

MAE 598/494 ACFD, Fall 2015, Project 3 (23 points)

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 unit in Fig. 1 is "cm". The system has a simple velocity inlet at left and pressure outlet at right. For this task, we consider a constant and uniform inlet velocity.

(a) Consider the following setup for Fluent: Material: water with constant density and viscosity; Inlet velocity: $u = 0.002$ m/s (uniform); Model: Viscous-laminar. Perform a transient simulation to $t = 1$ hour. For this simulation, the specific 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. Using 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 (instead of radius) of the cylinder as the length scale.
- (2) Contour plots of (i) velocity magnitude, (ii) y-component of velocity, and (iii) 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: air with constant density and viscosity; Inlet velocity: $u = 0.5$ m/s (uniform); Model: Viscous-turbulence k-epsilon. Produce deliverables (1)-(3) as described in Part (a).

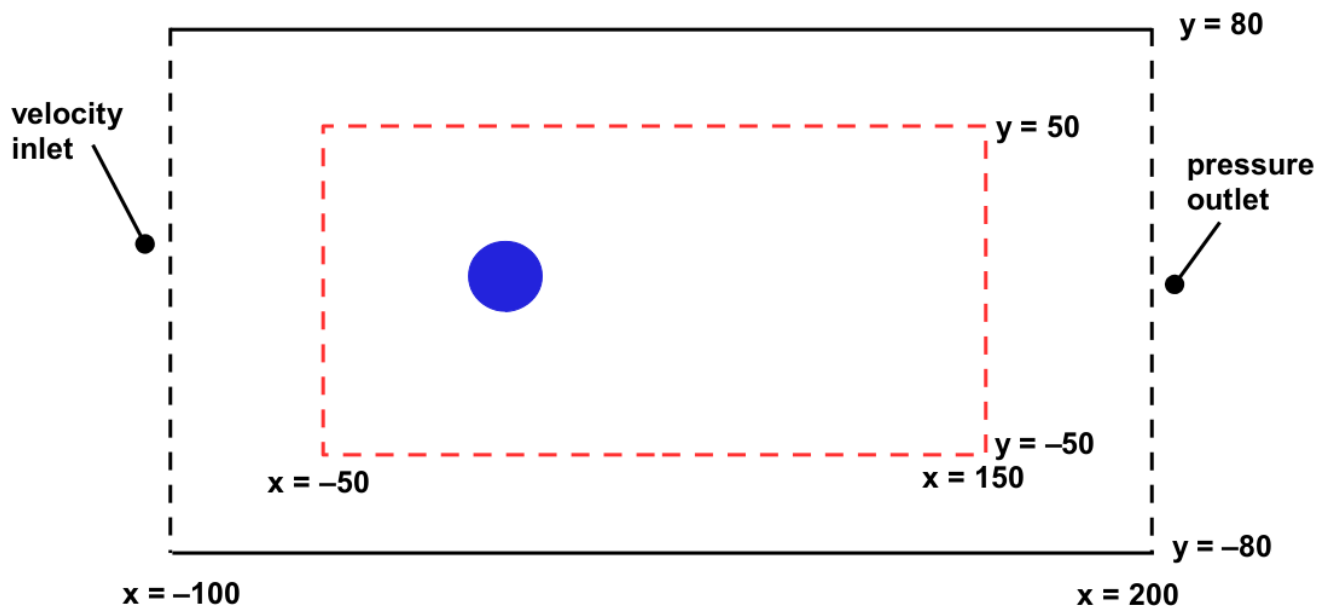


Fig. 1 The geometry of the system for Task 1.

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". The profile data for the "half fish", given in a text file named "half_fish1.txt", will be provided separately. Set the "unit" for the geometry modeler as "cm". With this setup, the numbers in the profile data are regarded as in "cm". Read the profile data to Fluent to complete the generation of geometry. Note that we did not specify the detailed dimension of the "half fish" in Fig. 2. It is not necessary because the detailed coordinates that define the shape of the body are already given by the profile data. As a quick reference, the detail of the profile of the "half fish" is illustrated in Fig. 3.

Consider the following setup for Fluent: Material: water with constant density and viscosity; Inlet velocity: $u = 0.0025$ m/s (uniform); Model: Viscous-laminar. Perform a transient simulation to $t = 10$ minutes. Similar to Task 1, for this simulation the specific 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. (Be aware that the domains for Task 1 and 2 are different.) Produce the following deliverables using the solution at $t = 10$ minutes:

(1) Contour plots of (i) velocity magnitude, (ii) y-component of velocity, and (iii) pressure.

(2) Calculation of the lift that the fluid exerts on the whole "half fish". Also, indicate the separate contributions by pressure and viscosity to the lift. Is the sign of the pressure contribution to the lift consistent with the pressure around the "half fish" as shown in the contour plot in Part (1)? For example, if the pressure contribution is negative, we should expect the pressure integrated over the surface of the upper part (above x-axis) of the silhouette of the "half fish" to be greater than its counterpart over the bottom (a straight horizontal segment) of the body, such that the net effect of pressure difference is to push the body down (i.e., in the negative y-direction).

[By definition, 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.]

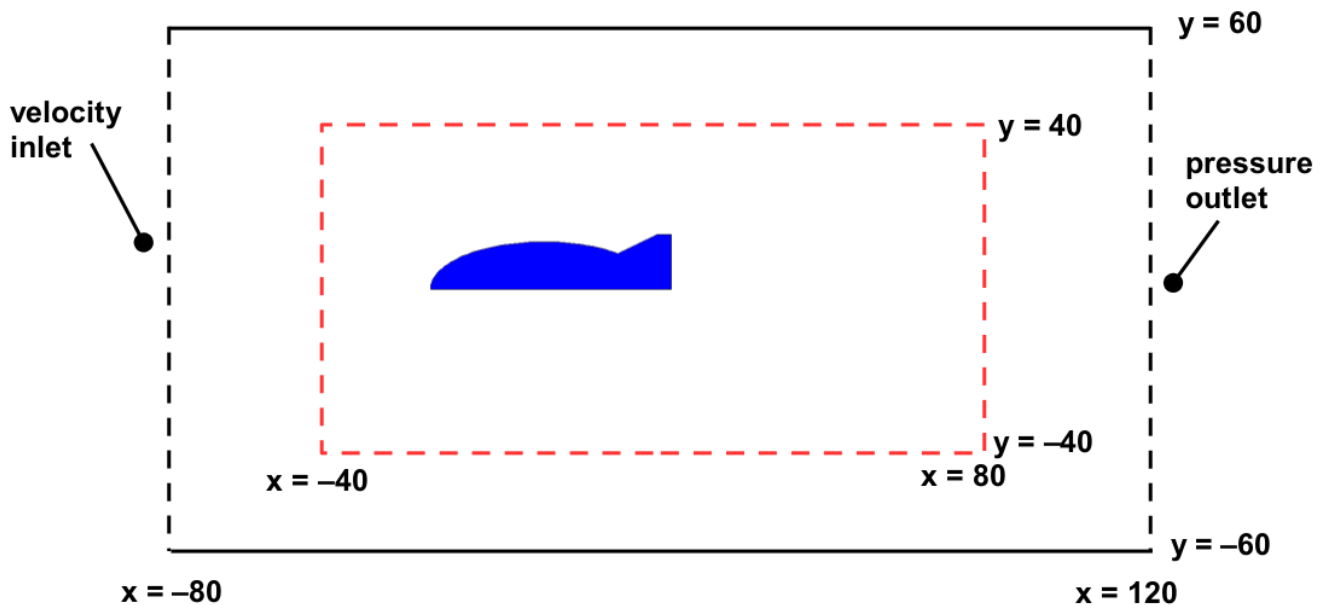


Fig. 2 The geometry of the system for Task 2.

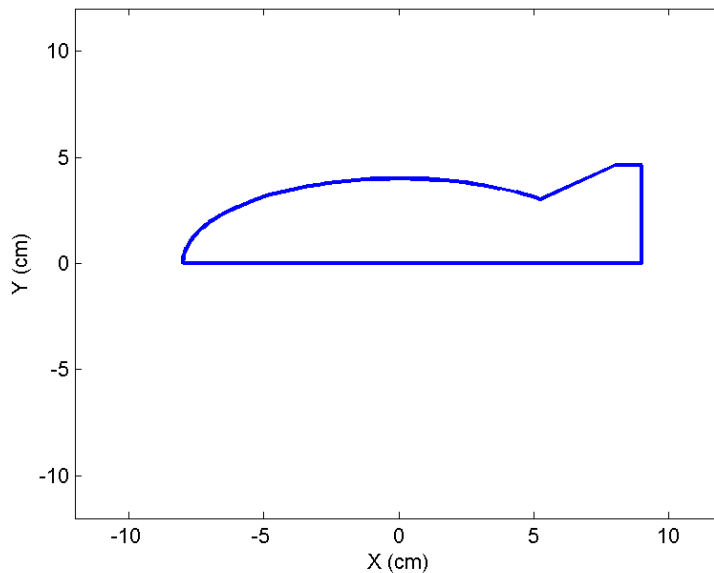


Fig. 3 The detail of the profile of the "half fish". Note that it consists of five segments.

Task 3

Consider a full 3-D version of the "virtual wind tunnel" used in Task 1 and 2 for simulating flow over a 3-D body. Here, we change the channel in Task 1 and 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 water with constant density and viscosity. Set the inlet velocity to a uniform 0.002 m/s and choose the Viscous-laminar model. Perform a transient simulation to $t = 5$ minutes. For this simulation, the specific 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 *sphere* indicated by the red dashed lines in Fig. 5. This 3-D sphere is centered at $(x,y,z) = (0,0,0)$ with a radius of 20 cm. Using the solution at $t = 10$ minutes to produce the following deliverables:

- (1) A figure that shows the mesh of the system. This is to demonstrate that mesh refinement is done correctly.
- (2) Contour plots of (i) velocity magnitude on the x-y plane, (iii) 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.

▲ 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 (cf. Challenge #2). 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.

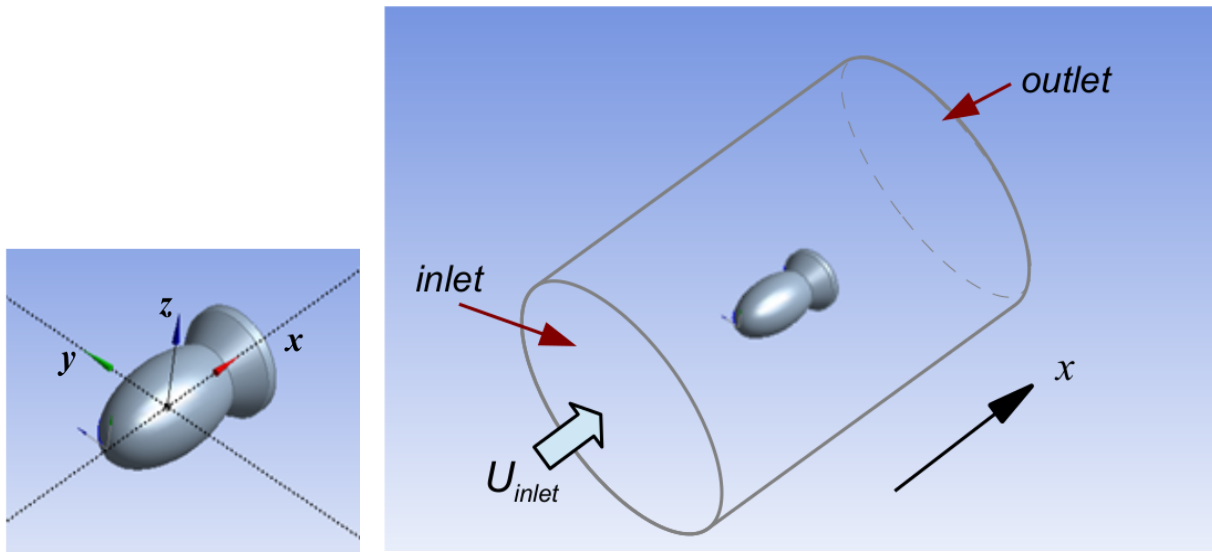


Fig. 4 The 3-D view of the geometry of the system for Task 3 and 4.

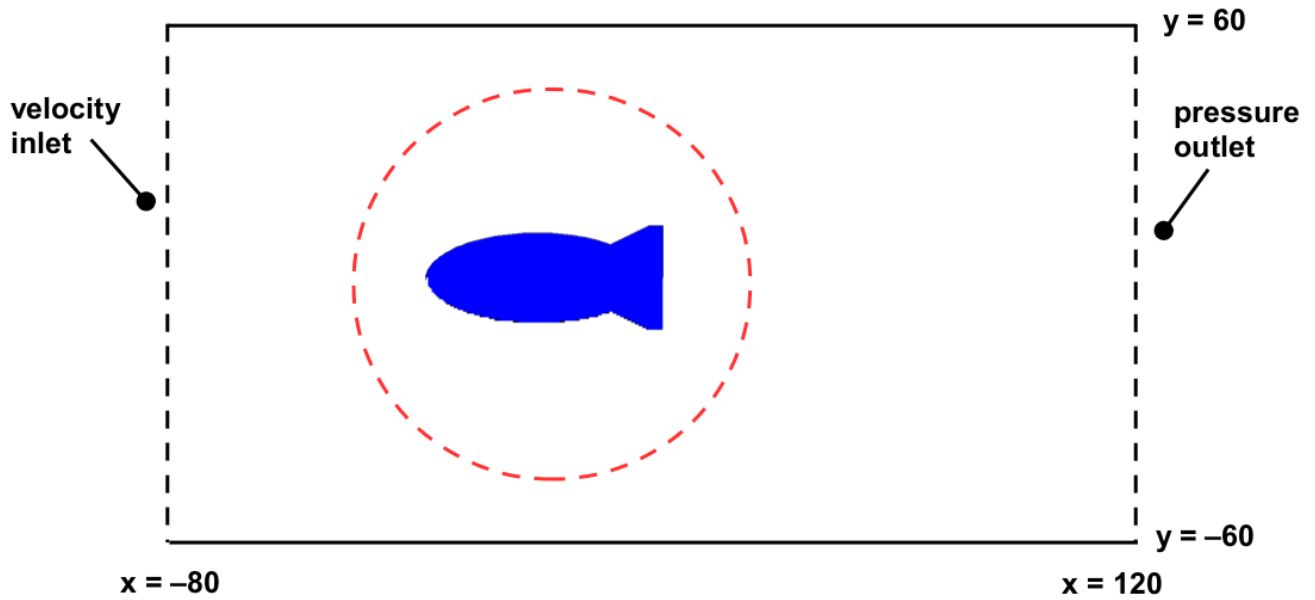


Fig. 5 The cross section of the system shown in Fig. 4 along the x-y plane.