MAE 598/494 Applied CFD, Fall 2015, Project #2 (20 points)
Please follow the rules for collaboration and use a properly filled Cover Sheet for the report. Hard copy of report is due before the start of class on the due date.

All tasks in this project are for *transient* simulations of multi-phase flows using Ansys-Fluent based on the VOF method. All cases are 2-D in the x-y plane, with gravity set to $g = -9.8$ m/s² in y-direction. All tasks use *Multiphase model* with *Volume of Fluid method* and with *Energy equation turned off*. In all cases, at the minimum please set "Sizing → Relevance center" to *fine* to generate the mesh. Further mesh refinements, local or global, are optional but encouraged. For other detailed setups for the solution method please refer to Tutorial 3A and 3B. The task marked by a triangle (▲) is for MAE 598 only. Work submitted by an MAE 494 student for that task will not be graded and will not be awarded any point.

Task 1
Consider a closed rectangular domain with wall at the boundary as illustrated in Fig. 1. Initially (at $t = 0$), the chamber is filled with *engine oil* except that a square subdomain in the lower left corner is filled with *water*. The density and viscosity (both set to constant) of each phase of fluid can be copied from the existing database in Fluent.

(a) Use viscous (turbulence) *k-epsilon model* to perform transient simulation and make the contour plots of the *volume fraction* of *engine oil* (if engine oil is defined as phase-1, this would be the volume fraction of phase-1) for the solution at $t = 2$ s, 6 s, and 10 s.

(b) If $x^*(t)$ denotes the position of the leading edge of water in x-direction (see Fig. 2) as a function of t, make a plot of $x^*(t)$ vs. t using the data points of $x^*(t)$ at $t = 0.5$ s, 1 s, 1.5 s, 2 s, and 2.5 s as obtained from the simulation in Part (a). Perform a linear least-square fit of the 5 data points. The "slope" from the linear fit is an estimate of the average velocity, U, of the leading edge of water. What value of U (in m/s) do you obtain? What are the Δx (in m) and Δt (in s) that you used in the simulation in Part (a)? Based on the values of U, Δx, and Δt, what is the Courant number, $C \sim U\Delta t/\Delta x$? Is it less than 1 as is generally desired for maintaining numerical stability?

(c) Repeat Part (a) but now use the *inviscid model*. Plot the solution at $t = 2$ s, 6 s, and 10 s in the same fashion as Part (a). Discuss the differences between the two runs in Part (a) and (c).
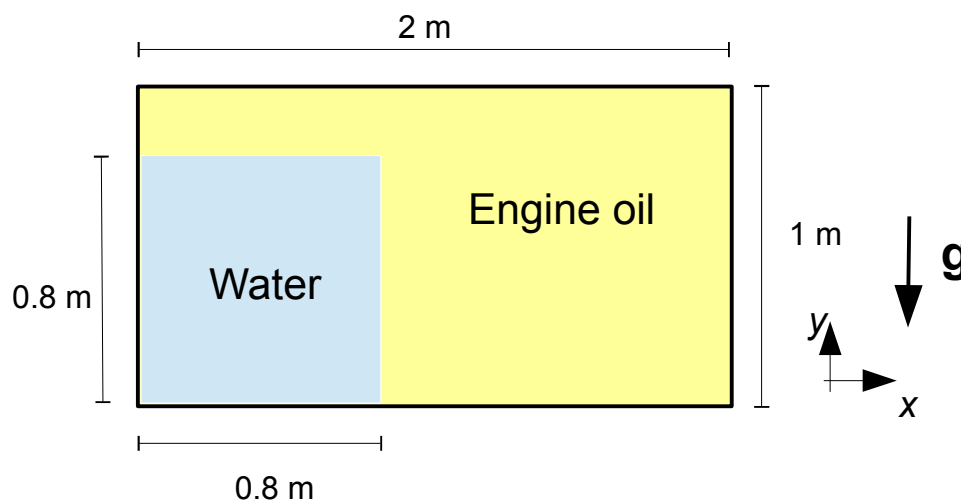

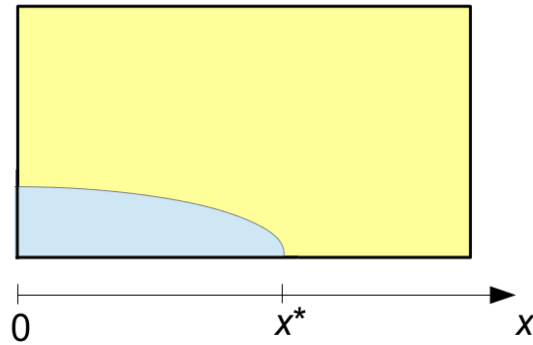**Fig. 1** The geometry and initial distribution of the phases for Task 1.

Fig. 2 The definition of x*, the x-position of the leading edge of water.

Task 2

A 2-D chamber, illustrated in Fig. 3, consists of a main chamber and two symmetric side pipes for the inlet and outlet. All boundaries are solid wall 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). Define air as *phase-1* (the *primary phase*) and water *phase-2*, the key setups at the boundaries are:

(i) *Volume fraction* = 1 for *phase-2* at the *velocity inlet*
(ii) *Velocity* = a given V_{inlet} (see below) for *mixture* at the *velocity inlet*
(iii) Gauge pressure = 0 for *mixture* at both inlet and outlet (this is the default anyway)

Using viscous (turbulence) *k-epsilon model*, perform transient simulations for the two cases with (a) V_{inlet} = 0.25 m/s, and (b) V_{inlet} = 0.5 m/s. For each case, make the contour plot of the *volume fraction* of *phase-1* (i.e., *air*) for the solution at $t = 1$ s.
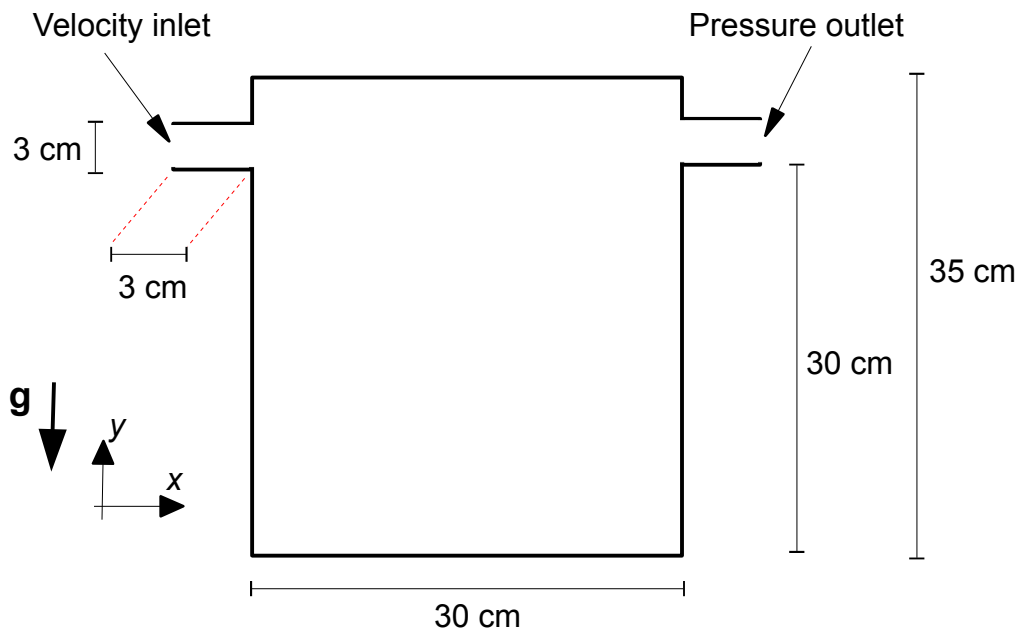

Fig. 3 The geometry of the system for Task 2.

Task 3

This case emulates the situation when natural gas is released into open air from an underground reservoir. We nominally choose *methane* (CH4) to represent natural gas. The density and viscosity (both set to constant) of methane can be copied from the existing database in Fluent.

The 2-D domain for the simulation is shown in Fig. 4. The bottom of the domain is wall except the opening at *velocity-inlet-1*, through which methane will be injected into the domain. The whole left boundary is *velocity-inlet-2* through which normal air will be blown in. The whole top boundary is *pressure-outlet-1*, and right boundary *pressure-outlet-2*. Define air as *phase-1* (the primary phase) and methane as *phase-2*, the key setups at the boundaries are:

(i) *Volume fraction* = 1 for *phase-2* at *velocity-inlet-1*
(ii) *Velocity* = 5 m/s for *mixture* at *velocity-inlet-1*
(iii) *Volume fraction* = 0 for *phase-2* at *velocity-inlet-2*
(iv) *Velocity* = a given V_{inlet} (see below) for *mixture* at *velocity-inlet-2*
(v) Gauge pressure = 0 for *mixture* at all inlets and outlets

Using viscous (turbulence) *k-epsilon model*, run two simulations with (a) V_{inlet} = 0.1 m/s, and (b) V_{inlet} = 3 m/s for *velocity-inlet-2*. For both cases, the inlet velocity at *velocity-inlet-1* (at bottom) is fixed at 5 m/s. For each case, make the contour plots of the *volume fraction* of *phase-1* for the solution at t = 10 s and 20 s. (Two plots for each case.)
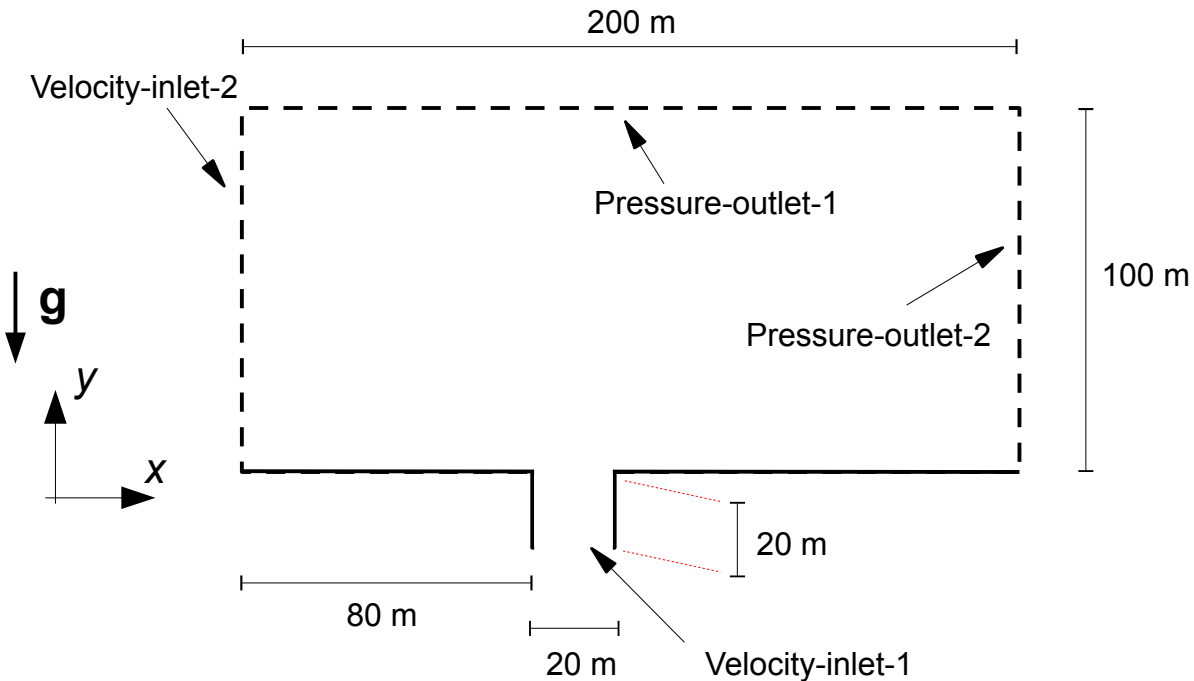

Fig. 4 The geometry of the system for Task 3.

▲ Task 4 (MAE 598 only)

Consider a closed 2-D rectangular chamber identical to that used in Task 1. Initially, the chamber is filled with *air*, (as *phase-1* and *primary phase*) except that a rectangular subdomain in the lower left corner is filled with *water* (as phase-2) and another rectangle in the lower right corner filled with *engine oil* (as phase-3). (See Fig. 5.) As a preparation for this task, note that a plot of the *volume fraction* of *phase-1* would not properly show the distribution of the three phases, since the areas covered by both *phase-2* and *phase-3* will have zero values. A customized variable is needed for displaying the three phases together. For example, using *Custom Field Function* (see Tutorial #2), if one defines

$$CF = 0.1*VF1 + 0.2*VF2 + 0.3*VF3 ,$$

where VF*n* is the volume fraction of the *n*-th phase, the contour plot of **CF** will have the values of 0.1, 0.2, and 0.3 for the areas covered by *phase-1*, *phase-2*, and *phase-3*. (A minor compromise is that the interface between *phase-1* and *phase-3* would show the color of *phase-2*. Also, while one could possibly show the contours of *density* of *mixture* as an alternative, the plot might not look good due to the large difference in density between air and the two liquid phases.)

Using viscous (turbulence) *k-epsilon model*, run a transient simulation and make contour plots of the variable **CF** as defined above, for the solution at $t = 0$, 1 s, 4 s, and 8 s. We ask that the initial state (at $t = 0$) be plotted to demonstrate that the initial setup is correct. Briefly discuss the result.

Hint: For this task, both phase-2 and phase-3 need to be patched for both subdomains. In other words, one needs to do "patch" 4 times to properly set up the initial condition.
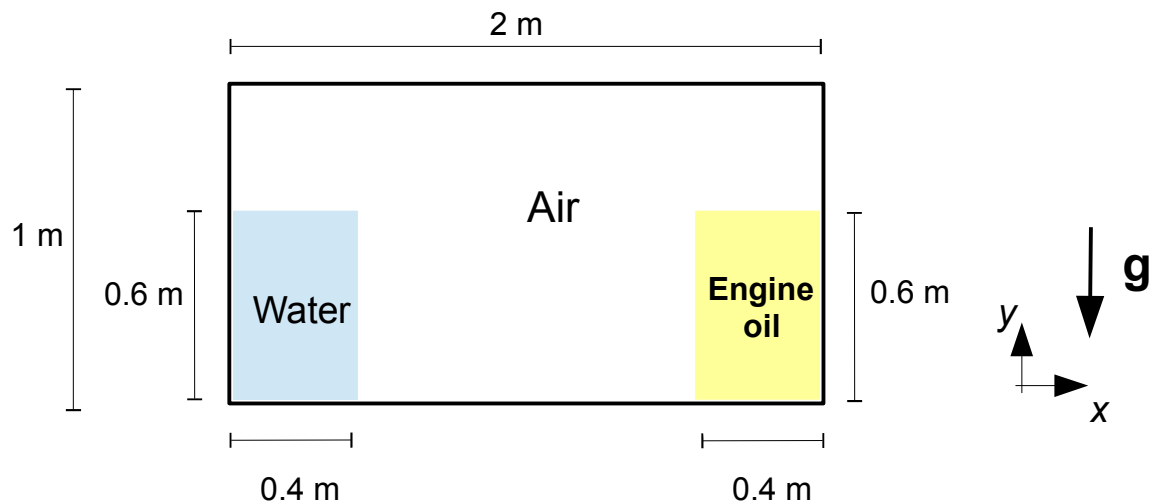

Fig. 5 The geometry and initial distribution of the phases for Task 4.