MAE 598/494 Applied CFD, Fall 2016, Project #2 (regular tasks = 20 points)
Please follow the rules for collaboration and use a properly filled Cover Sheet for the report. Hard copy of report is due at 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, except Part (b) of Task 4, are 2-D in the x-y plane, with gravity set to $g = -9.8$ m/s^2 in y-direction. (Thus, x is always the horizontal direction and y the vertical 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.

Part I. Regular tasks for MAE 598 and MAE 494

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 *liquid kerosene* except that a square subdomain in the lower left corner is filled with *engine oil*. 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-laminar* model to perform a transient simulation and make the contour plots of the *volume fraction* of *engine oil* (if engine oil is defined as phase-2, this would be the volume fraction of phase-2) for the solution at $t = 1$ s, 5 s, and 10 s.

(b) Repeat the task in (a) but run the simulation using *inviscid* model instead.

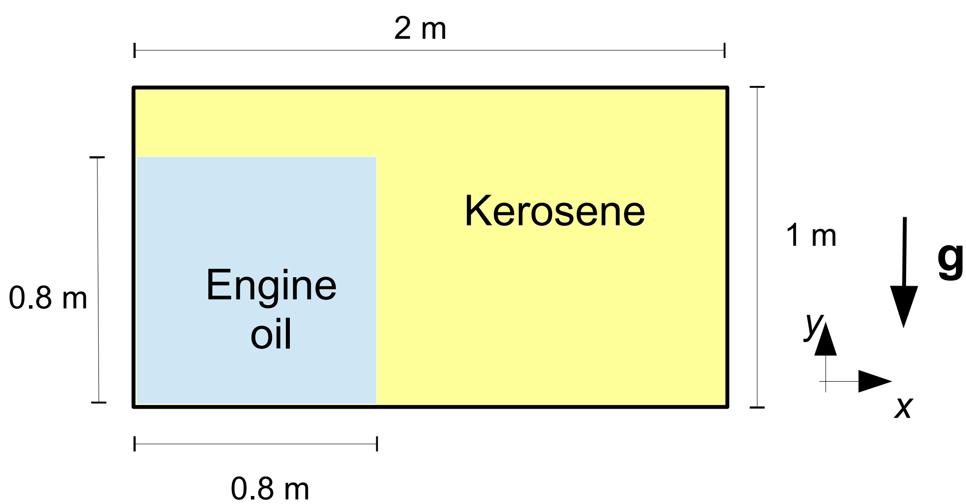

Fig. 1 The geometry and initial distribution of the phases for Task 1.

(c) For the system considered in part (a) and (b), the total potential energy, *PE*, and total kinetic energy, *KE*, are defined by

$$PE = \int \int \rho \; g \; y \; dx \; dy \quad , \qquad \text{Eq. (1)}$$

$$KE = \int \int \frac{1}{2} \rho (u^2 + v^2) \; dx \; dy \quad , \qquad \text{Eq. (2)}$$

where ρ is density and the double integral is over the entire rectangular domain. We define the "*baseline potential energy*", PE_0, as the value of PE for the terminal state (shown in Fig. 1A), i.e., the solution for the viscous case as $t \rightarrow \infty$ when the system would not move further. In other words, PE_0 is the portion of PE that is not available for conversion to KE. As such, we define the "*available potential energy*" as

$$APE \equiv PE - PE_0 \ . \qquad \text{Eq. (3)}$$

As the system evolves in time, we expect exchanges between APE and KE. For the viscous case, upon reaching the terminal state, both APE and KE will approach zero. From the simulations in part (a) and (b), plot APE, KE, and $APE+KE$ as a function of time (please collect the three curves in one plot) for the two cases using *viscous-laminar* and *inviscid* model. Each curve in the plot should cover at least the range of $0 \leq t < 25$ s and have a temporal resolution of at least 0.1 s. (In other words, each curve will consist of at least 250 data points.) Based on the plots, discuss the effect of viscosity on the behavior of the solution.

[Note: Since we consider a 2-D system, the "energy" in Task 1(c) has the unit of J/m instead of J. For example, if we fill the whole rectangular chamber in Fig. 1 with water, and assuming that the density of water is 1000 kg/m^3, the PE of the system will be 9800 J/m.]

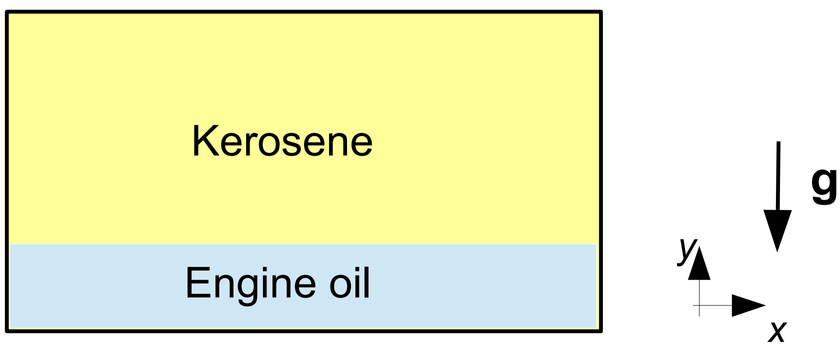

Fig. 1A The terminal state for the viscous case. The potential energy of this state is the "*baseline potential energy*", PE_0.

Task 2

A 2-D chamber, illustrated in Fig. 2, 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). 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:
(a) V_{inlet} = 0.3 m/s. For this case, make the contour plot of the *volume fraction* of *phase-2* (i.e., *water*)

for the solution at $t = 2$ s, 4 s, and 6 s.
(b) $V_{\text{inlet}} = 0.6$ m/s. For this case, make the contour plot of the *volume fraction* of *phase-2* (i.e., *water*) for the solution at $t = 1$ s, 2 s, and 3 s.

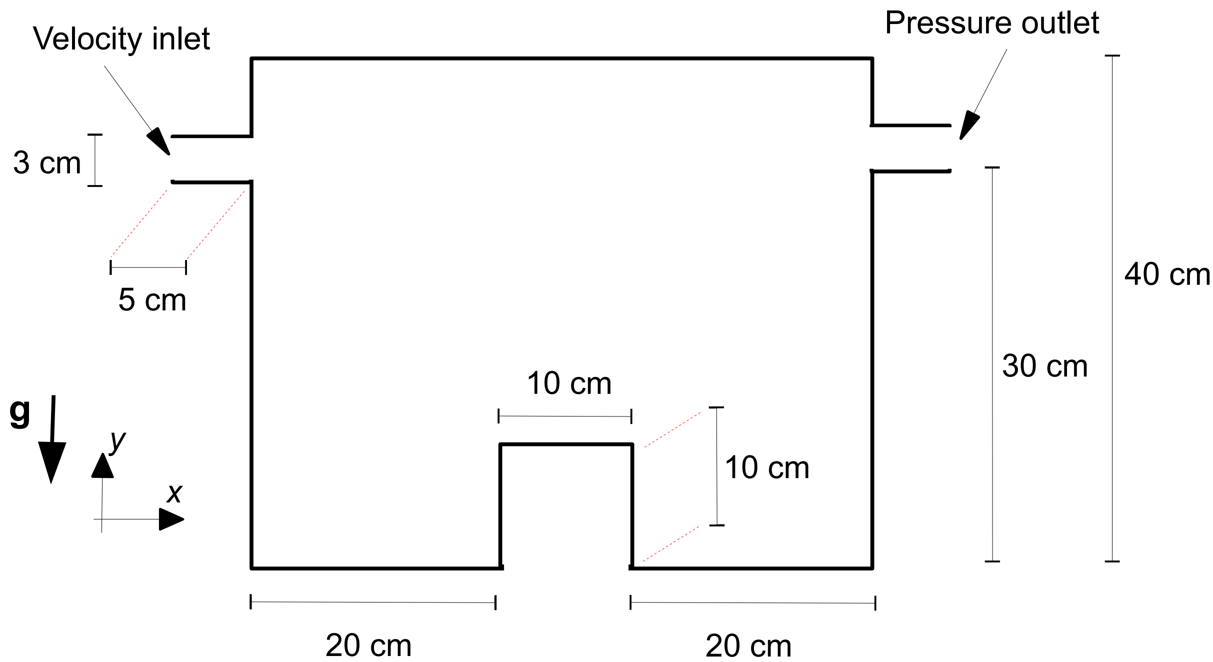

Fig. 3 The geometry of the system for Task 2.

Task 3
This case emulates the situation when natural gas is released into open air from an underground reservoir. We nominally choose *methane* (CH_4) 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. 3. The bottom of the domain is wall except the opening at *velocity-inlet-1*, through which methane is 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 *outlet-1*, and right boundary *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. (This is the default)
(vi) The boundary conditions at both *outlet-1* and *outlet-2* are set to *outflow.*

Using viscous (turbulence) *k-epsilon model*, run two simulations with (a) $V_{\text{inlet}} = 0.2$ m/s, and (b) $V_{\text{inlet}} = 2$ 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-2* for the solution at $t = 5$ s and 10 s. (Two plots for each case.)

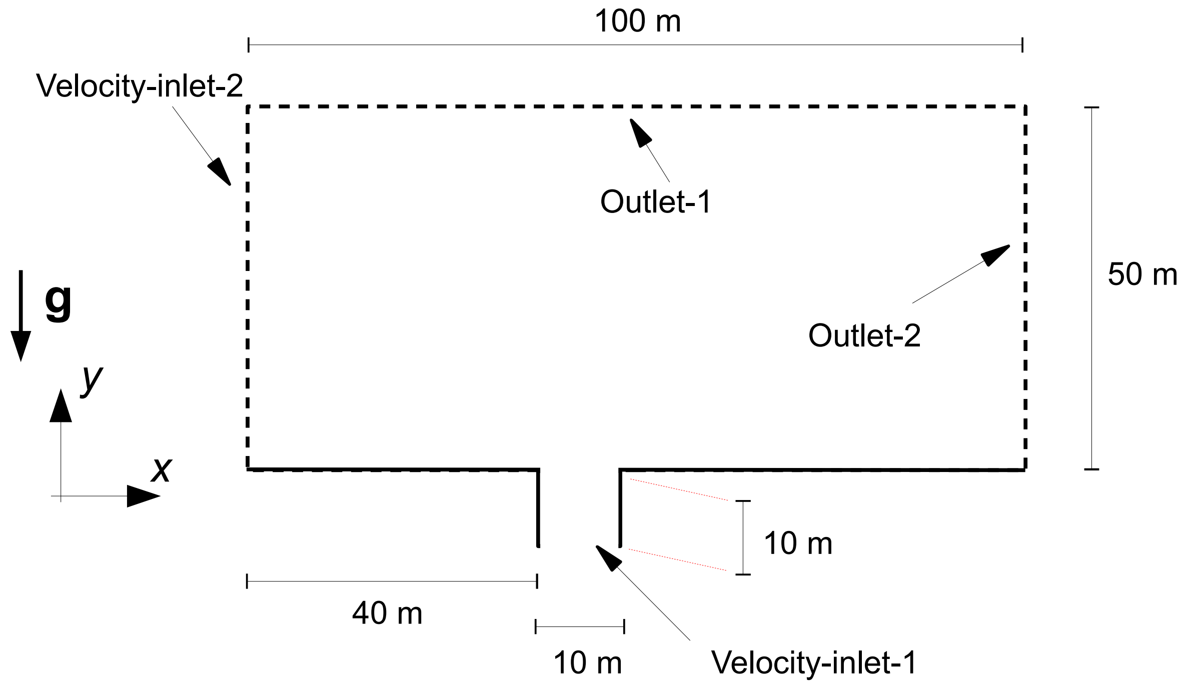

Fig. 3 The geometry of the system for Task 3.

Part II. Regular tasks(s) for MAE 598 only
Work submitted by an MAE 494 student for the task(s) in this part will not be graded and will not be awarded any point.

Task 4 (MAE 598 only)
(a) Consider a 2-D system with an inclined plate that forms a 15° angle with the ground as shown in Fig. 4. At $t = 0$, a drop of engine oil is placed on the plate and it is shaped like a semicircle with a radius of 1 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, 1 s, and 10 s.

(b) Consider the 3-D version of the system in Part (a) with the initial drop of engine oil shaped like a *hemisphere* with a radius of 1 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, 1 s, and 10 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: The use of *viscous-laminar* model is recommended 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 boundary conditions. This information should be included in the report.

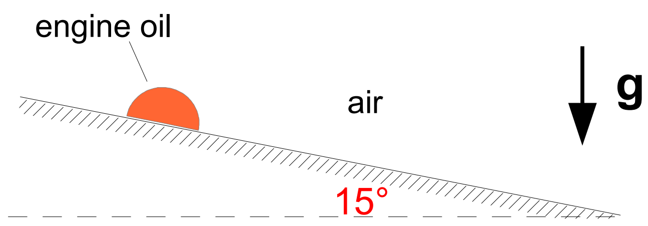

Fig. 4 The initial state of the system considered in Task 4.

Part III. "Bonus pool" challenges

See instructions in Project 1 on the rules for collaboration and preparation of reports for the challenges.

Challenge #3 (pool = 80 points, cap = 4 points)
Consider the system in Part (b) of Task 3 (i.e., with V_{in} = 2 m/s at velocity-inlet-2). Suppose that methane is leaked from the underground reservoir not continuously but sporadically, with the volume fraction of methane at velocity-inlet-1 described by a sequence of top-hat functions as shown in Fig. 5. (In other words, the pipe from the underground reservoir "puffs" methane and clean air alternatively at 5 s intervals.) With this modification but otherwise retaining all other conditions, repeat Part (b) of Task 3 and make contour plots of the volume fraction of methane at t = 5 s, 10 s, 15 s, and 20 s. Specifically, you are required to use a User Defined Function (UDF) to control the temporal variation of the volume fraction of methane at the inlet. The printout of the UDF should be included in the report. A submission without the UDF (i.e., if you use an alternative method, bypassing UDF, to produce the correct simulation) will be awarded only 50% of credit.

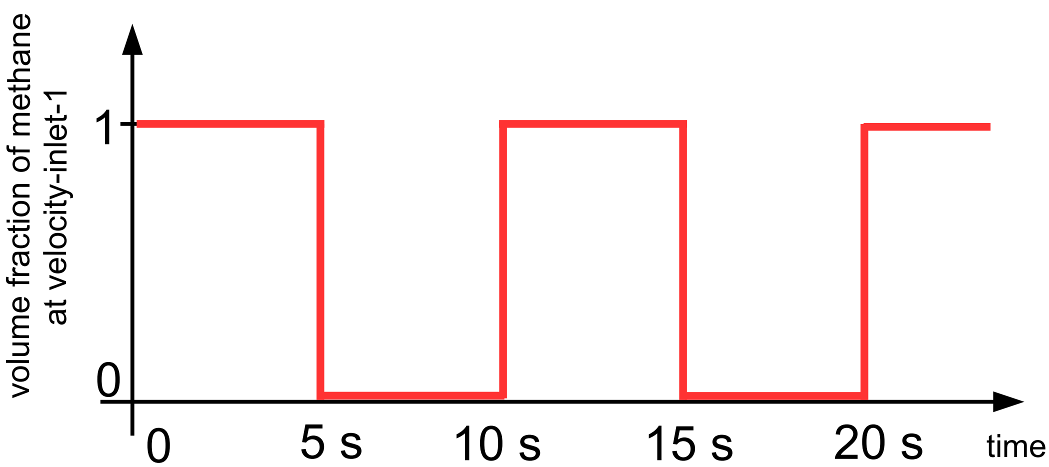

Fig. 5 The temporal variation of the volume fraction of methane at velocity-inlet-1.

Challenge #4 (pool = 100 points, cap = 5 points)
Background: Consider a 2-D system of two open containers connected in the bottom by a pipe, as shown in Fig. 6. Initially, the water level in the left container is higher that in the right container and there is no motion in the entire domain. As the system evolves in time, we expect an oscillation of the water levels in the containers. When the water level decreases in the left container, as a response to it air flows 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. In that sense, neither of the top openings is strictly an "inlet" or "outlet". Due to the effect of viscosity, the oscillation will be damped over time.

(a) Use Ansys-Fluent (choose *viscous-laminar* model) to simulate the oscillation described in *Background*. Specifically, the boundary conditions at the two top openings (red dashed lines in Fig. 6) should be set up such that air will flow through those openings as a passive response to the oscillation of the water body in the system. Extend the simulation to at least three 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) Investigate how the boundary conditions at the top openings affect the solution. For example, will Ansys-Fluent still proceed to run the simulation if both or one of the openings are set as "wall"? Try at least three different combinations of boundary conditions (for the left and right containers) other than the one already used in Part (a). [In a combination, the left and right containers could have different boundary conditions.] Discuss the results.

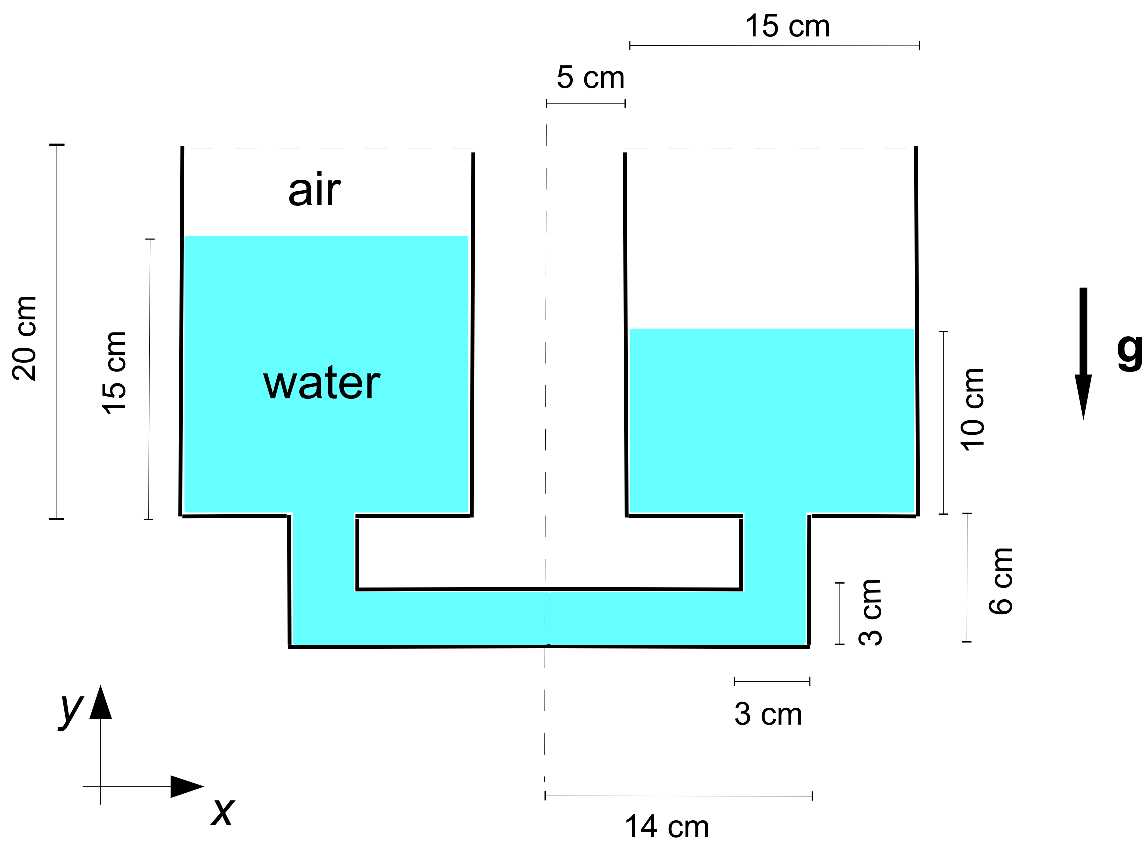

Fig. 6 The apparatus considered in Challenge #4. The vertical dashed line is the line of symmetry for the geometry. The water level as shown is for the initial time.