MAE 494/598, Fall 2016   Project #1   (Regular tasks = 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.

Part I. Regular tasks for MAE598/494

Task 1

A simple hot water tank, illustrated in Figs. 1-3, consists of a main cylindrical tank and two small 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 a constant 65°C. For this exercise, the temperature of the water entering the inlet is set to 25°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 to exceed 25°C. Using Ansys-Fluent, the main objective of this task is to analyze how the steady-state temperature of the outflow is affected by the vertical positions of the inlet and outlet.

Use the same setup for material and boundary conditions as Tutorial #1, except the following:
(1) Turn "gravity" on and set it as the regular gravity, i.e., – 9.8 ms^{-2} in the z-direction.
(2) 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.
(3) Choose *second order discretization*. (See Step 10 of Tutorial #2.)

The key geometric parameters are defined in Figs. 2 and 3. For this task, use H = 1.0 m, D = 0.5 m, d = 0.04 m, and L = 0.1 m. Only the height of the center of the side pipe (Z_1 for the inlet, Z_2 for the outlet) will be varied. Choose u = 0.05 m/s as the inlet velocity. The velocity and temperature at the inlet are uniform for all cases.

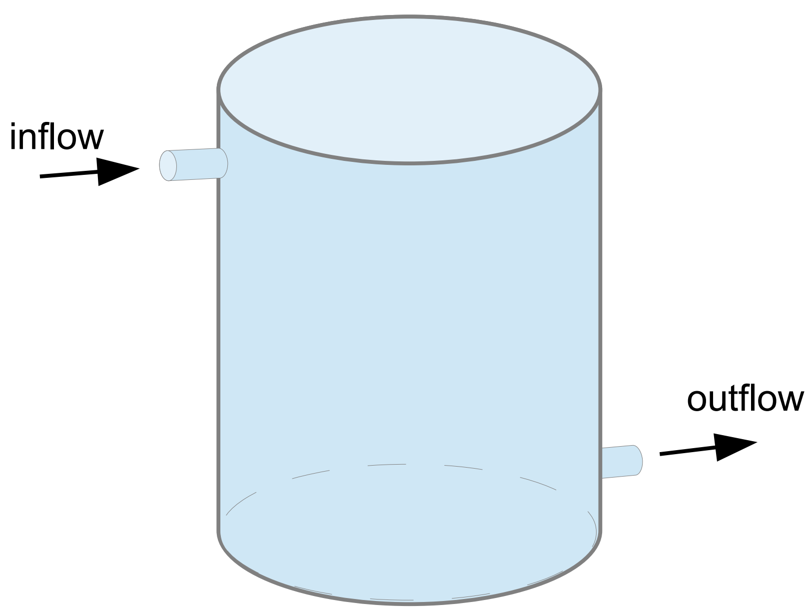

Fig. 1 The water tank system which consists of a main cylinder and two circular side pipes for the inlet and outlet.

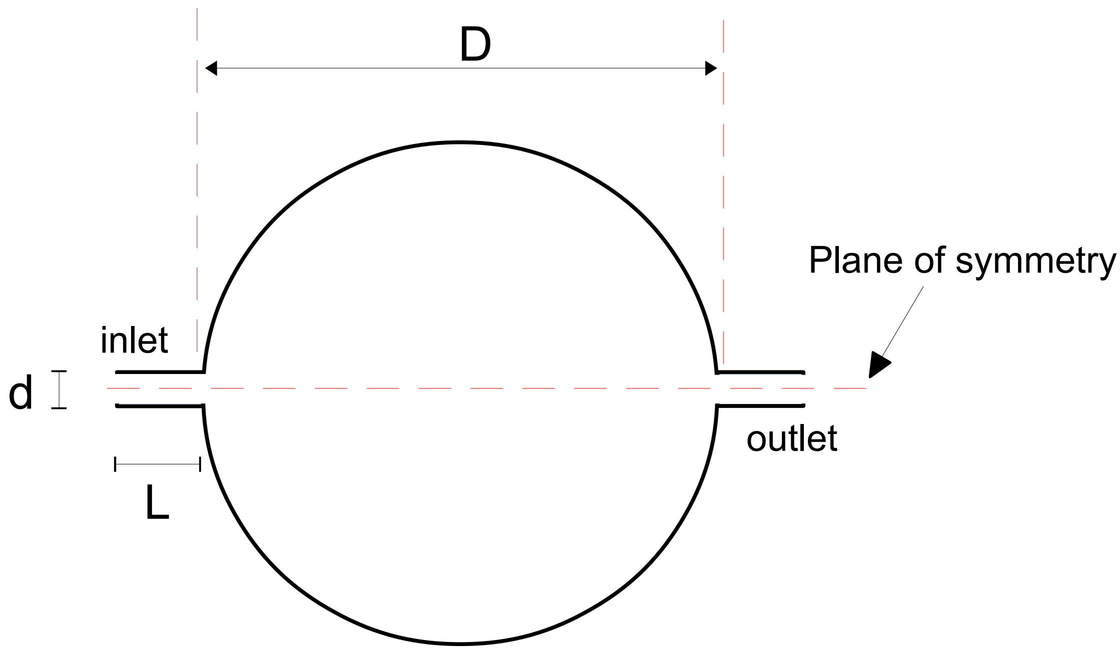

Fig. 2 Top view of the water tank system with a circular cross section. Key parameters: D is the diameter of the main cylinder; d is the diameter of both side pipes; L is the length of both side pipes.

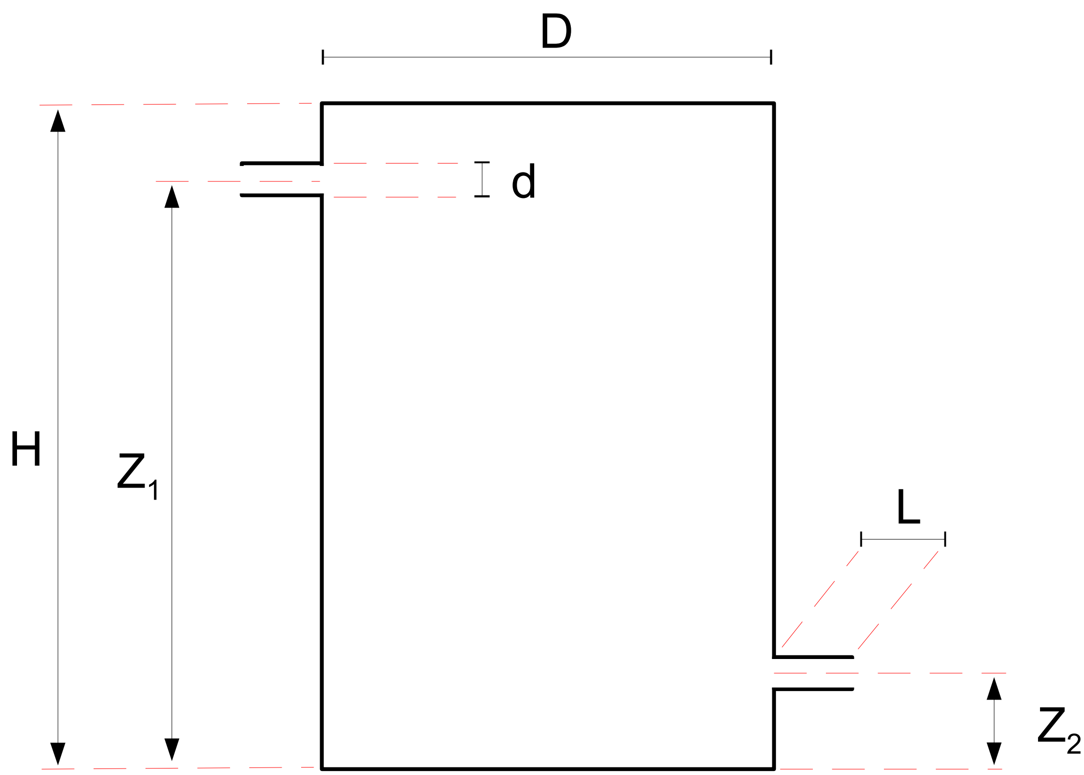

Fig. 3 The vertical cross section of the water tank system along its plane of symmetry. Key parameters: H and D are the height and diameter of the main cylinder; Z_1 and Z_2 are the heights of the centers of the side pipes for the inlet and outlet, respectively; L is the length of both side pipes; d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

Choose "turbulence k-epsilon" model and seek "steady state" solution. Consider the following two designs with contrasting vertical positions of the side pipes for inlet and outlet:

Case A: $Z_1 = 0.8$ m, $Z_2 = 0.2$ m
Case B: $Z_1 = 0.2$ m, $Z_2 = 0.8$ m (i.e., swapping the vertical positions of inlet and outlet from Case A)

Run Ansys-Fluent simulations for the two cases to obtain the temperatures 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

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

where v_n is the non-uniform velocity normal to the outlet and T is the non-uniform temperature at the outlet, and the integral is performed over the circular opening of the outlet. Please use this definition of T_{out} through the entire project.

The deliverables for this task are

(1) The values of outlet temperature, T_{out}, from the simulations of the two cases.
(2) Contour plots of temperature on the plane of symmetry for the two cases.
~~(3) Contour plots of temperature on the horizontal cross sections at Z = 0.2 m and Z = 0.8 m, for the two cases~~. [Revised: deliverable (3) is no longer required for Task 1-3 and Challenge #1.]
(4) Plots of stream lines for the two cases. (An example of acceptable format can be found in p. 9 of the set of slides named "More slides from the first lecture" posted at our class website. Follow the example in the right panel of that slide that shows the 3-D structure of the stream lines.)
(5) A discussion of the possible reasons for the differences in T_{out} between the two cases based on the information obtained from the numerical simulations.

Task 2

This task will assess the dependence of T_{out} on the geometry of the main water tank by considering two additional designs as shown in Fig. 4. In both cases, all design parameters follow <u>Case B in Task 1</u> (specifically, $Z_1 = 0.2$ m, $Z_2 = 0.8$ m) except that the horizontal cross section of the tank is changed from circular to elliptical. By choosing D_1 and D_2 (see Fig. 4) such that $D_1 D_2 = D^2$ (where D is the diameter of the circular design in Fig. 2), the total volume of the main tank remains the same as the design in Task 1. Specifically, consider:

Case C: $D_1 = 0.625$ m, $D_2 = 0.4$ m. The inlet and outlet are aligned with the major axis of the ellipse. See upper panel of Fig. 4.
Case D: $D_1 = 0.4$ m, $D_2 = 0.625$ m. The inlet and outlet are aligned with the minor axis of the ellipse. See lower panel of Fig. 4.

Perform Ansys-Fluent simulations for the two cases and produce the same deliverables as described in Task 1. In deliverable (5), compare the efficiency of the elliptical design with the circular one in Task 1.

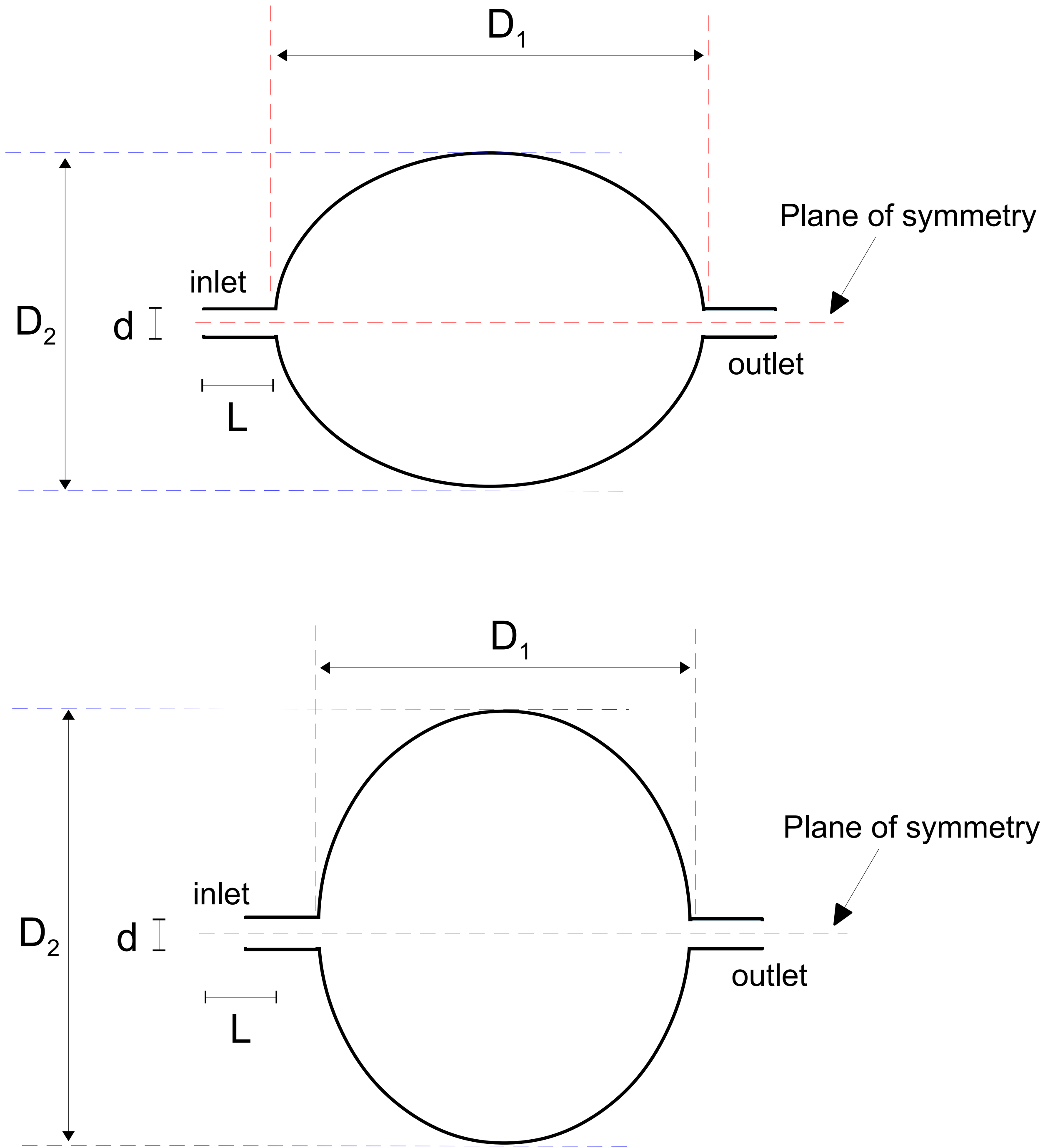

Fig. 4 The two "elliptical designs". Top: The inlet and outlet are aligned with the major axis. Bottom: The inlet and outlet are aligned with the minor axis. Not drawn to scale.

Task 3

Consider Case A in Task 1 but now set density to constant and turn off gravity. Repeat the simulation and produce the counterparts of deliverables (1)-(5) in Task 1. In deliverable (5), discuss what critical process(es) are missing when density is set to constant, and how they affect the value of T_{out}.

Part II. Regular task for MAE598 only

Participants of MAE 494 do not need to complete the task in this part. Work submitted by MAE 494 students for this part will not be graded and will not be awarded any point.

Task 4

(a) Consider Case B in Task 1. In that case, the boundary condition at the bottom of the water tank is "temperature = 65°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 into the tank from the bottom plate. Since water does not flow through the bottom plate, this heat flux is due to heat conduction. Using ideas similar to those considered in Task 3 of Homework 1, evaluate the total flow rate of heat from the bottom plate into the tank. (As is in Task 3 of HW1, this quantity should have the unit of Watts.)

(b) Divide the total flow rate of heat, as determined from (a), by the area of the bottom plate to obtain the average heat flux at the bottom plate. Repeat the simulation of Case B in Task 1 by replacing the "constant temperature" ("temperature = 65°C") boundary condition at bottom plate with that of an imposed constant heat flux, using the value of the average heat flux determined earlier in this task. Does this simulation produce the same value of T_{out} as the original simulation for Case B in Task 1? Provide a brief interpretation of your finding.

Part III "Bonus pool" challenges

Please submit the (hard copy) answer to each challenge SEPARATELY, one challenge per report. Do not mix different challenges or mix the challenges with regular tasks.

Each correct (or nearly correct) answer earns one share of the bonus. Occasionally, a "half correct" answer might be granted a half share. Low quality answers will receive no credit at all. The actual point(s) earned per share will be the total bonus points for a challenge divided by the number of shares granted for that challenge. The only exception is if the "points per share" exceeds a preset cap of reward. For example, Challenge #1 has 80 points in the pool and a cap of 4 points. If 40 shares are granted, each will be equivalent to 2 points. If only 15 shares are granted, each will be equivalent to 4 points, i.e., the preset cap.

The maximum an individual can earn from the challenges is 15 points for the semester. Solutions to challenges submitted by an individual who already has 15 points from the bonus pools will not be graded and will not be awarded any point.

The rules on collaboration for challenges are the same as those for regular tasks, except that (i) No need to use the "cover sheet" for submissions of challenges; (ii) No need to state the detail of collaboration and percentage contribution. Instead, simply include a one-line statement of "No collaboration" or "Collaboration with [name of collaborator]" in the first page of the report.

Challenge #1 (80 points in the pool; cap = 4 points)

In Case B of Task 1, a uniform temperature of 65 °C is imposed at the bottom plate. Consider a modification of the design in which non-uniform (but still time-independent) temperature is imposed at the bottom plate. The imposed temperature is axially symmetric, i.e., $T \equiv T(r)$ where r is the radial distance from the center of the bottom plate. Specifically, $T(r)$ in °C is given as

$$T(r) = 65 \exp(-r/\mathrm{D}) \ , \qquad \text{Eq. (2)}$$

where D = 0.5 m. [Thus, the imposed T is 65 °C at the center of the circular plate but drops to 65 exp(–0.5) ≈ 39.4 °C at the edge of the plate.] Use *User Defined Function* (UDF) to impose the non-uniform temperature, given by Eq. (2), at the bottom plate and repeat the simulation of Case B in Task 1. Produce the counterparts of deliverables (1)-(4) as described in Task 1. In addition, to demonstrate that the setup of the new boundary condition is correct, the following deliverables are also required:

(5) The printout of the code for the UDF

(6) Contour plot of temperature on the horizontal cross section at Z = 0 (i.e., at the bottom of the main tank).

Challenge #2 (80 points in the pool; cap = 4 points)

Using the design of Case A in Task 1, we wish to determine the dependence of temperature at different parts of the system on the flow velocity at the inlet, V_{in}. Specifically, we wish to obtain the values of two quantities, T_{out} and T_{mid}, from 10 simulations with V_{in} = 0.01, 0.02, 0.03, ..., 0.09, 0.1 m/s. Here, T_{out} is defined by Eq. (1) as in Task 1, with the integration performed over the surface of the outlet.

T_{mid} is also similarly defined by the right hand side of Eq. (1) but with the integration performed over the horizontal cross section at Z = 0.5 m (i.e., the circular cross section that cuts through the middle of the tank). While this task could be accomplished by manually changing V_{in} repeatedly and executing 10 separate simulations , Ansys-Fluent has a special tool called *Parametric Design* which can be used to set up and automatically execute the 10 simulations in sequence. This challenge requires that you use Parametric Design to complete the 10 simulations. The deliverables are

(1) A description, including screenshot(s) of critical step(s), of the setup for parametric design. This description should also include the detail of how the values of T_{out} and T_{mid} are extracted from the 10 runs under the framework of parametric design.

(2) Line plots of T_{out} and T_{mid} as a function V_{in}. These plots summarize the outcome of the 10 runs.

Credit will be given only to the submissions that use parametric design to complete the task. No point will be awarded if deliverable (1) is missing or incorrect, even if correct plots are provided for deliverable (2). In other words, no point will be awarded if the plots in (2) are generated by manually executing 10 separate runs then stitching the results together.