MAE 494/598 Applied CFD, Fall 2018, Project 3 – Multiphase 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:

All tasks in this project should use the VOF model in ANSYS-Fluent for multiphase flow simulations. In all tasks, set the density of the individual phase of fluid (water, air, etc.) to constant and run the simulation with *pressure-based solver* (which is the only option when VOF model is activated in Fluent.) Since none of the tasks involves thermodynamic processes, *Energy equation* can be turned off. The choice of laminar or turbulence model will be given in the individual tasks. Except for Task 1b, all simulations are with surface tension turned off (which is the default).

Transient simulations for multiphase flows can be time consuming (in terms of CPU time). Please plan ahead to ensure completion of all tasks before the deadline. In general, a reasonably fine mesh and a small enough time step size should be used to ensure good quality of the simulation.

Task 1

This task considers a 2-D flow in a 2-D apparatus as shown in Fig. 1. The coordinate data file (`faucet_data.txt`) for the geometry has been generated by a Matlab code. The data file and code are posted separately to the class website. The data consists of the numbers for (x, y, z) coordinate but not unit. For Task 1a and 1b, choose "cm" as the unit. For Task 1c, choose "m" as the unit. For all of Task 1a-1c, the system has a velocity inlet at the entrance of the curved pipe, and three pressure outlets (all with zero gauge pressure) along the left, right, and bottom edges of the main chamber, as indicated in Fig. 2a. All other surfaces are walls.

For Task 1a and 1b, the gravity vector is pointing in the negative y-direction, as shown in Fig. 2a. For Task 1c, one flips the direction of the gravity vector, as shown in Fig. 2b.

(a) Set gravity to $g = -0.1$ m/s^2 in y-direction. At $t = 0$, fill the main rectangular chamber with ***air*** and the curved pipe with ***water***, as shown in Fig. 2a where blue and red are air and water, respectively. Set the backflow phase at all *pressure outlets* as *air*. At $t > 0$, inject ***water*** through the inlet with an *inlet velocity* of 0.1 m/s (normal to the surface of inlet and *into* the system). Use *Laminar* model and run the transient simulation to $t = 1.0$ s. The deliverables are contour plots of the *volume fraction of water* at $t = 0.3$ s and $t = 1.0$ s.

(b) Repeat the simulation in Part (a) with the same setup except that *Surface tension* is activated. (For the surface tension coefficient of water, use the default value from Fluent database.) The deliverables are contour plots of the *volume fraction of water* at $t = 0.3$ s and $t = 1.0$ s. Please also briefly comment on how the result of this simulation differs from that of Part (a).

(c) This task uses the same geometry data file as Part (a) but with the unit set to "m". (The system is much larger now!) Set gravity to $g = +9.81$ m/s^2 in y-direction. (By setting g as positive, we flip the gravity vector or flip the geometry upside-down, as compared to the setup in Task 1a.) For this task, use turbulent *k-epsilon* model. At $t = 0$, fill the main rectangular chamber with ***air*** and the curved pipe with

methane (using default properties in Fluent database), as shown in Fig. 2b where blue and red are air and methane, respectively. Set the backflow phase at all *pressure outlets* as *air*. At $t > 0$, inject ***methane*** through the inlet with an *inlet velocity* of 5 m/s. Run the transient simulation to $t = 4$ s. The deliverables are contour plots of the *volume fraction* of *methane* at $t = 1.5$ s and $t = 4$ s. [Note that methane is lighter than air. That's why we flip the geometry or direction of gravity in this task. The simulation emulates the situation when natural gas is leaked from an underground vault into open air.]

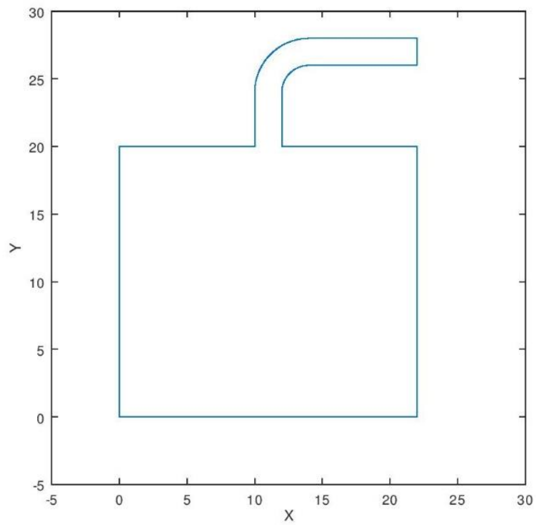

Fig. 1 The geometry used in Task 1a-1c.

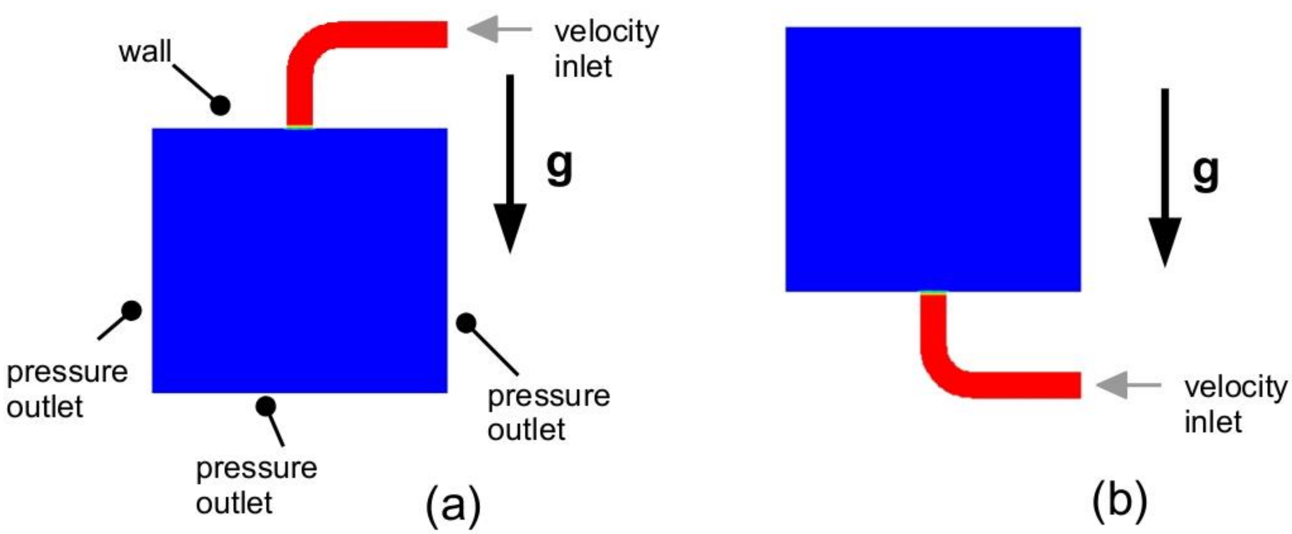

Fig. 2 (a) The setup for Task 1a and 1b. (b) The setup for Task 1c.

Task 2

(a) Consider a 3-D system with an inclined plate that forms a 15° angle with the ground. At $t = 0$, a droplet of ***glycerin*** (using default properties from Fluent database) is placed on the plate and it is shaped like a hemisphere with a radius of 3 cm. The droplet is otherwise surrounded by open air. Figure 3(a) is the cross-sectional view of the system along the vertical plane that cuts through the center of the droplet. Figure 3(b) provides a 3D view (explained below), with the bottom plate shown in blue.

Use *Laminar* model to simulate the evolution of this droplet in time. As the key deliverable, show the 3D shape of the blob of glycerin at $t = 0$, 0.1 s, and 0.2 s. It is part of your job to find a way to present the 3D structure of the blob. (A suggestion is to show the iso-surface of VF = 0.9 where VF is the volume fraction of glycerin. This is how Fig. 3b was made.)

Note: In the physical system, the droplet of glycerin is surrounded by open air without top or side boundaries. Since Ansys-Fluent can only perform a simulation over a finite domain, those boundaries need to be specified (yet in a way that will only minimally affect the main process we want to simulate). It is part of your job to define the domain and boundary conditions. *This information should be included in the report.*

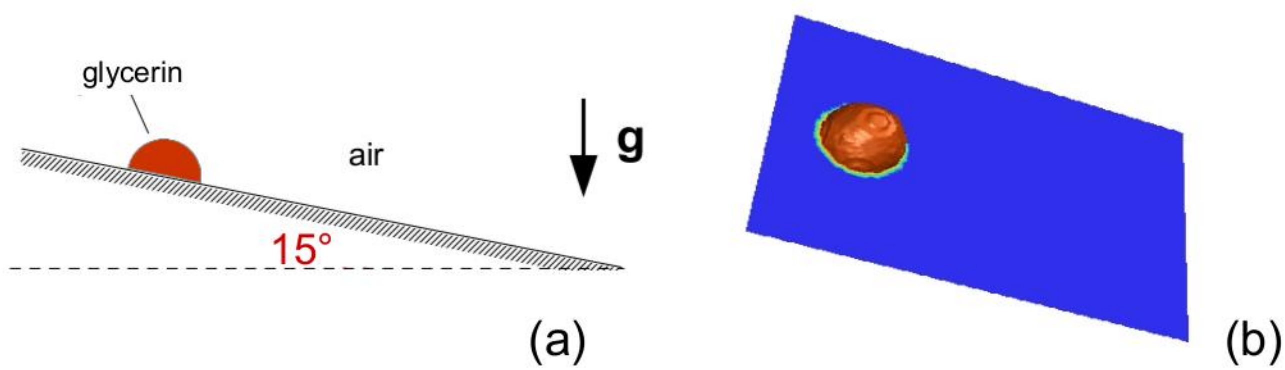

Fig. 3 The setup of the system at $t = 0$ for the simulation in Task 2.

Task 3

This simulation is for a 3-D flow. Consider a U-shaped 3D pipe, identical in geometry to that used in Task 3 of Project 2 but specifically oriented with the two legs of "U" pointing upward. The geometry of the pipe in relation to the direction of gravity vector is shown in Fig. 4a. [Otherwise, all specifications of the geometry can be found in Project 2.] Physically, the U-pipe has two openings into open space with *air* outside. (Envision that we just place the U-pipe in the middle of the classroom.) For this task, we wish to set the boundary conditions for the two top openings *such that air can freely go in and out of each of the openings*. Otherwise, there is no imposed pressure difference between the two openings, and no imposed inward or outward velocity at the two openings. The air flow through the openings will be passively driven by the spontaneous movement of water inside the pipe. As the water level rises in one of the legs of the pipe, it expels air out of the corresponding opening at the top. The reverse would happen when the water level falls. It is part of your job to choose appropriate boundary conditions to ensure that this is achieved in the simulation. Use *Laminar* model for this simulation.

At $t = 0$, the left and right pipes are filled with unequal amount of water. The water level in the left pipe is higher. The depths of *air* in the left and right pipes are 10 cm and 70 cm, respectively, as shown in Fig. 4b where blue and red indicate air and water. Initially, the fluid is sitting still with no motion. From this initial state, run a transient simulation. As the system evolves in time, we expect an oscillation of the water levels in the left and right pipes. When the water level decreases in the left pipe, the water level in the right pipe increases spontaneously. Due to the effect of viscosity, the oscillation will be damped over time. As $t \to \infty$, the water levels in the two pipes should become equal. This level at equilibrium (which is 40 cm from the top) is marked by a green line in Fig. 4b.

(a) Set the boundary conditions for the top openings to ensure that the oscillation can be properly simulated. Run the transient simulation over at least one full period of oscillation. (One cycle of oscillation is completed when the water level of the left pipe is back to maximum.) The deliverables for this part are:

(i) A description of the boundary conditions you choose for the top openings.

(ii) A line plot of the water level, *relative to the equilibrium level*, of the left pipe as a function of time. The "left pipe" is the one that initially contains more water. At $t = 0$, this "relative" water level of the left pipe is +30 cm. Note that this quantity can turn negative over half of the cycle of the oscillation. (If the air-water interface is not strictly horizontal, use the averaged depth.) Determine the approximate period of the oscillation, i.e., the time for the oscillation to complete one cycle.

(iii) Let t_1 be the time when the water levels of the left and right pipes first become equal, and t_2 the second time the two levels become equal (t_1 and t_2 are approximately the times at 1/4 and 3/4 period of the oscillation). Make contour plots of the x-velocity on the plane of symmetry at $t = t_1$ and $t = t_2$. <u>The x-direction is as indicated in Fig. 4, i.e., x increases from left pipe to right pipe. Please be careful about this specification</u>. We are particularly interested in the structure of the velocity over the curved section (particularly at the bottom) of the U-pipe. Please adjust the contour interval to highlight the pattern of velocity over that section. At the bottom of the pipe, does the maximum velocity occur near the inner edge or outer edge?

(b) (For MAE598 only) Investigate how the boundary conditions at the top openings affect the solution. For example, one can readily check that Fluent will not run the simulation if both openings are set as "wall". Try at least three different combinations of boundary conditions (in each combination, the boundary conditions at the left and right openings can be different) other than the one that you already used in Part (a) and the "wall and wall" combination mentioned above. Discuss whether those combinations work in producing the oscillation. Otherwise, no need to show any plots.

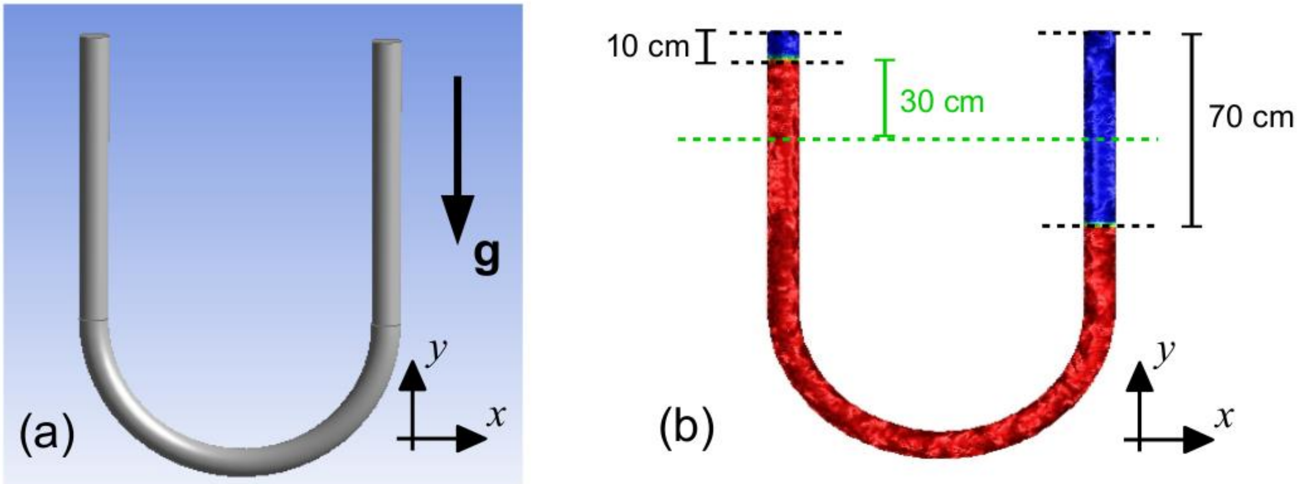

Fig. 4 (a) The geometry of the U-pipe. (b) The initial state for the simulation.