Project #1 Internal flow with thermal convection

MAE 494/598, Fall 2017, Project 1 (20 points)
Hard copy of report is due at the start of class on the due date. The rules on collaboration will be released separately. Please always follow the rules.

All tasks, except Task 5, are for both MAE598 and MAE494. Task 5 is for MAE598 only.

Background: A prototype of a hot water tank, illustrated in Figs. 1-3, has a main cylinder and two side pipes for the inlet and outlet. All solid surfaces of the system are thermally insulated, except that the temperature at the bottom of the main cylinder is externally maintained at constant 60°C. For all tasks, the temperature of the water entering the inlet is set to 30°C. As cool water flows through the tank, it is heated up by the hot plate at the bottom. Thus, at the steady state we expect the temperature of the outflow (at the outlet) to exceed 30°C. Using Ansys-Fluent, the main objective of this project is to analyze how the steady-state temperature of the outflow is affected by the setup related to the effect of buoyancy in thermal convection.

The key geometric parameters are defined in Figs. 2 and 3. For all tasks, use H = 1.0 m, D = 0.5 m, d = 0.04 m, L = 0.1 m, Z_1 = 0.85 m, and Z_2 = 0.15 m. For all tasks, impose a velocity inlet with u = 0.05 m/s and inlet temperature of 30°C. The velocity and temperature at the inlet are uniform.

Task 1

Use the geometry as shown in Figs. 1-3. Use the same setup for material and boundary conditions as Tutorial #1, except the following:
(i) Turn "gravity" on and set it to regular gravity, i.e., – 9.8 ms^{-2} in the z-direction, as shown in Fig. 1.
(ii) Instead of setting density as constant, switch to "Boussinesq" to allow buoyancy-driven thermal convection. With this setup, the *operating density*, *operating temperature*, and *thermal expansion coefficient* also need to be given according to the Boussinesq approximation. This detail will be discussed in class.
(iii) Set the boundary condition for the outlet to "outflow", instead of "pressure outlet".
(iv) Choose *second order discretization*. (See p. 1-39 of Tutorial #2.)

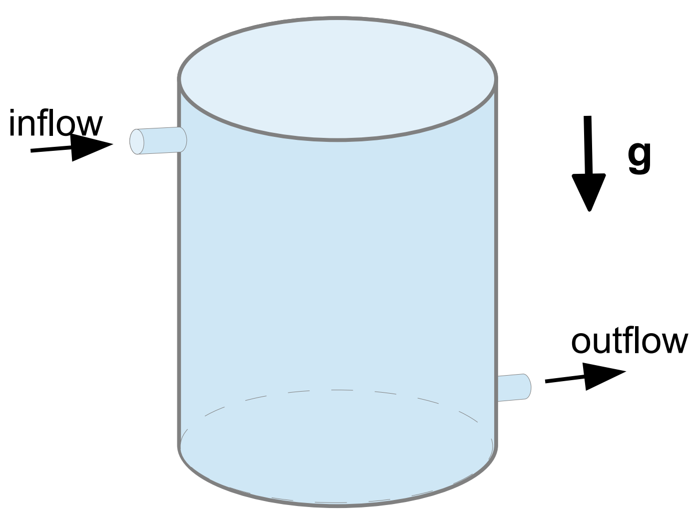

Fig. 1 The water tank system which consists of a main cylinder and two circular side pipes for the inlet and outlet.

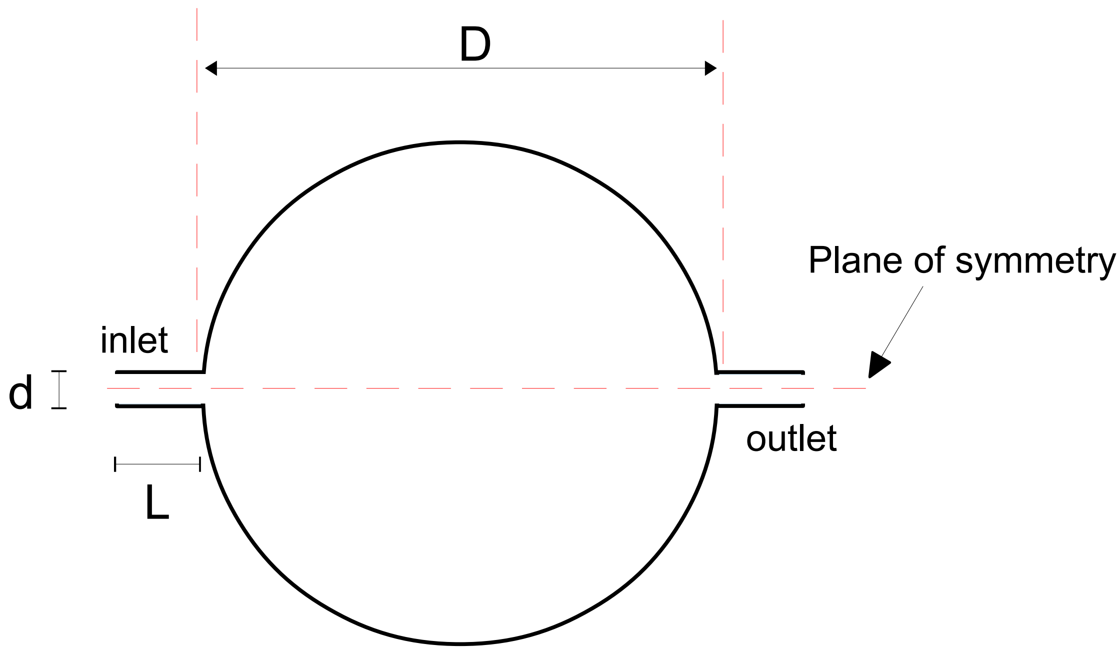

Fig. 2 Top view of the water tank system with a circular cross section. Key parameters: D is the diameter of the main cylinder; d is the diameter of both side pipes; L is the length of both side pipes.

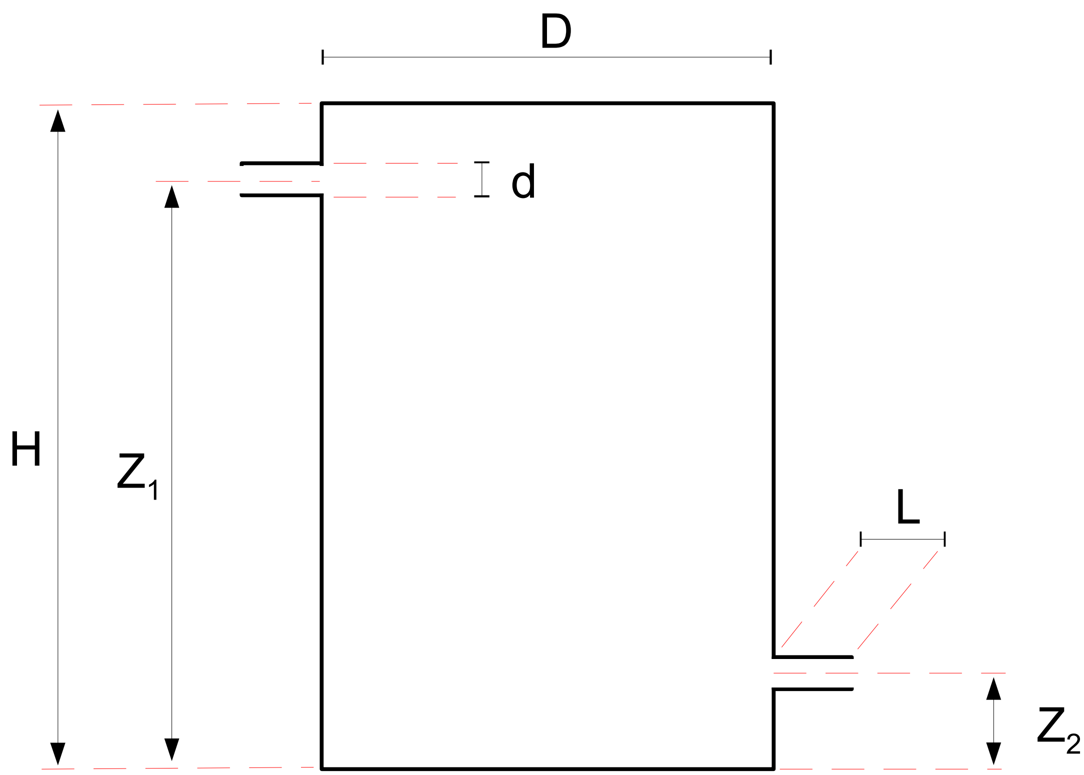

Fig. 3 The vertical cross section of the water tank along its plane of symmetry. Key parameters: H and D are the height and diameter of the main cylinder; Z_1 and Z_2 are the heights of the centers of the side pipes for the inlet and outlet, respectively; L is the length of both side pipes; d is the diameter of both side pipes.

In addition to the four modifications, (i)-(iv), for the setup, local mesh refinement is strongly recommended. For this system, intensive heat transfer occurs at the bottom where temperature gradient is strong. Local mesh refinement over the layer of fluid above the bottom plate will help improve the accuracy of the simulation. This can be done by using a "gradient based criterion" to pick the region for mesh refinement (see pp. 1-45 to 1-49 in Tutorial #2). The relevant gradient here is that of temperature.

Choose "turbulence k-epsilon" model and seek "steady" solution. Run a simulation to obtain the temperature at the outlet. Because the temperature and velocity at the outlet are generally non-uniform (*cf*. Task 3 of HW1), a more meaningful definition of the averaged temperature is

$$T_{out} = \frac{\int\int v_n \, T \; d\,A}{\int\int v_n \; d\,A} \quad , \qquad \text{Eq. (1)}$$

where v_n is the velocity component normal to the outlet and T is the temperature at the outlet. The integral is performed over the surface of the outlet. Please use this definition of T_{out} through the entire project. The deliverables for this task are:

(1) The value of outlet temperature, T_{out} .

(2) Contour plot of temperature on the plane of symmetry.

(3) Contour plot of velocity magnitude on the plane of symmetry.

(4) Plot of stream lines. (An example of acceptable format can be found in p. 15 of the set of typed slides from the first lecture.)

Task 2

Repeat Task 1 but with the main cylinder oriented horizontally such that the axis of the main cylinder is normal to the vector of gravity. The two side pipes are oriented vertically, with inlet on top and outlet at the bottom, as shown in Fig. 4. The deliverables for this task are the same as those listed in Task 1.

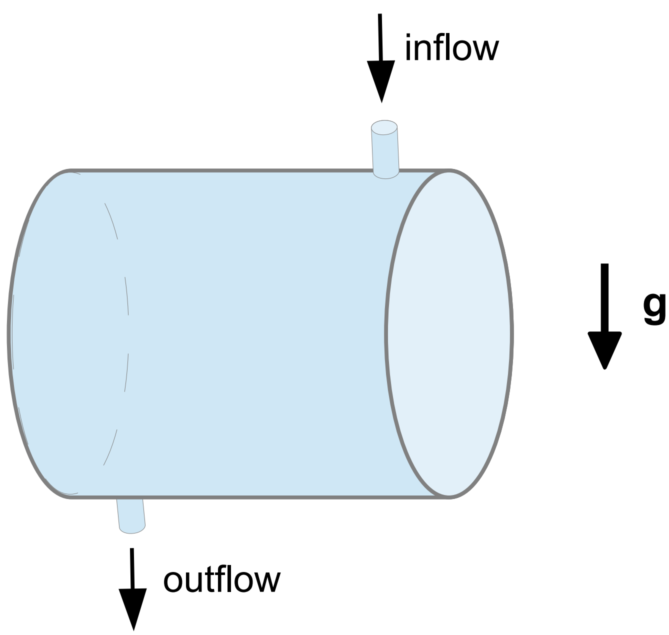

Fig. 4 The geometric setup for Task 2. The "hot plate" with constant 60°C temperature is now at the left and is oriented parallel to the gravity vector.

Hint: For Task 2, it is not necessary to rebuild the geometry. Instead, one can just artificially reset the gravity vector to pointing in the x-direction (or y-direction, depending on the setup of the original geometry in Task 1).

Task 3

Repeat Task 1 or Task 2 but now set density to constant and turn off gravity (i.e., use the old setup in Tutorial #1). Repeat the simulation and produce the deliverables (1)-(4) as listed in Task 1. (With constant density and no gravity, the orientation of the water tank will not affect the result. Thus, for this task one can use the geometry of either Task 1 or Task 2.) In addition, complete the following deliverable:

(5) Compare the results from Task 1, 2, and 3. Based on the differences in the temperature and velocity fields among the 3 cases, try to explain what causes the differences in the outlet temperatures among those cases.

Task 4

Use the geometry, mesh, and parameter setting of Task 1 but now seek *transient* solution. Perform two runs:

(a) Initialize the system such that at $t = 0$ the water in the tank (including both side pipes) has a uniform temperature of 30°C. Denote the time-dependent outlet temperature as $T_{out}(t)$ and the outlet temperature from the steady solution of Task 1 as T_S. Run the transient simulation at least until

$$\left| T_S - T_{out}(t) \right| < 0.3\ °\mathrm{C} \qquad \text{Eq. (2)}$$

(b) Repeat (a) but initialize the run by setting water temperature in the tank to 40°C at $t = 0$. For this case, instead of using the criterion in Eq. (2) to determine the length of the run, just run the simulation to the same termination time as in (a). (This is to prepare the production of the key deliverable as described in the next paragraph.)

From the results of (a) and (b), make a plot of $T_{out}(t)$ as a function of time for both cases. Please collect the two curves in one plot. Moreover, mark the value of T_S (e.g., by an additional horizontal line) in the plot such that one can clearly see how the transient solution approaches the steady solution with increasing time.

Note: (i) For the transient solution, one needs to select the time step size and the numerical scheme for time integration. Those details will be discussed in class. (ii) For a meaningful comparison between the steady and transient solutions, a sufficient number of iterations should be used in Task 1 to ensure that the steady solution has converged.

Additional note for Task 1-4: It is acceptable to run these simulations using either the "full tank" geometry or "half tank" geometry (the latter by invoking symmetry in the simulation). Please indicate your choice in the report. Note that, unlike Task 3 of HW1, the choice would not affect the calculation of T_{out} in Eq. (1), since the integrals in the numerator and denominator will both be over either the full circular disk or the half disk. So, the effect of halving the area of inlet/outlet will cancel out.

Task 5 - For MAE598 only

Participants of MAE 494 do not need to complete this task. Work submitted by MAE 494 students for this task <u>*will not be graded and will not be awarded any point*</u>.

(a) Consider the same setting as Task 1. In that case, the boundary condition at the bottom of the water tank is "temperature = 60°C". At the steady state, since the temperature in the interior of the tank is generally lower than the imposed temperature at the bottom plate, there is a net heat flux (by heat conduction) into the tank from the bottom plate. First, use the "Flux Report" function in Ansys-Fluent to obtain the total rate of heat transfer over the bottom plate. (See p. 1-28 in Tutorial #2 for the use of Flux Report. Note that the value from the report is for the entire bottom plate and is in unit of J s^{-1}.)

(b) Divide the total rate of heat transfer, as determined from (a), by the area of the bottom plate to obtain the average heat flux (in J $s^{-1}m^{-2}$) at the bottom plate. Repeat the simulation in Task 1 by replacing the "constant temperature" ("temperature = 60°C") boundary condition at bottom plate with that of an imposed <u>constant heat flux</u>, using the value of the average heat flux determined in this task. Does this simulation produce the same value of T_{out} at the steady state as the original simulation in Task 1? Provide a brief interpretation of your finding.

(c) Make a contour plot of temperature on the bottom plate.

Note: For this task, be careful about the choice of "full geometry" vs. "half geometry". If "half geometry" (with assumed symmetry) is chosen, the flux report would return the total rate of heat transfer through a half disk (instead of full circular disk) of the bottom plate. Then, the correct value of heat flux in (b) should be obtained by dividing the reported rate of heat transfer by the area of the half disk.