Project #3 Multi-phase flow

MAE 598/494 Applied CFD, Fall 2017, Project #3 (20 points)
Hard copy of report is due at the start of class on the due date. As usual, please follow the rules on collaboration. A cover sheet is required.

All tasks in this project are for *transient* simulations of multi-phase flows using Ansys-Fluent based on the VOF method. For all tasks, gravity is turned on with the direction of gravity indicated in the individual figures from Figs 1-5. All tasks, except Task 4b, are 2D. All tasks are for both MAE598 and MAE 494, except Task 4b which is for MAE598 only.

Task 1

(a) A 2-D chamber, illustrated in Fig. 1, consists of a main chamber and two symmetric side pipes for the inlet and outlet. All boundaries are solid walls except the *velocity inlet* at left and *pressure outlet* at right. Initially, the entire chamber including the side pipes is filled with *air*. At $t > 0$, *water* is injected through the inlet which produces a jet that gradually fills the chamber with water (as air is simultaneously pushed out of the chamber through the outlet). The key setups at the boundaries are: (i) *Volume fraction* = 1 for *water* at the *velocity inlet,* (ii) *Velocity* = 1 m/s for *mixture* at the *velocity inlet*, and (iii) *Gauge pressure* = 0 for *mixture* at both inlet and outlet (this is the default). Using viscous (turbulence) *k-epsilon model*, perform a transient simulation to $t = 0.4$ s. The deliverable of this task is a contour plot of the *volume fraction* of *water* at $t = 0.4$ s.

(b) Same as Task 1a, except that at $t = 0$ the chamber is already filled with liquid kerosene (using the properties from Fluent database) to the depth of 15 cm, as illustrated in Fig. 2. Using othewise the same setup as Task 1a, run the transient simulation to $t = 0.4$ s. The deliverables are two contour plots for (i) the total density of the mixture, and (ii) the volume fraction of water. (Note that there are 3 phases in this simulation. Be careful in setting up the volume fractions in the initial and boundary conditions.)

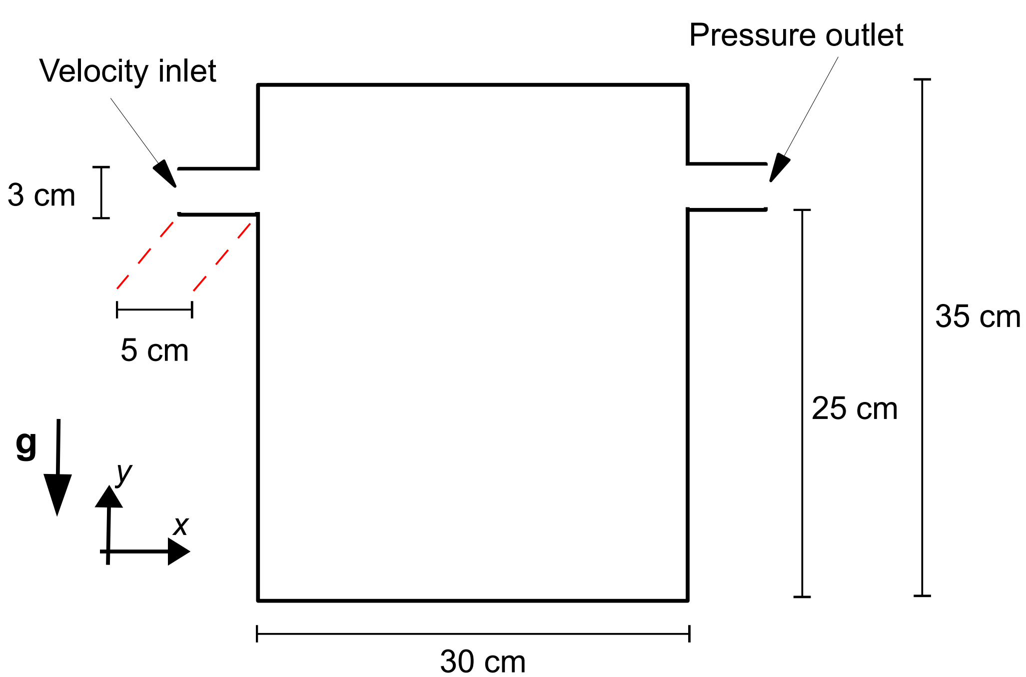

Fig. 1 The geometry of the system for Task 2.

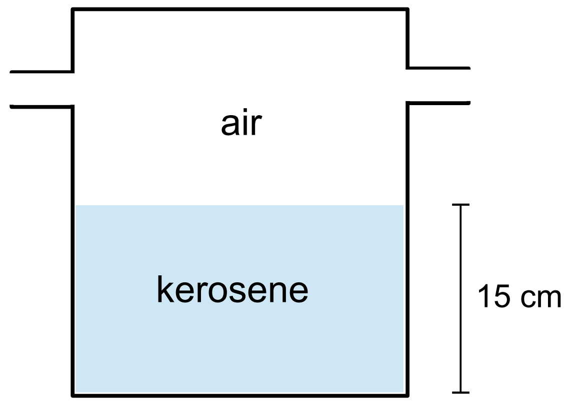

Fig. 2 The initial state for Task 1b.

Task 2

(a) This task emulates the situation when natural gas is leaked from an underground reservoir into a warehouse. The geometry of the (2D) warehouse is given in Fig. 3. The opening through which natural gas enters the building is marked by "A". Fresh air is pumped into the warehouse through the opening at left, marked by "B". An additional opening at top right, marked by "C", is a simple window which allows air or gas to pass through both ways. We nominally choose *methane* (CH_4) to represent natural gas. The density and viscosity (both set to constant) of methane can be copied from the existing database of Fluent.

Using the 2-D domain as shown in Fig. 3, the boundary conditions for the simulation are:

(i) Setting opening "A" as a *velocity inlet* with 2 m/s inlet velocity. Pure *methane* is pumped into the building through this inlet.

(ii) Setting opening "B" as a *velocity inlet* with 1 m/s inlet velocity. Pure *air* is pumped into the building through this inlet.

(iii) Setting opening "C" as a *pressure outlet*. All inlets and outlet have zero gauge pressure.

At $t = 0$, the entire building, including all "side pipes", is filled with *air*. Using viscous (turbulence) *k-epsilon* model, run a transient simulation to $t = 5$ s. The deliverables are contour plots for the *volume fraction* of *methane* at $t = 2$s and $t = 5$s.

(b) Repeat Task 2a but now increase the inlet velocity at inlet "B" to 5 m/s. The deliverables are contour plots for the *volume fraction* of *methane* at $t = 2$s and $t = 5$s. In addition, for this task only, make the contour plots for x-velocity and y-velocity at $t = 5$s. (The directions of x and y are as indicated in Fig. 3.)

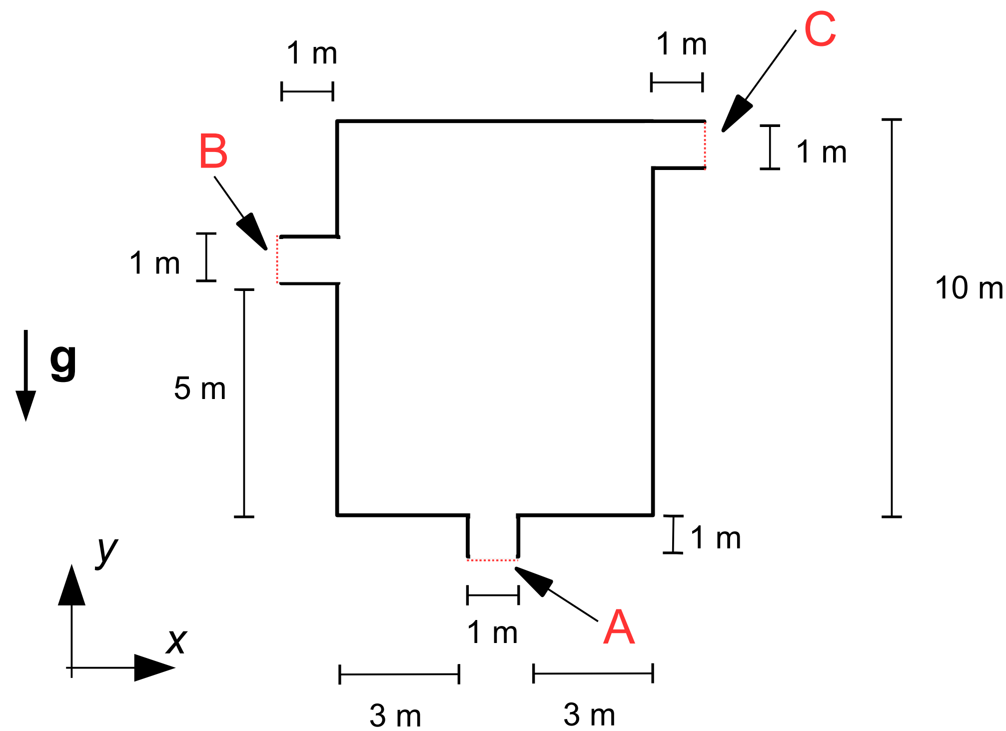

Fig. 3 The geometry of the system for Task 2.

Task 3

Background: Consider a 2-D system that consists of two identical open containers connected in the bottom by a short pipe, as shown in Fig. 4. Initially, the water level in the left container is 35 cm while that in the right container is only 15 cm. There is no motion (in air or water) in the entire domain at $t = 0$. This task asks one to simulate the evolution of water levels in the two containers with time. As the system evolves in time, we expect an oscillation of the water levels. When the water level decreases in the left container, it drives air to flow into that container through the top opening. Simultaneously, the water level in the right container increases, pushing air out of the container through the top opening. In the "reverse phase" of the oscillation (when the water level increases in the left container), we expect a reversal of the direction of air flow through each of the top openings. Given so, the boundary conditions for the top openings of both containers must be set such that air is allowed to flow through the openings both ways.

(a) Use *viscous-laminar* model to simulate the oscillation described in *Background*. Specifically, the boundary conditions at the two top openings (indicated by "A" and "B" in Fig. 4) should be set up such that air can flow through those openings in both directions as a passive response to the oscillation of the water body in the system. Extend the simulation to at least two cycles of oscillation. (The first cycle is completed when the water level of the left container is back to maximum, and so on.) The deliverables for this part are:

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

(ii) A line plot of the water levels of the left and right containers as a function of time. The "water level" here is the depth of water in the container. If the air-water interface is not strictly horizontal, use the averaged depth. Please collect the two curves of the water levels of the left and right containers in a single plot. Determine the approximate period of the oscillation. Determine the specific times when the water levels of the two containers become equal. As a reference for deliverable (iii), we define t_1 and t_2 as the first and second time the water levels of the two containers become equal.

(iii) Plots of the velocity vector field (in the fashion of Fig. 1.7 in Tutorial #2) at $t = t_1$ and $t = t_2$.

(iv) A contour plot of the volume fraction of water at the time when the water level of the right container peaks for the first time. (This is the time when the system completes a half cycle of oscillation from the initial time.)

(b) Repeat Task 3a but now run the simulation using *invicid* model. For this task, you only need to provide deliverables (i) and (ii) from Task 3a. In addition, briefly explain the difference between the results produced by *viscous-laminar* and *invicid* model.

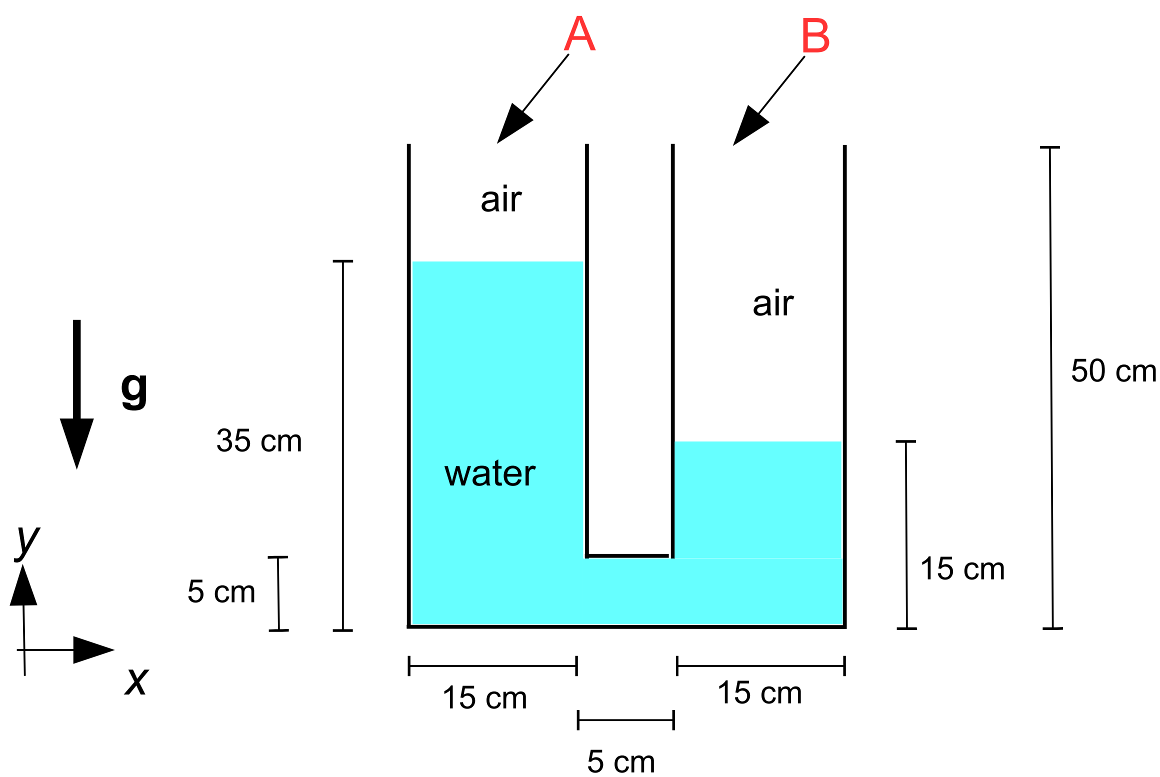

Fig. 4 The apparatus considered in Task 3. The water levels as shown are for the initial time.

Task 4

(a) Consider a 2-D system with an inclined plate that forms a 30° angle with the ground as shown in Fig. 5. At $t = 0$, a drop of *engine oil* is placed on the plate and it is shaped like a semicircle with a radius of 3 cm. Use Ansys-Fluent to simulate the evolution of this drop and show the shape of the blob of engine oil at $t = 0$, 0.1 s, and 0.3 s. (It suffices to show the contour plots of the *volume fraction* of *engine oil* at those times.) The properties of engine oil can be copied from the existing database of Fluent.

(b) [for MAE 598 only] Consider the 3-D version of the system in Task 4a with the initial drop of engine oil shaped like a *hemisphere* with a radius of 3 cm. Use Ansys-Fluent to simulate the evolution of this drop and show the shape of the blob of engine oil at $t = 0$, 0.1 s, and 0.3 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 engine oil.)

Note: We recommend the use of *viscous-laminar* model for this problem. In the physical system, the drop of engine oil 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 associated mesh) and boundary conditions. This information (your choices of the domain, mesh, and boundary conditions) should be included in the report. For the 3-D simulation in Task 4b, we do not recommend the "symmetry" set up. Please run the simulation using the full geometry.

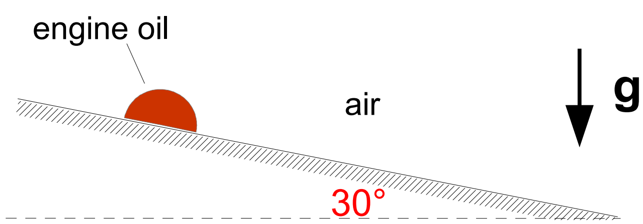

Fig. 5 The initial state of the system considered in Task 4.