MAE 560/460 Applied CFD, Fall 2022
Project 2 – Multiphase flow; Flow with interface (27 points)

Please follow the rules on collaboration as described in the first page of the document for Project 1. A Statement of Collaboration is required.

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.* 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.

For Task 1, set *operating density method* to *mixture-averaged*. For all other tasks, set it to *minimum-phase-averaged* (which is the default). [Note: In older (pre-2020) versions of Ansys, the default is "*not set*", which is equivalent to *mixture-averaged* in the current version of Ansys.]

Task 1

In this task, we simulate the leaking of natural gas from an underground vault into open air, in a pure 2-D setting. The computational domain is as shown in Fig. 1. A pipe at bottom 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-omega* model with default setting. Since both phases are gases, the effect of surface tension can be ignored.

(a) Case A: Set the two boundaries marked by A and C to *pressure outlet* with zero gauge pressure. Both are open to *air*. The top boundary marked by B is set to *wall* but with *zero shear stress*. (This emulates a "frictionless wall.") The boundary marked by D is set as a *pressure inlet* with gauge pressure = 50 Pa. *Methane* is pumped through inlet D into the domain. (All other unnamed boundaries are regular wall with no-slip boundary condition, i.e., the default setting for wall.) At t = 0, fill the entire domain (including the bottom pipe) with *air*. Initialize the system with gauge pressure = 0 and velocity = 0. For the turbulence parameters, initialize *turbulence kinetic energy* (k) = 1 m^2s^{-2}, and *specific dissipation rate* (*omega*) = 1 s^{-1}. Perform the transient simulation to t = 6 s. The deliverables are:

(D1) Contour plots of the *volume fraction of methane* at t = 3 s and t = 6 s.

(b) Case B: Use the same setting as Case A except that the gauge pressure at inlet D is increased to 150 Pa. In addition, the left boundary marked by A is replaced by a *velocity inlet*, with an imposed velocity profile for the x-velocity given by $u = 0.6\,y - 0.02\,y^2$ where u is in m/s and y in m. (Otherwise, gauge pressure is set to zero at this inlet.) This gives a parabolic profile with u = 0 at the ground (y = 0) and top of the domain (y = 30 m), and u attains the maximum of 4.5 m/s at y = 15 m. (See a sketch in Fig. 1.) *Air* is pumped into the domain through inlet A. At t = 0, initialize the system in the same way as Case I. Perform the transient simulation to t = 6 s. The deliverable is

(D2) A contour plot of the *volume fraction of methane* at t = 6 s.

(D3) As an additional deliverable for both Task 1a and 1b, describe the *mesh resolution*, *time step size*, and *maximum number of iterations per time step*, used in the simulations in (a) and (b). (If different settings are used for the two cases, please indicate so.)

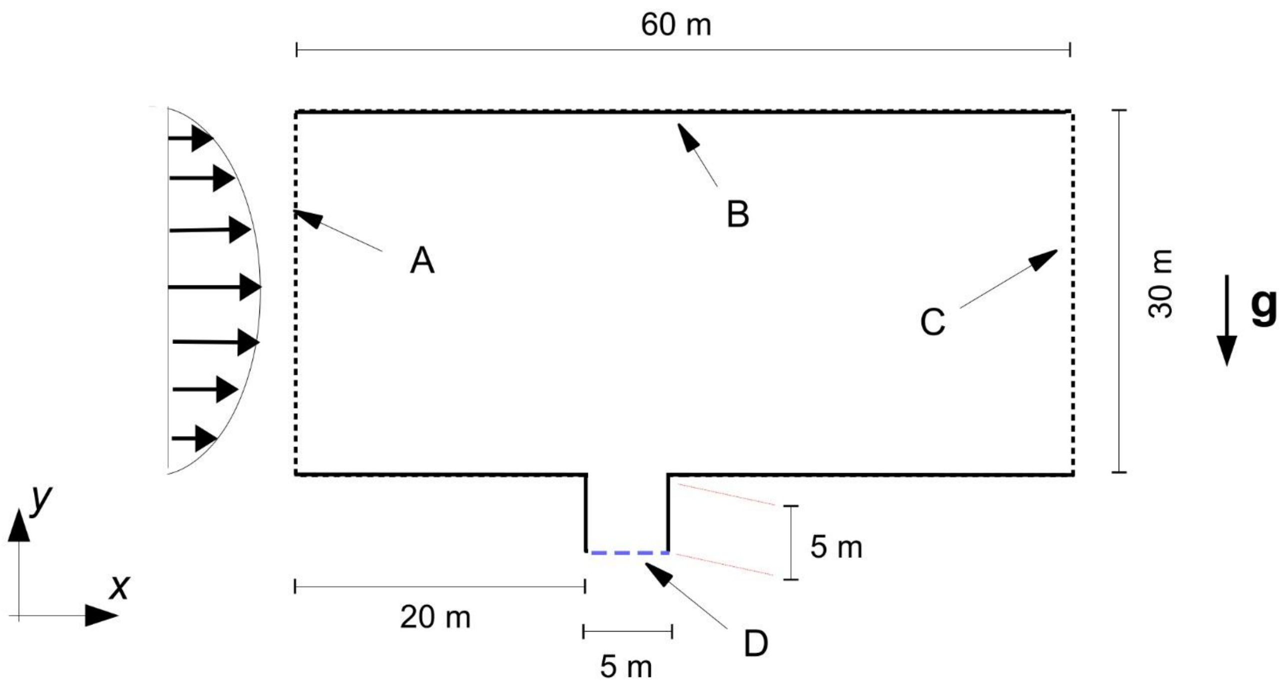

Fig. 1 The geometry of the computational domain for Task 1.

Task 2

In this task, we simulate the process of a falling *water* droplet colliding with a flat surface of *liquid diesel*, in a pure 2-D setting. The geometry of the system is a simple 25 cm x 25 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 *liquid diesel* (*"diesel-liquid"* in Fluent database) to the depth of 8 cm, and the rest filled with *air*. (See Fig. 2.) In addition, a circular droplet with *radius* of 1.5 cm (*diameter* of 3 cm) is placed in the middle of the bucket, with its center placed 20 cm above the floor. More precisely, if the coordinate of the lower-left corner of the bucket is $(x, y) = (0,0)$, the center of the droplet is $(x, y) = (12.5 \text{ cm}, 20 \text{ cm})$.

For the transient simulation, set gravity to the regular $g = -9.81$ m/s^2 in the y-direction as indicated in Fig. 2. Initialize the system with zero gauge pressure and zero velocity. Use *Laminar* model and turn on *surface tension* modeling. For the interface between water and air, use the default value of surface tension coefficient (under liquid water) from Fluent database. Otherwise, ignore surface tension effect for the interface between water and liquid diesel, and between liquid diesel and air. At $t > 0$, the water droplet will begin to fall and eventually impact the surface of liquid diesel. Perform the transient simulation to $t = 0.2$ s.

Since the simulation involves 3 phases, we need to find a way to display all of them in the same plot. For example, consider the *custom field function* defined by

$$\boldsymbol{CF} = 0.1*VF1 + 0.2*VF2 + 0.3*VF3 \,,$$

where *VF1*, *VF2*, and *VF3* are the volume fractions of the 3 phases. With this definition, ***CF*** will be 0.1 over the region covered by Phase 1, 0.2 over the region covered by Phase 2, and so on. A contour plot of ***CF*** will show 3 different colors representing the 3 phases. Note that many variations of the definition of ***CF*** will work. It suffices that the VFs of the 3 phases are multiplied by different numbers. (For example, ***CF*** = 1**VF1*+3**VF2*+5**VF3* will also work. The detail is your choice.)

The deliverables are:

(D4) A description of the *mesh resolution*, *time step size*, and *maximum number of iterations per time step*, used in the simulation.

(D5) Contour plots of the above-mentioned *custom field function* ***CF*** at t = 0.1 s, 0.16 s, and 0.2 s. (Three separate plots.)

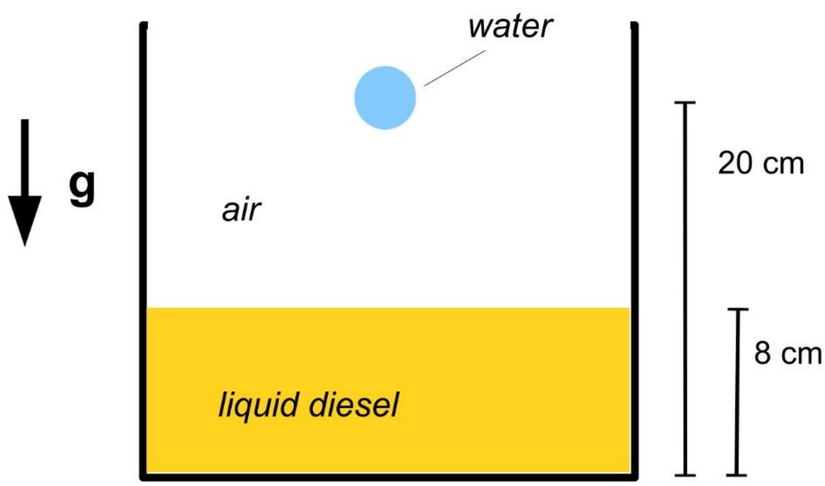

Fig. 2 The geometry and initial state used in Task 2. Not drawn to scale.

Task 3

Consider a 3-D system with an inclined plate that forms a 30° 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 cm (*diameter* of 4 cm). The droplet is surrounded by open air. Gravity is the regular $g = -9.81$ m/s^2 in the vertical direction. Figure 3a shows the cross-sectional view of the system along the vertical plane that cuts through the center of the droplet. Figure 3b is the 3-D isometric view (explained below) of the initial state of the system, with the bottom plate shown in blue. This task will simulate the temporal evolution of the droplet. In the actual physical system, the droplet of *glycerin* is surrounded by open air without top or side "wall" boundaries. In the numerical simulation, one needs to set a computational domain and specify appropriate boundary conditions to emulate this physical condition. Appropriate mesh resolution and time step size should be used to ensure that the result is robust. Use *Laminar* model and use the default constant values for the density and viscosity of *glycerin* and *air*. Turn on surface tension modeling and set surface tension coefficient for the interaction between *glycerin* and *air* to a constant of 0.06 N/m.

Perform a transient simulation to $t = 0.2$ s. The deliverables are

(D6) A description of the *computational domain*, *boundary conditions*, *mesh resolution*, *time step size,* and *maximum number of iteration per time step* used in the simulation.

(D7) Three plots in the fashion of Fig. 3b that show the 3-D shape of the blob of *glycerin* at $t = 0$, 0.08 s, and 0.2 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 = 0.95 where VF is the volume fraction of *glycerin*. This is how Fig. 3b was made. [We recommend that the bottom plate is also shown, in the fashion of Fig. 3b. This will help demonstrate how far the blob has advanced down the slope at different times.]

(D8) Three contour plots of the *volume fraction* of *glycerin* on the plane of symmetry, at $t = 0$, 0.08 s, and 0.2 s.

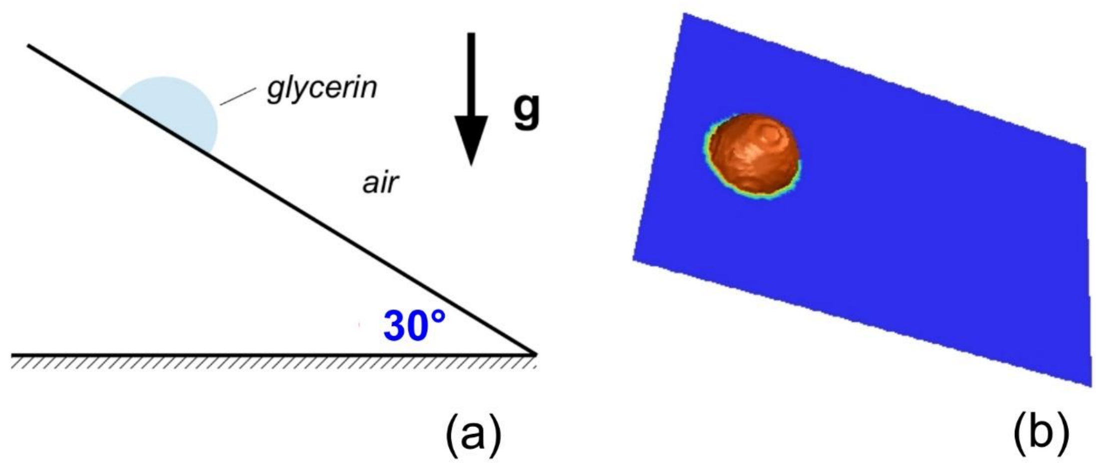

Fig. 3 The setup of the system at $t = 0$ for the simulation in Task 3.

Task 4 (For MAE560 only)

For simplicity, we turn off surface tension for this task. Use *k-omega* model with default setting.

Consider a prototype of a kitchen sink that consists of a hemispherical basin with a *radius* of 25 cm, and a cylindrical drainage pipe at bottom that is 5 cm long with a *radius* of 2 cm. The 3D view of the system is shown in Fig. 4a, and a cross-sectional view along the plane of symmetry in Fig. 4b. The top of the basin and bottom of the drainage pipe are both open to *air*. Gravity is the regular $g = -9.81$ m/s^2 in the vertical direction as indicated in Fig. 4a. This task includes two simulations:

Case A: Consider an initial state with the kitchen sink (including the bottom pipe) filled with *water* to the level of 5 cm from the top, as shown in Fig. 4b (water and air are red and blue, respectively). From this initial state, perform a transient simulation to the time, $t = t_A$, at which half of the *water* in the initial state is drained out of the system.

Case B: Repeat the setting in Case A but replace *water* with *engine oil* (using the physical properties from Fluent database.) Run the simulation to the time, $t = t_B$, at which half of the *engine oil* is drained out of the system.

The deliverables are:

(D9) A description of the *boundary conditions* at top and bottom openings, *mesh resolution*, *time step size*, and *maximum number of iterations per step* used in the simulations. (For a fair comparison in D10, those parameters should be the same for case A and B.)

(D10) The values of t_A and t_B as determined from the two simulations. Please write the value to first digit accuracy (i.e., in increment of 0.1 s). (For example, 12.7 s should not be truncated to 12 s or 13 s.) Which of *water* and *engine oil* drains faster?

(D11) For Case B and at $t = t_B$, a plot of *velocity vectors* on the horizontal plane that is 15 cm below the top of the basin (see Fig. 4c). [Note: Using the default setting for vector plot in Fluent will produce too many vectors so as to obscure the key structure of the flow field we wish to see. Please make appropriate adjustments ("skip" some vectors and rescale the rest) to improve clarity of the plot.]

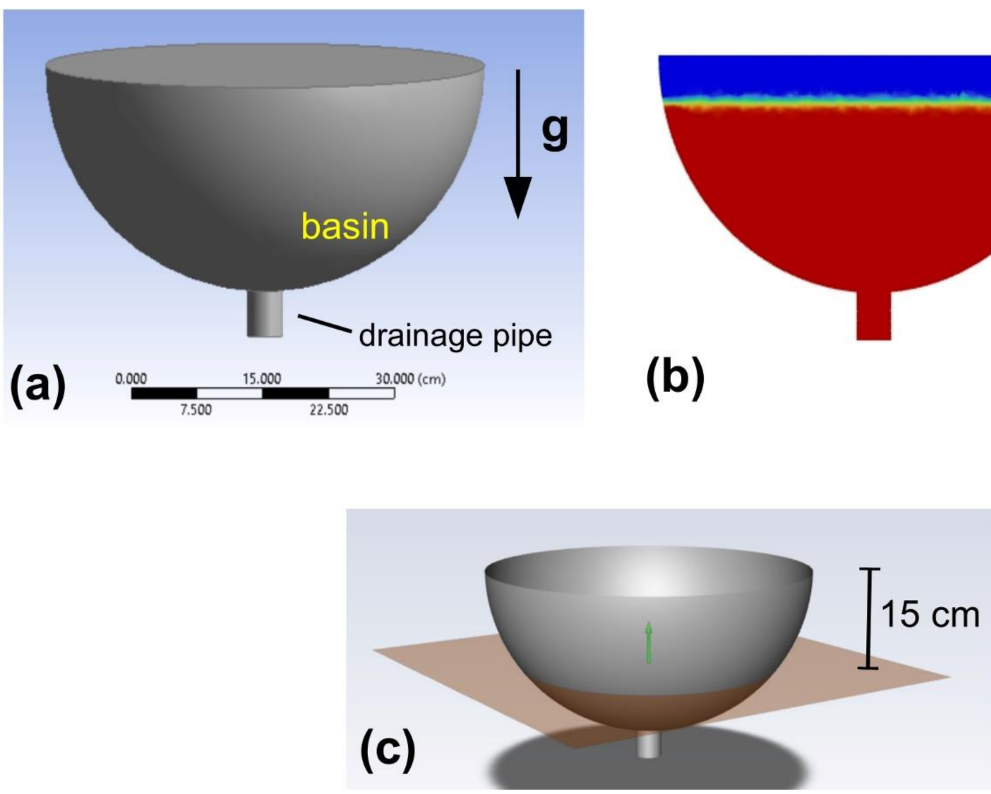

Fig. 4 (a) The geometry of the kitchen sink used for Task 4. (b) Cross-sectional view with the setting for the initial condition. (c) The horizontal plane used for deliverable D11.