MAE 560/460 Applied CFD, Fall 2020
Project 2 – Multiphase flow (25 points)

The report for this project is due (Arizona time) 11:59 PM, Friday, November 13th. Please upload the report to Canvas as a single PDF file. Please follow the rules on collaboration as detailed in Project 1. **A statement on collaboration is mandatory for all**.

All tasks, except Task 4, are for both MAE460 and MAE560. Task 4 is for MAE560 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. Transient simulations for multiphase flows can be time consuming. 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 simulates the leaking of natural gas from an underground vault into open air, in a pure 2-D setting. Consider the computational domain shown in Fig. 1. The lateral and top boundaries are all open to air. A side pipe is connected to a pressurized reservoir of *methane* (representing natural gas). This task includes two transient simulations. In both, use the default constant values of density and viscosity from Fluent database for *air* and *methane*, and set gravity to the regular $g = -9.81$ m/s^2 in the vertical direction (the "y direction" in Fig. 1). Use turbulence *k-epsilon* model with default setting. Since both phases are gases, the effect of surface tension can be ignored.

(a) Case I: Set all three boundaries marked by A, B, and C to *pressure outlet* with zero gauge pressure, and with backflow phase set to *air*. The boundary marked by D is set as a *pressure inlet* with gauge pressure = 75 Pa. Methane is pumped through boundary D into the domain. At $t = 0$, fill the entire domain (including the side pipe) with *air*. Initialize the system with gauge pressue = 0 and velocity = 0. For the turbulence parameters, set initial *turbulence kinetic energy* to $1 \times 10^{-5} m^2 s^{-2}$, and *turbulence dissipation rate* to $1 \times 10^{-6} m^2 s^{-3}$. Perform the transient simulation to $t = 7$ s. The deliverable is a contour plot of the volume fraction of methane at $t = 7$ s.

(b) Case II: Use the same setting as (a) except that the left boundary marked by A is replaced by a *velocity inlet*, with an imposed velocity profile for the x-velocity given by $u = 0.4\, y - 0.008\, y^2$ where u is in m/s and y in m. This gives a parabolic profile with $u = 0$ at the surface ($y = 0$) and top of the domain ($y = 50$ m), and u attains the maximum of 5 m/s at $y = 25$ m. (See a sketch in Fig. 1.) Otherwise, the gauge pressure at A is still set to 0. At $t = 0$, initialize the system in the same way as Case I. Perform the transient simulation to $t = 7$ s. The deliverable is a contour plot of the volume fraction of methane at $t = 7$ s.

As an additional deliverable for this task, describe the mesh resolution and time step size used in the simulations in (a) and (b).

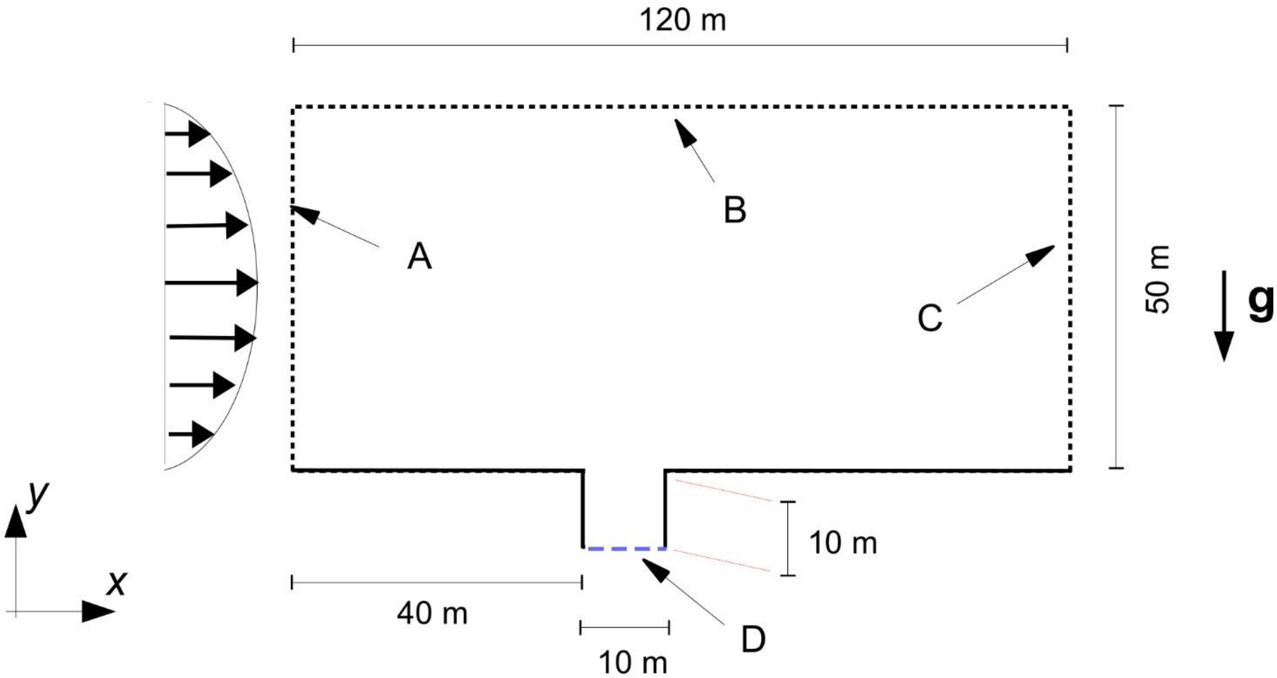

Fig. 1 The geometry of the computational domain for Task 1.

Task 2

In this task, we simulate the process of a falling water droplet impacting on a flat water surface, in a pure 2-D setting. The geometry of the system is a simple 50 cm x 50 cm square bucket that is open (to *air*) at the top and with the other three sides being walls, as shown in Fig. 2. At $t = 0$, the bucket is partially filled with *water* to the depth of 20 cm, and the rest filled with *air*. (See the illustration in Fig. 2.) In addition, a circular droplet with *diameter* of 5 cm is placed in the middle of the bucket, with its center placed 20 cm above the water surface. More precisely, if the lower-left corner of the bucket is $(x, y) = (0,0)$, the center of the droplet is $(x, y) = (25 \text{ cm}, 40 \text{ cm})$.

For the transient simulation, set gravity to $g = -9.81$ m/s^2 in the vertical direction. Initialize the system with zero gauge pressure and zero velocity. Use *Laminar* model and turn on *surface tension* modeling. Use default values from Fluent database for the density and viscosity of water and air, and surface tension for air-water interaction. At $t > 0$, the droplet will begin to fall and eventually impact the water surface. Run the simulation to $t = 0.5$ s. The deliverables are

(i) Contour plots of the *volume fraction of water* at $t = 0.2$ s, 0.3 s, and 0.5 s. (Three separate plots.)

(ii) A contour plot of *velocity magnitude* at $t = 0.5$ s.

(iii) A description of the mesh resolution and time step size used in the simulation.

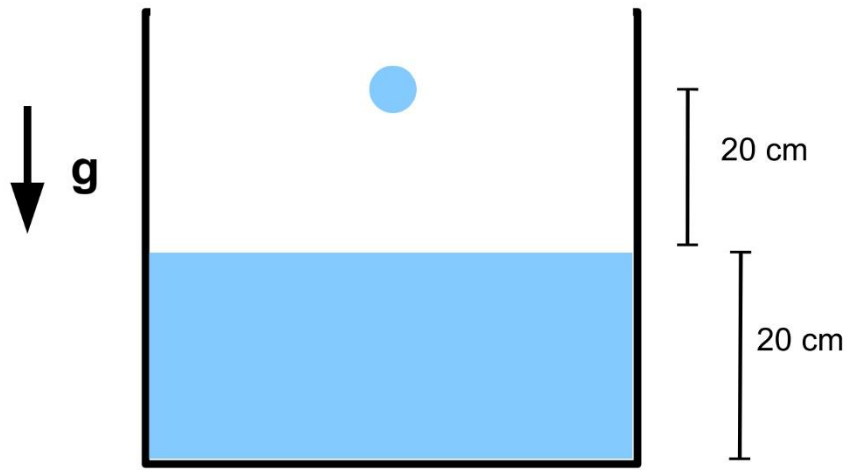

Fig. 2 The geometry and initial state used in Task 2.

Task 3

Consider a 3-D system with an inclined plate that forms a 45° angle with the ground. At $t = 0$, a droplet of *glycerin* is placed on the plate and it is shaped like a hemisphere with a *radius* of 1.5 cm (*diameter* of 3 cm). The droplet is otherwise surrounded by open air. Set gravity to the regular $g = -9.81$ m/s^2 in the vertical direction. 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 3-D isometric view (explained below), with the bottom plate shown in blue. This task will use Ansys-Fluent with VOF method to simulate the temporal evolution of the droplet. In the physical system, the droplet of glycerin is surrounded by open air without top or side boundaries. To perform the numerical simulation, one needs to set a computational domain and specify the boundary conditions (in a way that will only minimally affect the main process to be simulated). Appropriate mesh resolution and time step size should be used to ensure that the result is robust. For this simulation, it is appropriate to use *Laminar* model and use the default values for the physical properties (density, viscosity) of *glycerin* and *air*. Surface tension modeling should be turned on. The surface tension coefficient for glycerin-air interaction can be set to a constant of 0.06 N/m.

Perform a transient simulation to $t = 0.1$ s. The key deliverables are

(i) A description of the *computational domain*, *boundary conditions*, *mesh resolution*, and *time step size* used in the simulation.

(ii) Three plots in the fashion of Fig. 3b that show the 3-D shape of the blob of glycerin at $t = 0$, 0.05 s, and 0.1 s. It is part of your job to find a way to present the 3-D 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.

(iii) Three contour plots of the *volume fraction* of *glycerin* on the plane of symmetry, at $t = 0$, 0.05 s, and 0.1 s.

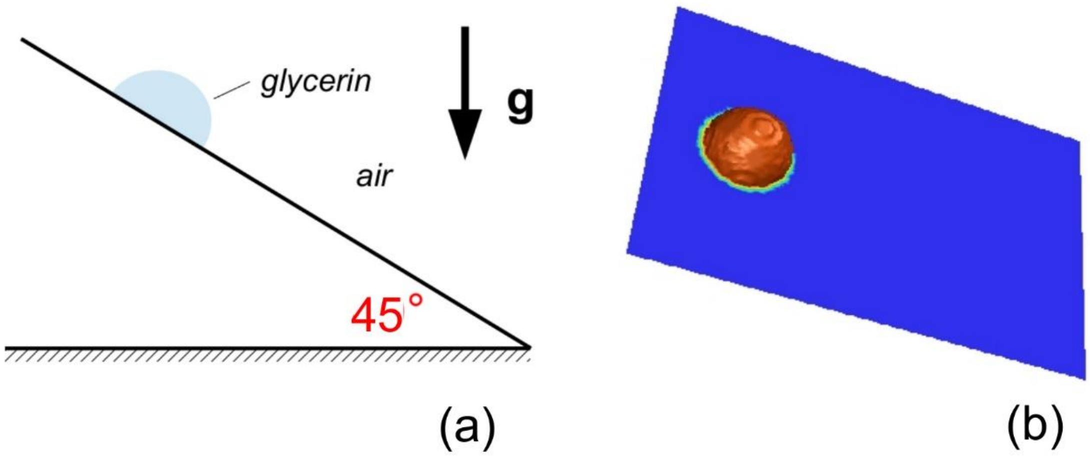

Fig. 3 The setup of the system at t = 0 for the simulation in Task 3.

Task 4 **(For MAE560 only)**

Please turn off surface tension for this task. (We use this setup to eliminate some ambiguities in the definition of "h" in one of the deliverables.)

Consider the flow in a U-shaped 3-D pipe with a circular cross section. The geometry of the pipe along the plane of symmetry is shown in Fig. 4. The *radius* of the pipe is 2 cm (*diameter* is 4 cm). The U-shaped pipe can be constructed by merging 3 segments together. Two are straight pipes, each with a length of 20 cm. The third, the "curved section", traces a half annulus (spanning 180° angle) with inner radius = 8 cm and outer radius = 12 cm in the plane of symmetry. For the simulation in this task, the full 3-D pipe is oriented with the two "legs" pointing upward while gravity is pointing downward, as illustrated in Fig. 5a. The simulation for this task can be performed using either the full-pipe geometry, or half-pipe geometry by invoking symmetry. Please indicate your choice.

Physically, the U-pipe has two openings into open space with *air* outside. 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 level of water rises in one of the legs of the pipe, it expels air out of the corresponding opening at the top. The reverse would happen in the other leg of the pipe, in which the level of water falls and air is sucked into that leg through the opening at the top. It is part of your job to choose appropriate boundary conditions to ensure that this is achieved in the simulation. (There are many possible choices for this purpose. Also, note that the boundary conditions for the two openings can be different.) 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 5 cm and 15 cm, respectively, as shown in Fig. 5b where blue and red indicate *air* and *water*. Set the densities of air and water as constant using the data from Fluent database. Initially, the whole fluid body (consisting of air and water) 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 levels of water in the left and right legs of the pipe. When the water level decreases in the left leg of the pipe, the level in the right leg increases spontaneously. Due to the effect of viscosity, the oscillation will be damped over time. As $t \rightarrow \infty$, the levels of liquid in the two pipes should become equal. This level at equilibrium (which is 10 cm from the top) is marked by a green dashed line in Fig. 5b. A key quantity, h, is defined as the water level in the left leg relative to the equilibrium level (i.e., the green line in Fig. 5b). At $t = 0$, $h = +5$ cm. (Note that h can turn negative at a later time.)

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 are:

(i) A description of the boundary conditions you choose for the two top openings that allow Fluent to properly simulate the oscillation.

(ii) A line plot of h (the water level in the left leg *relative to the equilibrium level*) as a function of time. Note that h 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. 5, 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?

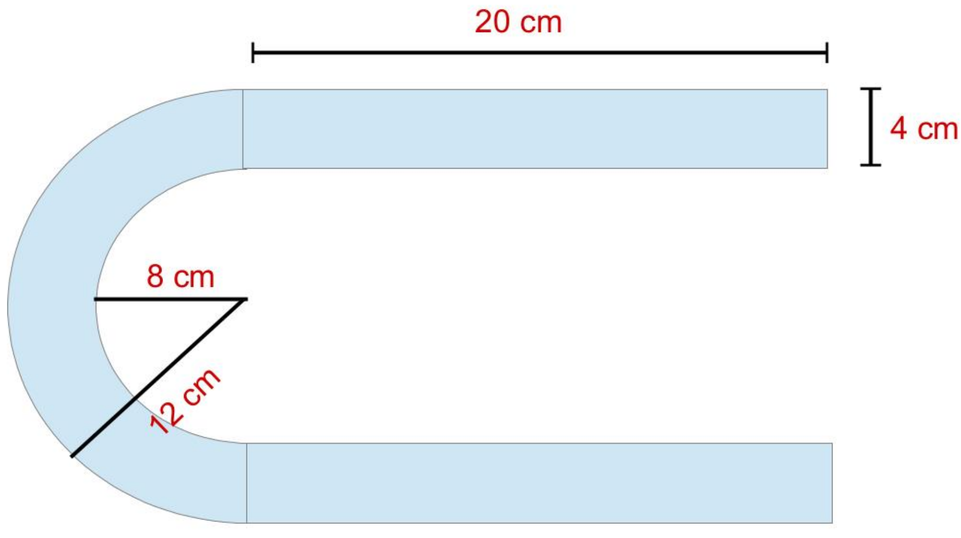

Fig. 4 The geometry of the U-pipe along the plane of symmetry. Not drawn to scale.

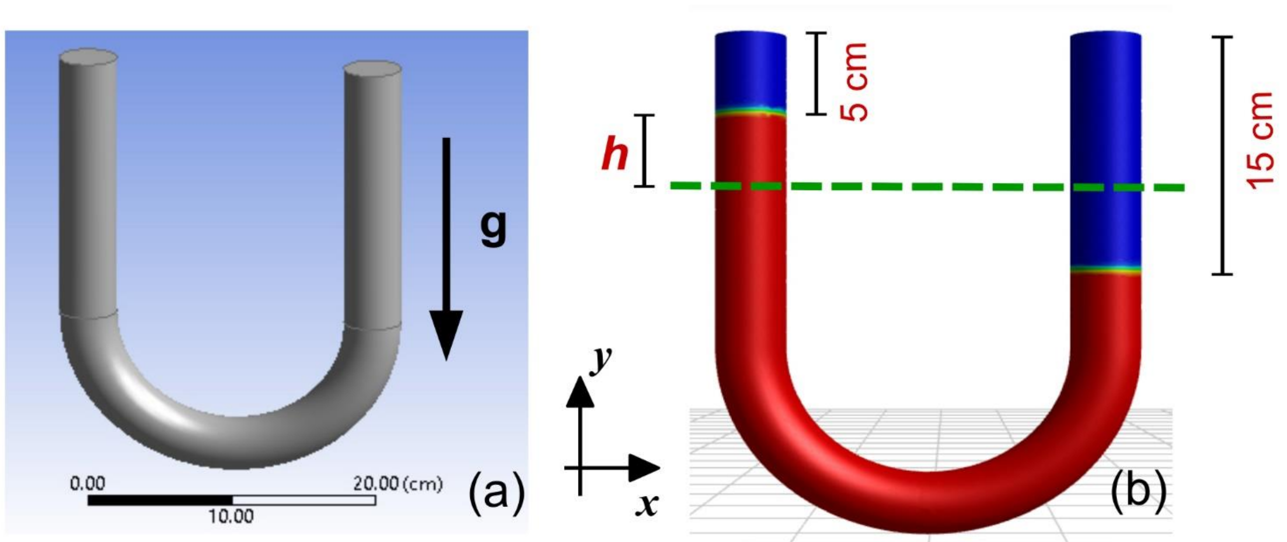

Fig. 5 (a) The geometry of the U-pipe. (b) The initial state for the simulation. Blue and red are air and water, respectively.