STOP

**Please read the rules before forming any collaboration for the project.
A violation of the rule(s) given in this page will be considered a violation
of ASU's Academic Integrity Policy.**

Rules on collaboration for project:

(1) Collaboration is not allowed unless all involved follow rules (2)-(3) and unless the extent of collaboration is properly disclosed in a statement in the first page of the report for the project. See additional instruction below for the required content of the statement.

(2) For each project, each person can have maximum of one collaborator. Be aware that a collaborator's collaborator counts as a collaborator. For example, if Alice collaborates with Bob and Bob collaborates with Charles, Charles counts as a collaborator of Alice. All three violate the rule. In other words, collaboration can only be carried out within an isolated "team of two". Please talk to a potential collaborator to ensure that this rule is not violated before establishing any collaboration.

(3) In a legitimate collaboration, each individual must make a non-negligible contribution to the collaborative effort. Taking the solution or code from another student without making a reciprocal contribution to it is not legitimate collaboration and is not allowed. To certify that the collaboration is legitimate, the contribution from each person to the collaborative effort must be documented in the mandatory statement as described in (1).

The statement on collaboration

This statement is mandatory and must be placed in the beginning of the first page of report. If no collaboration occurred, simply state "**No collaboration**". This implies that the person submitting the report *has not helped anyone or received help from anyone in the class* on this assignment. If collaboration occurred, provide the name of collaborator (only one allowed), a list of the task(s) on which collaboration occurred, and descriptions of the extent of collaboration. For example:

Name of collaborator: Joe Smith	
Task(s), specific detail	Contribution to collaborative effort
Task 1, Setup of geometry and mesh	Helped overcome key steps for mesh refinement
Task 3, Whole task	Worked with collaborator to set up the transient simulation and analyze results

Hard copy of report is due at 1:30 PM on the due date. Please follow the rules for collaboration as described in the cover page of this document. **A statement on collaboration is mandatory for all**. All tasks, except Task 4, are for both MAE598 and MAE494. Task 4 is for MAE598 only.

Background: A prototype of a water heater, illustrated in Figs. 1-3, has a main cylindrical tank and two cylindrical 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 50°C. (In Task 4, an equivalent heat flux is imposed.) For all tasks, the temperature of the water entering the inlet is set to 15°C. As cool water flows through the tank, it is heated up by the hot plate at the bottom. Thus, the temperature of the outflow generally exceeds 15°C. Using Ansys-Fluent, this project will quantify how the temperature of the outflow is affected by the setup of the system.

The key geometric parameters are defined in Figs. 2 and 3. For all tasks, use H = 1.0 m, D = 0.8 m, d = 0.05 m, L = 0.1 m, Z_1 = 0.85 m, and Z_2 = 0.15 m. In all descriptions of the tasks, we orient the axis of the main cylinder in *y*-direction and the axes of the side pipes in *x*-direction, as illustrated in Fig. 1a. (You may use a different definition of the directions, as long as consistency is maintained through the project.) Gravity, when turned on, is set to point in the negative *y*-direction.

For all tasks, the boundary conditions are **velocity inlet** for the inlet and **outflow** for the outlet. The velocity at the inlet is set to a constant 0.05 m/s (for the velocity component normal to the surface of inlet) and inlet temperature is set to a constant 15°C. The system is filled with water. For all tasks, the viscosity, specific heat, and heat conductivity of water are set to the default constant values from Fluent database. Energy equation is turned on, and turbulence k-epsilon model is chosen for the simulations.

Task 1

This task involves two simulations, both using the following common setup (in addition to the basic setup as already described in **Background**):

(i) Turn **gravity** on and set it to – 9.81 ms⁻² in *y*-direction (see Fig. 1a).
(ii) Seek **steady solution**.
(iii) Instead of setting density to constant, switch to **Boussinesq** to allow buoyancy-driven thermal convection. With this setup, the appropriate *operating density*, *operating temperature*, and *thermal expansion coefficient* also need to be given according to the Boussinesq approximation. By the Boussinesq setting, the **pressure based solver** can still be used even though density varies with temperature. Critical details on the Boussinesq setting will be given in class.

Simulation #1 uses the configuration in Fig. 1a with cold water entering the main tank through the inlet near the top, and exiting the tank through the outlet near the bottom. In Simulation #2, all setups are the same but we swap inlet and outlet. Namely, cold water enters the system though the inlet near the bottom, and exits it through the outlet near the top, as shown in Fig. 1b. For both cases, the blue and red arrows in Fig. 1 indicate cooler and warmer water. The key deliverables are, for each simulation,

(i) Outlet temperature of the steady state. Since temperature may not be uniform across the surface of the outlet, this quantity should be defined as the average of temperature over the surface of the outlet,

$$T_{out} = \frac{1}{A} \iint_A T\, dA \quad ,$$

Eq. (1)

where A is the surface area of the outlet and the integration is performed over the surface of the outlet. (The T_{out} so defined can be computed using Fluent's built-in utilities: Reports → Surface Integral → Area Weighted Average.)

(ii) A contour plot of *temperature* in the *plane of symmetry* (which runs through the axes of the main tank and two side pipes). [Note: In the steady state, there is still a tight temperature gradient near the bottom while the temperature elsewhere in the main tank is relatively uniform. In order to clearly show the key feature in the temperature field, namely, the "cool waterfall" coming down from the inlet, it is recommended that the contour intervals be manually adjusted.]

(iii) A contour plot of the *y-velocity* (not to be confused with *velocity magnitude*) in the *plane of symmetry*.

Recommendations on geometry and mesh

For all tasks, it is acceptable to either use the geometry of the full tank, or invoke symmetry and run the simulations with a half tank (and half side pipes). Once the choice is made, all simulations must be run with the same geometry. Please clearly indicate your choice of geometry in the report. (Some modes of fluid motion are inherently non-symmetric for a turbulent flow. They will be suppressed if symmetry is invoked. On that ground, we slightly prefer full geometry. However, we do not expect this detail to significantly affect the key deliverables.)

We do not designate a specific configuration of mesh to use for the simulations. Nevertheless, two features are essential: (1) Enhanced resolution near the wall and within the two side pipes, similar to that used in Homework #1; (2) Enhanced resolution in the vicinity of the bottom plate, which can be achieved by using the Adapt → Region or Adapt → Gradient utilities. Further details about these utilities will be discussed in class.
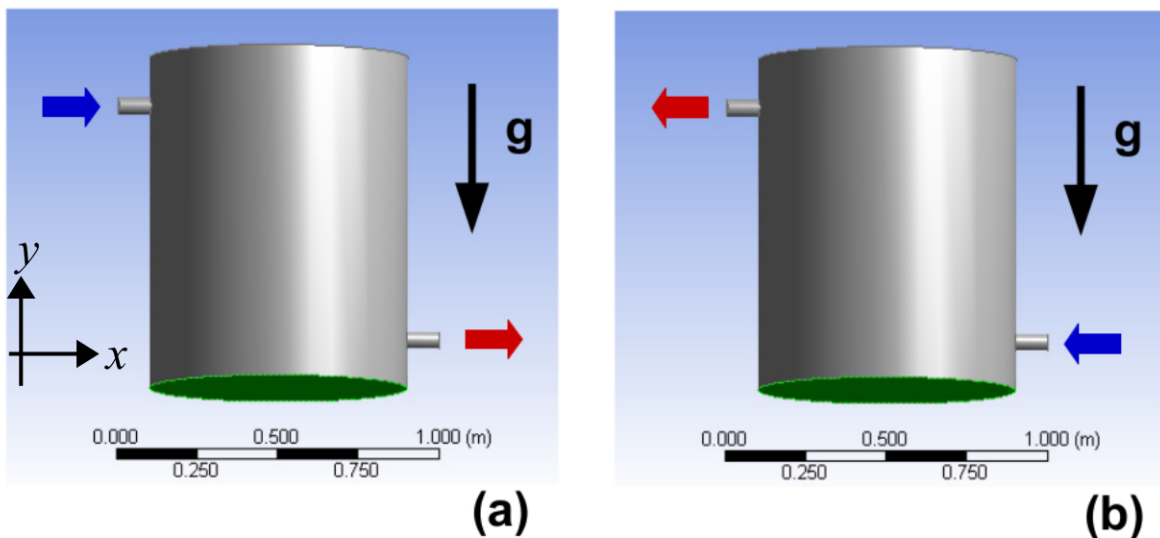

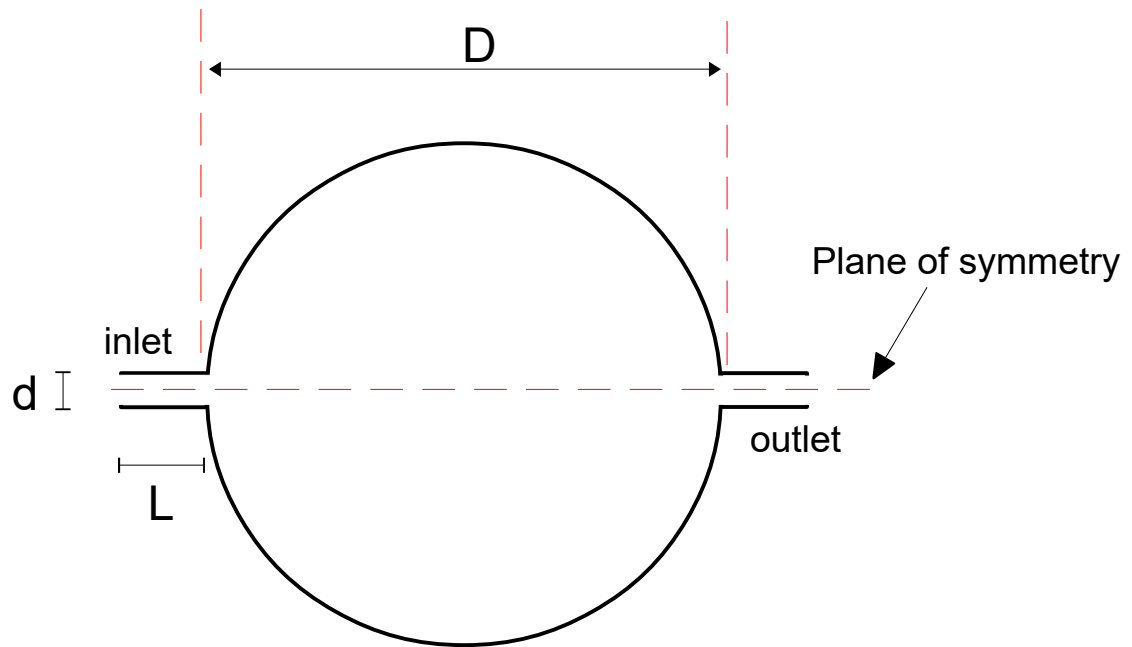**Fig. 1** The general setup of the prototypical water heater. The bottom plate is in green.

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.
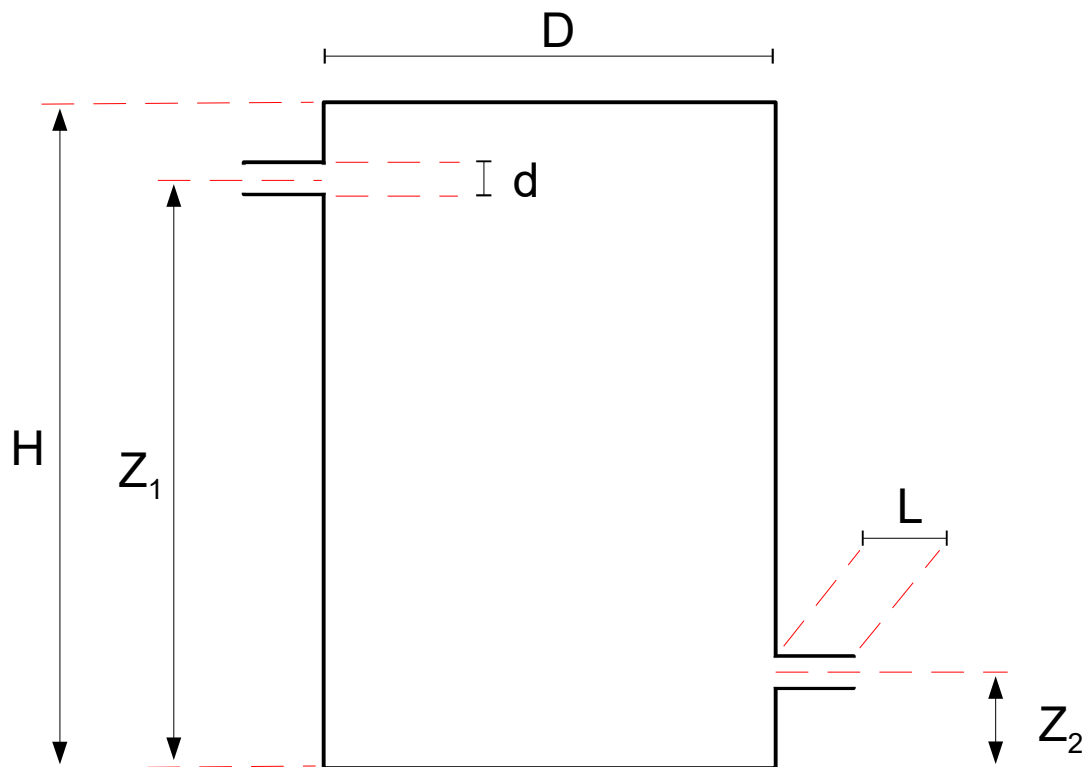

Fig. 3 The vertical cross section of the water tank along its plane of symmetry. Key parameters: H and D are the height and diameter of the main cylinder; Z_1 and Z_2 are the heights of the centers of the side pipes; L is the length of both side pipes; d is the diameter of both side pipes.

Task 2

Repeat Simulation #1 in Task 1 but set density to constant (using the value of the *operating density* from Simulation #1) and turn off **gravity**. All other setups, including the mesh, should be identical to those for Simulation #1 in Task 1. The deliverables for this task are

(i) Outlet temperature of the steady state, as defined by Eq. (1).

(ii) A contour plot of the *y-velocity* in the *plane of symmetry*.

(iii) A contour plot of the *x-velocity* in the *plane of symmetry*.

Task 3

Repeat Simulation #1 in Task 1 but now seek **transient** instead of *steady* solution. Initialize the system such that at $t = 0$ the water in the tank (including both side pipes) has a uniform temperature of 15°C and uniform velocity of $(u, v, w) = (0.05, 0, 0)$ m/s (i.e., the velocity in the tank is the same as inlet velocity). For *turbulence intensity*, just initialize with the default values given by Fluent. [Note: Leave "*Compute from*" <u>blank</u> when performing the initialization. DO NOT designate "compute from inlet".]

Denote the time-dependent outlet temperature as $T_{out}(t)$ and the outlet temperature from the steady solution of Simulation #1 in Task 1 as T_S. Run the transient simulation at least until

$$\left| T_S - T_{out}(t) \right| \leq 2 \,°C \quad . \qquad\qquad\qquad\qquad \text{Eq. (2)}$$

For example, if (hypothetically) the T_S from the steady solution is 35 °C, the transient simulation needs to be run to a time when $T_{out}(t)$ reaches 33 °C. [Note: Just like the steady solution, at every time step the $T_{out}(t)$ for the transient solution needs to be computed as an area-weighted average defined by Eq. (1).]

The deliverables of this task are

(i) A contour plot of *y-velocity* in the *plane of symmetry* at the <u>end time</u> of the transient simulation.

(ii) A plot of $T_{out}(t)$ as a function of time, over the entire range of time for the transient simulation.

(iii) A plot of another quantity, $S(t)$, defined below, as a function of time over the entire range of time for the transient simulation. Here, S (which depends on time) is defined by

$$S = \frac{1}{A} \iint_A \frac{\left(|V| - V \right)}{2} \, dA \quad , \qquad\qquad\qquad\qquad \text{Eq. (3)}$$

where the integration is performed over the **plane of symmetry**, V is the *y-velocity*, and A is the surface area of the *plane of symmetry*. This quantity is a measure of the strength of the *downward* velocity associated to the "cool waterfall" from inlet. Note that upward velocity does not contribute to the integral, because $|V| - V = 0$ when $V > 0$. By the definition in Eq. (3), S should be positive and having the unit of velocity (in m/s).

[Note: For the transient solution, one needs to select the numerical scheme for time integration. For this task, it is fine to use the default (which is an implicit scheme). The transient simulation requires further setups for the *time step size*, *number of time steps*, and *maximum number of iteration per time step*. Those details, along with relevant built-in utilities for processing the transient simulation, will be discussed in class.]

Task 4 - For MAE598 only
Participants of MAE 494 do not need to complete this task. Work submitted by MAE 494 students for this task will not be graded and will not be awarded any point.

(a) Consider the same setting as Simulation #1 in Task 1 (for the steady solution). In that case, the boundary condition at the bottom of the water tank is "temperature = 50°C". In the steady state, the heat supplied by the bottom plate helps maintain the difference between outlet and inlet temperature. First, use the "Reports → Fluxes" function in Fluent to obtain the total rate of heat transfer over the bottom plate. The value from the report is for the entire bottom plate and should be in unit of W. (Alternatively, this rate of heat transfer could be inferred from the difference between the heat fluxes at the outlet and inlet, in the same fashion as the analysis in Task 3 of Homework #1. In the steady state, this difference should equal the supply of heat by the bottom plate.)

(b) Divide the total rate of heat transfer, as determined in (a), by the area of the bottom plate to obtain the average heat flux (in unit of W m^{-2}) at the bottom plate. Repeat Simulation #1 in Task 1 by replacing the "constant temperature" ("temperature = 50°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.

The deliverables of this task are

(i) Three numbers: The total rate of heat transfer for the bottom plate (in W) as determined in (a), the equivalent heat flux (in W m^{-2}) to impose as the boundary condition for the bottom plate, and the T_{out} from the new simulation in (b).

(ii) A brief description of how the three numbers in (i) are computed, and a comparison of the T_{out} between the original (with imposed temperature) and new (with imposed heat flux) simulations. Do the two produce the same value of T_{out}? Provide a brief interpretation of your finding.