MAE 494/598 Applied CFD, Fall 2019, Project 3 – Multiphase flow (20 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 4 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. 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

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, as shown in Fig. 1. At $t = 0$, half of the bucket is filled with water (to the depth of 25 cm) and the rest with air. In addition, a circular droplet with diameter of 4 cm is placed in the middle of the bucket and at 15 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 located at $(x, y) = (25\text{ cm}, 40\text{ cm})$. Given the regular gravity with $g = -9.81\text{ m/s}^2$ in the vertical direction, at $t > 0$ the droplet will begin to fall and eventually impact the water surface. Performing a transient simulation using Ansys-Fluent, the key deliverables are contour plots of the *volume fraction of water* at $t = 0.15$ s, 0.25 s, 0.3 s, and 0.5 s. (Four separate plots.) It suffices to show the sub-domain that covers the droplet and the portion of water body and surface that has been deformed due to interaction with the droplet.

Additional detail: Set the top boundary as *pressure outlet* with zero gauge pressure, and with backflow phase set to *air*. At $t = 0$, 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.

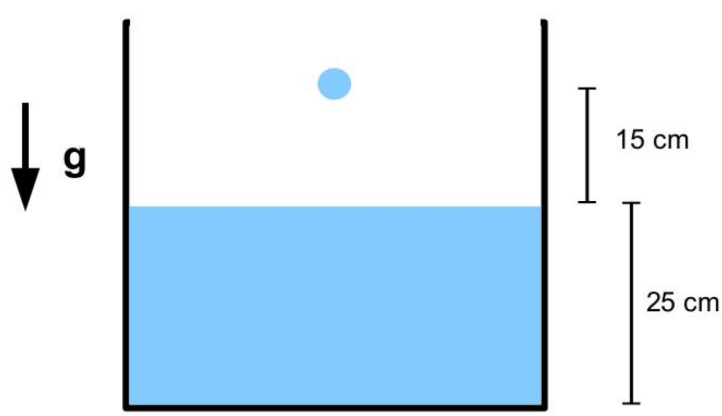

Fig. 1 The geometry and initial state used in Task 1.

Task 2

Consider a 3-D system with an inclined plate that forms a 40° 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 2.5 cm. The droplet is otherwise surrounded by open air. Regular gravity with $g = -9.81$ m/s^2 in the vertical direction applies. Figure 2(a) is the cross-sectional view of the system along the vertical plane that cuts through the center of the droplet. Figure 2(b) provides a 3D isometric view (explained below), with the bottom plate shown in blue. This task will use Ansys-Fluent with VOF method to simulate the evolution of the droplet in time. 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) Three plots in the fashion of Fig. 2b 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 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. 2b was made.

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

The report for this task should also include a description of the *computational domain*, *boundary conditions*, *mesh resolution*, and *time step size* used in the simulation.

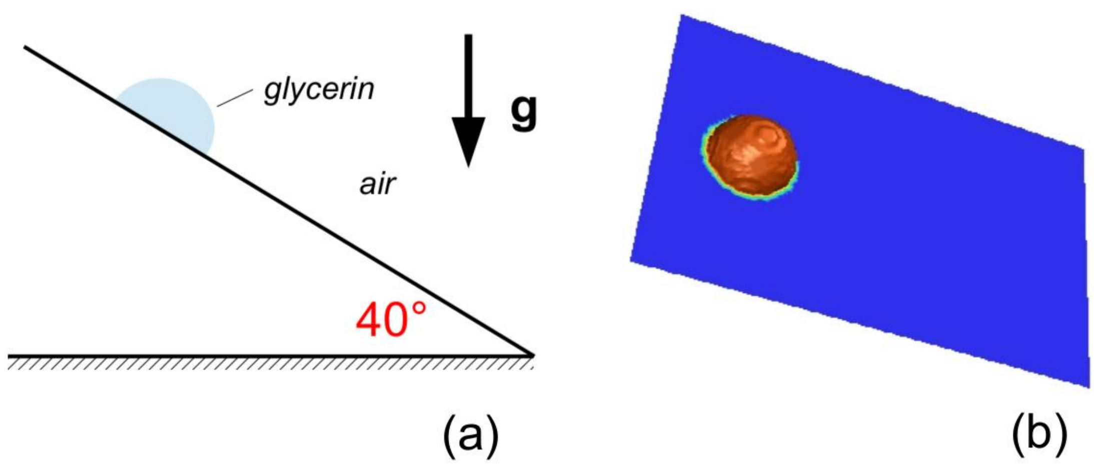

Fig. 2 The setup of the system at $t = 0$ for the simulation in Task 2.

Task 3

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. 3. 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. 3). Use turbulence k-epsilon model with default setting. Since both phases are gases, the effect of surface tension can be ignored.

(a) Case A: 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 = 50 Pa. Methane is pumped through boundary D into the domain. At $t = 0$, the entire domain, including the side pipe, is filled with *air*. Perform the simulation to $t = 10$ s. The deliverables are contour plots of the volume fraction of methane at $t = 7$ s and $t = 10$ s.

(b) Case B: 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 $u = 0.2\,y$, where u is in m/s and y in m. This gives a linear profile with $u = 0$ at the surface ($y = 0$) and $u = 10$ m/s at the top of the domain where $y = 50$ m. (See a sketch in Fig. 3.) Otherwise, the gauge pressure at A is still set to 0. At $t = 0$, the entire domain, including the side pipe, is filled with *air*. Perform the simulation to $t = 12$ s. The deliverables are contour plots of the volume fraction of methane at $t = 9$ s and $t = 12$ s.

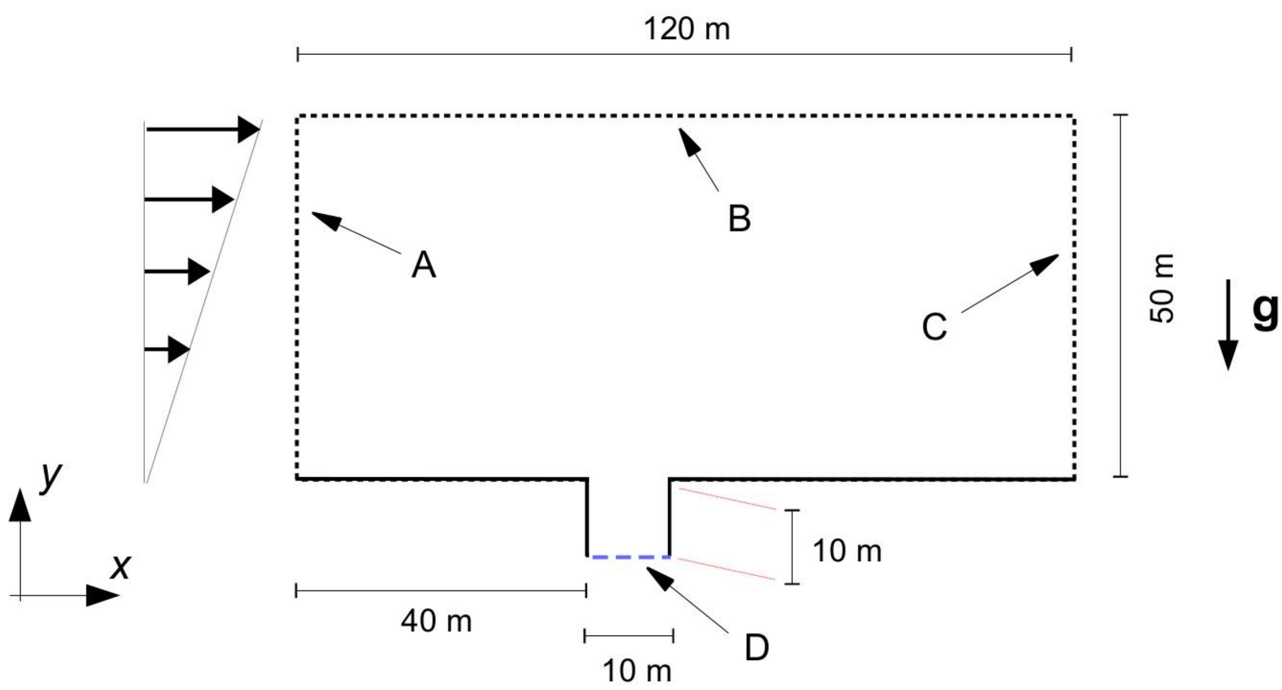

Fig. 3 The geometry of the computational domain for Task 3.

Task 4 (for MAE 598 only)

Consider a 2-D chamber with two side pipes as shown in Fig. 4. It has two openings marked by A and B. At $t = 0$, the chamber is filled with liquid alcohol to the depth of 15 cm. The rest of the chamber, including the two side pipes, is filled with air. Set A as a *velocity inlet* through which *water* is injected into the domain with 1 m/s inlet velocity. (The gauge pressure at A is set to 0.) Set B as a *pressure outlet* with zero gauge pressure and with backflow phase set to *air*. With this setting, run a transient simulation to $t = 0.4$ s. The deliverable is a contour plot of the *density* of the *mixture* at $t = 0.4$ s.

Additional detail: Use the default constant values of density and viscosity from Fluent database for air, water, and alcohol. (For alcohol, choose "ethyl-alcohol-liquid".) Use turbulence k-epsilon model with default setting. For this task, ignore the effect of surface tension.

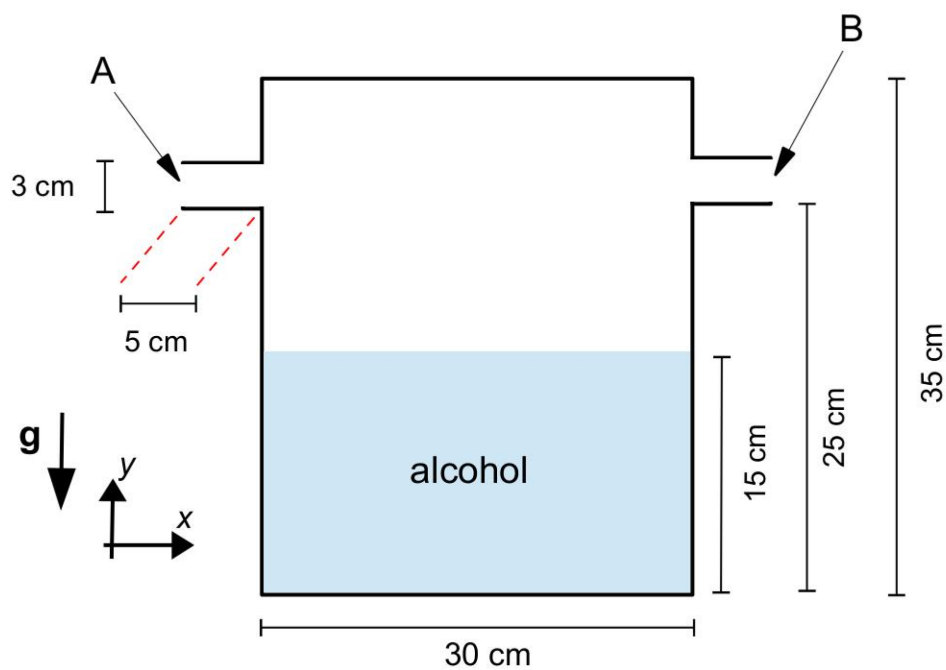

Fig. 4 The geometry of the chamber and the initial state for Task 4.