velocity at the inlet and provide the printout of your UDF.

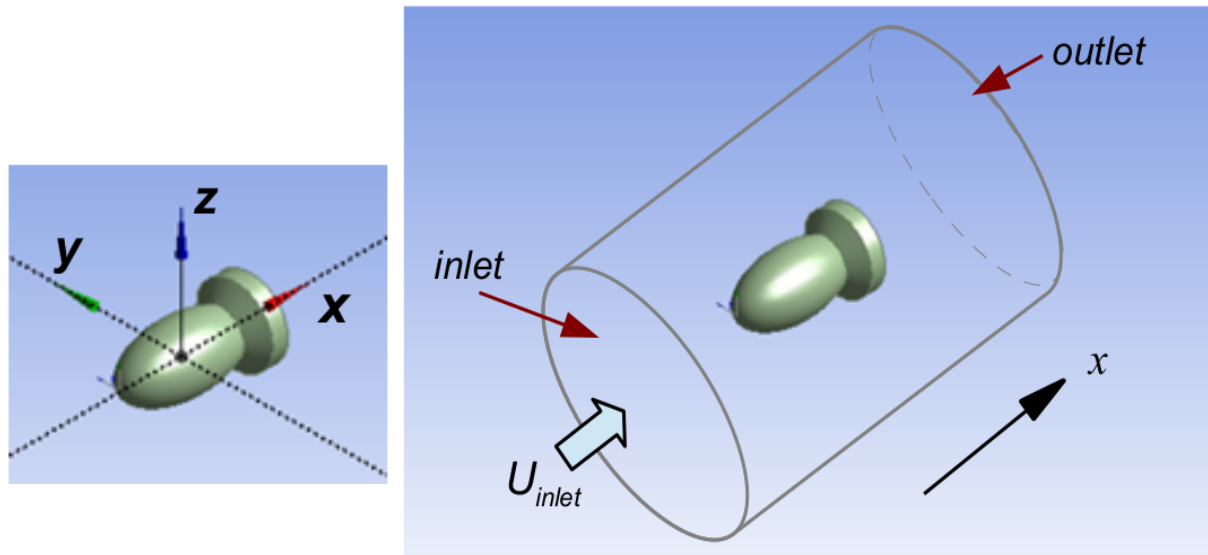


Fig. 4 The 3-D view of the geometry of the system for Task 3 and 4.

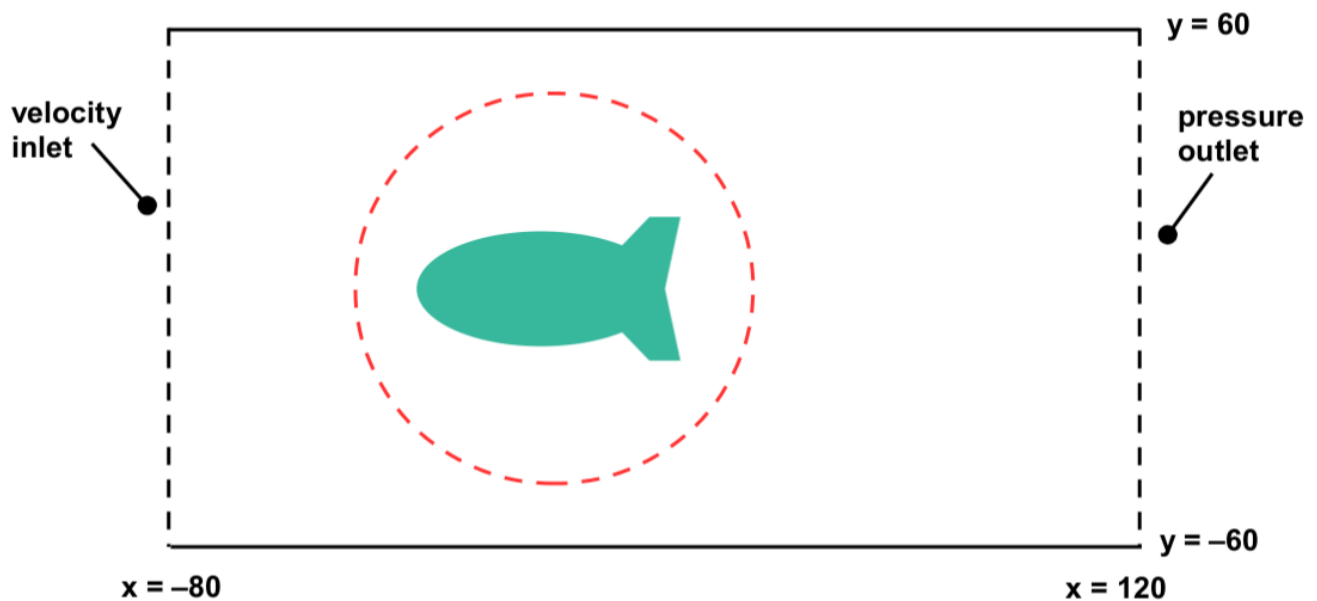


Fig. 5 The cross section of the system shown in Fig. 4 along the x-y plane.
The "fish" and red circle are not drawn to scale.

Part III. "Bonus pool" challenges

Challenge #5 (pool = 80 points, cap = 4 points)

(a) Use the setting of Task 1(a) as the starting point. First, re-run the case and calculate the lift (the y-component of the force that the fluid exerts on the cylinder) at 10 s or finer interval. Plot the lift as a function of time. Determine the amplitude and period of oscillation of the lift.

(b) Repeat (a), but now change the circular cylinder to an elliptical cylinder with approximately the same body area. Consider the two cases: (i) A cylinder elongated in y-direction, (ii) A cylinder elongated in x-direction. (See illustration in Fig. 6). Run the simulations for the two cases. Plot the lift as a function of time for each case. Determine the amplitude and period of oscillation for the two cases and compare them to the case with a circular cylinder in (a).

Note: For this challenge, it is acceptable to calculate/plot *lift coefficient* instead of *lift force*. However, if lift coefficient is chosen, it should be used consistently for all calculations in this challenge.

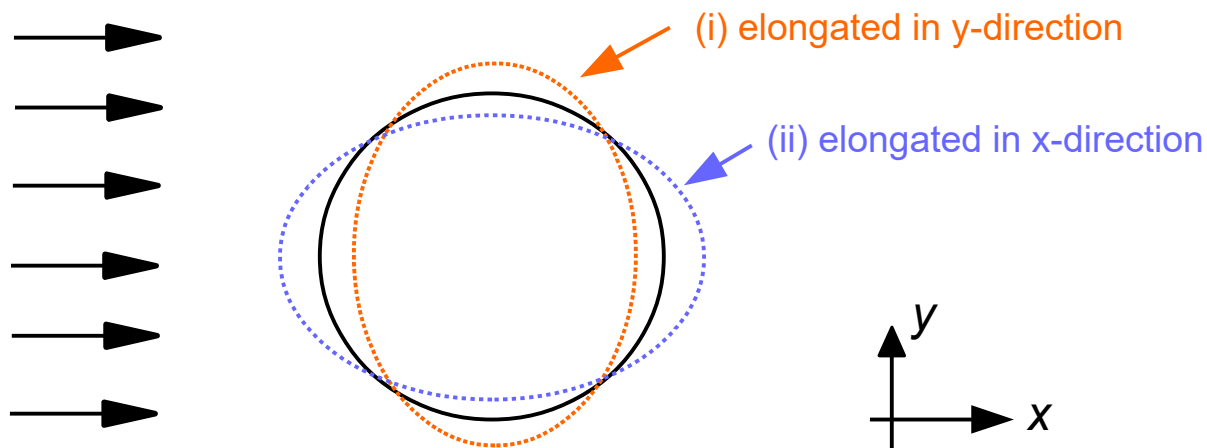


Fig. 6 The setup of the elongated cylinders for Challenge #5.

Challenge #6 (pool = 80 points, cap = 4 points)

Consider Task 3 except that the 3-D "fish" is now tilted, forming an angle θ with the x-z plane as illustrated in Fig. 7. (Note that Fig. 7 only shows the cross section on the x-y plane while this task requires that 3-D simulations be performed.) Task 3 corresponds to the case when $\theta = 0$. Here, perform additional simulations with $\theta = 15^\circ, 30^\circ$ and 45° . The deliverables are

- (1) A description of the procedure that is used to "tilt" the fish.
- (2) Contour plots of the velocity magnitude on the x-y plane (i.e., the plane as illustrated in Fig. 7) for the three cases with $\theta = 15^\circ, 30^\circ$ and 45° .
- (3) Line plots of lift and drag on the whole "fish" as a function of θ . (Essentially, just connect the four data points of the values of lift for $\theta = 0, 15^\circ, 30^\circ$ and 45° , and similarly for the drag.)

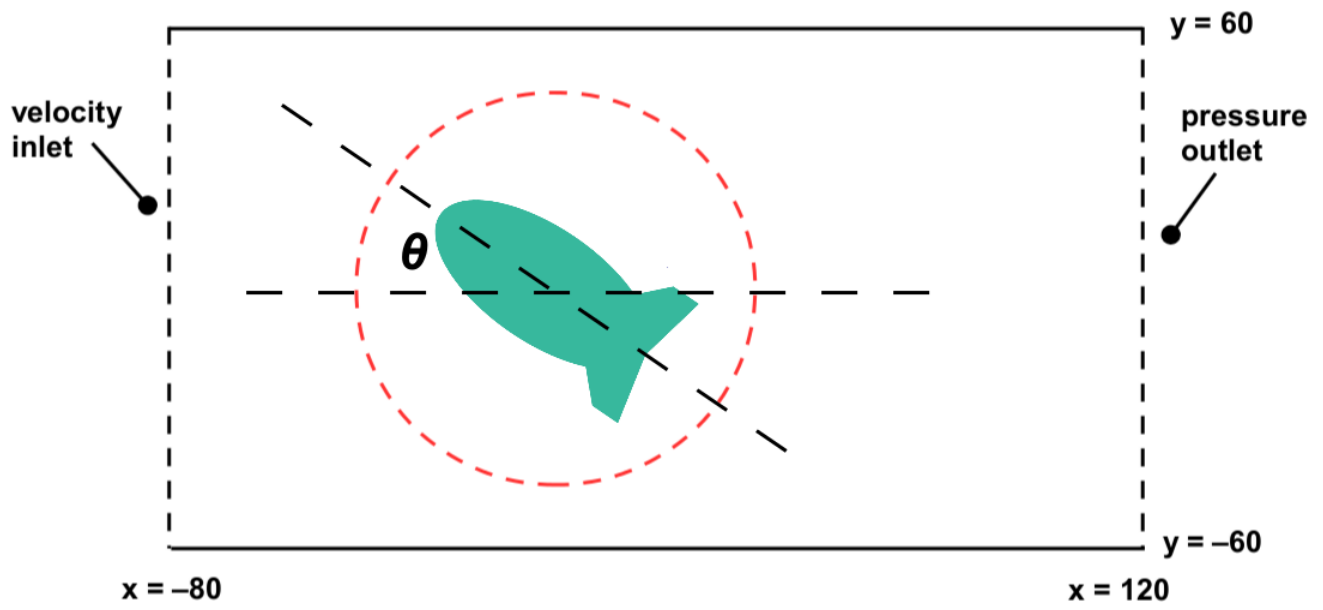


Fig. 7 Illustration of the "tilt" (by rotation around the origin) of the fish as considered in Challenge #6. The "fish" and red circle are not drawn to scale.