MAE 494/598, Fall 2015 Project #1 (18 points)

Overview

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 70°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 be higher than 25°C.

Using Ansys-Fluent, the main activities of this project are to analyze (i) How the steady-state temperature of the outflow is affected by the vertical positions of the inlet and outlet; (ii) How the steady-state temperature of the outflow is affected by the mass flow rate at the inlet; (iii) How the choice of "turbulence" vs. "laminar" model affects the outcome of the simulation. The additional task marked by a triangle (▲) is for participants of MAE 598 only. Participants of MAE 494 do not need to complete that task. Work submitted by an MAE 494 student for that task will not be graded and will not be awarded any point.

For all tasks, set the density, specific heat, thermal conductivity, and viscosity of water as constants exactly as in Tutorial #1. Use the same setup of boundary conditions for the inlet and outlet as Tutorial #1. Use "turbulence k-epsilon model" except for Task 4. Turn "energy equation" on.

The key geometric parameters are defined in Figs. 2 and 3. For all tasks, use H = 1.2 m, D = 0.6 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. Use u = 0.05 m/s as the inlet velocity except for Task 3. The velocity and temperature at the inlet are uniform for all cases.

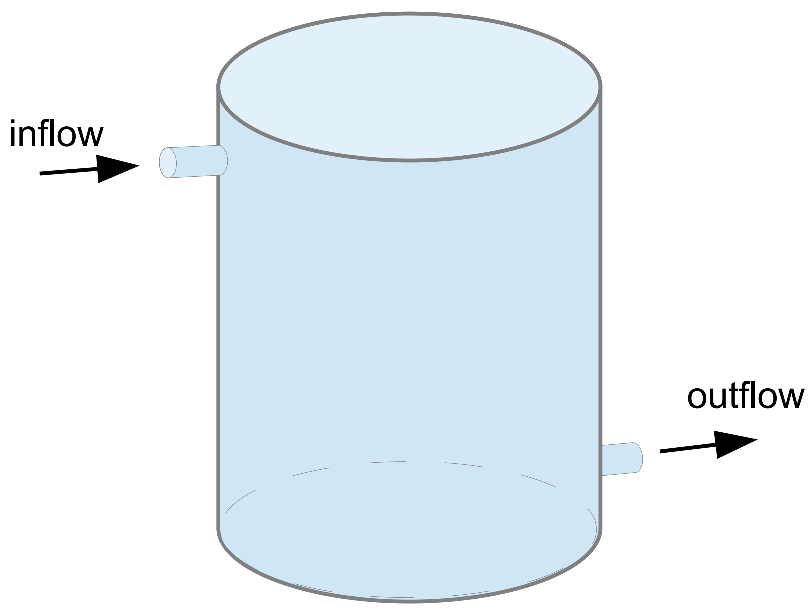

Fig. 1 The water tank system which consists of a main cylinder and two circular side pipes for the inlet and outlet.

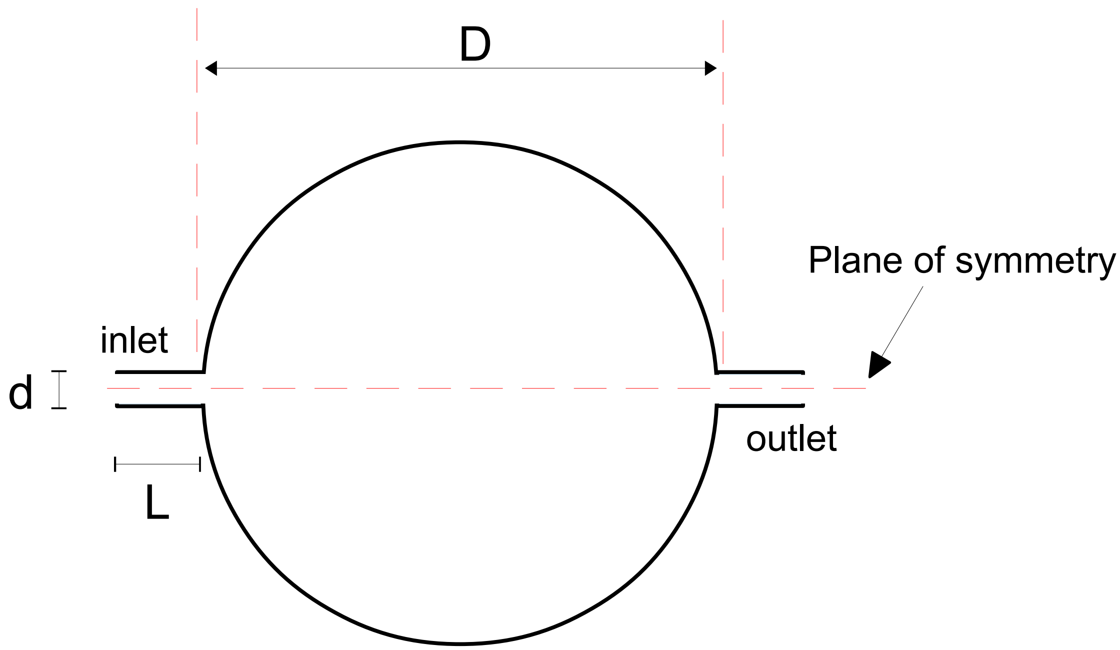

Fig. 2 Top view of the water tank system. Key parameters: D is the diameter of the main cylinder; d is the diameter of both side pipes; L is the length of both side pipes.

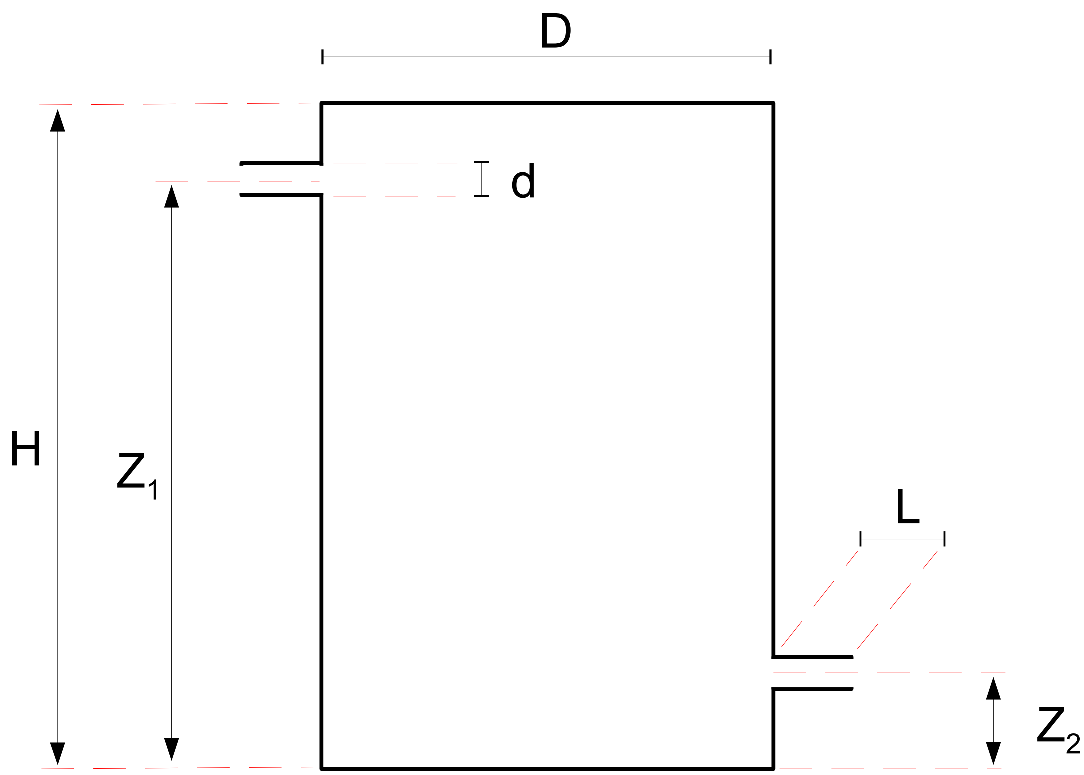

Fig. 3 The vertical cross section of the water tank system along its plane of symmetry. Key parameters: H and D are the height and diameter of the main cylinder; Z_1 and Z_2 are the heights of the centers of the side pipes for the inlet and outlet, respectively; L is the length of both side pipes; d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

Task 1

Choose "turbulence k-epsilon" model and "steady state" solution. Consider the nine combinations of the vertical positions of the side pipes with Z_1 = (0.2 m, 0.6 m, 1.0 m) and Z_2 = (0.2 m, 0.6 m, 1.0 m). Run Ansys-Fluent for those 9 cases to obtain the temperatures at the outlet. Summarize the results in a table (see an example below).

<table>
<tr><td colspan="2" rowspan="2">Outlet temperature (in °C)</td><td colspan="3">Z_1</td></tr>
<tr><td>0.2 m</td><td>0.6 m</td><td>1.0 m</td></tr>
<tr><td rowspan="3">Z_2</td><td>0.2 m</td><td></td><td></td><td></td></tr>
<tr><td>0.6 m</td><td></td><td></td><td></td></tr>
<tr><td>1.0 m</td><td></td><td></td><td></td></tr>
</table>

Because the temperature and velocity at the outlet are generally not uniform (*cf.* Task 3 of HW1), an averaging of temperature over the surface of the outlet is needed to fill the table. The most meaningful definition of the averaged temperature in this case is

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

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 surface of the outlet. This is the preferred definition of T_{out} for this project. If you wish to take a short cut, accepting a voluntary 0.2 point deduction (for the whole project), the following definition is also allowed:

$$T_{out} = \frac{1}{A}\int\int T \, dA , \qquad \text{Eq. (2)}$$

where A is the surface area of the outlet. Useful strategies for evaluating Eq. (1) or (2) can be found in the discussion of Challenge #1. Please use the same definition of T_{out} through the entire project. In the report, indicate which definition of T_{out} is used. If Eq. (1) is chosen, please provide a brief description of how the calculation is carried out, in the fashion of the posted answers for Challenge #1.

Task 2

For the three cases from Task 1 with the highest, fifth highest (i.e., median), and lowest outlet temperature, plot (i) the contour maps of temperature for the three horizontal (circular) cross sections at z = 0.2 m, 0.6 m, and 1.0 m ("z" is the vertical distance measured from the bottom of the main cylinder); (ii) the contour maps of temperature for the vertical cross section along the plane of symmetry (i.e., the cross section shown in Fig. 3); (iii) the contour maps of the *velocity component parallel to the inlet velocity*, for the vertical cross section along the plane of symmetry. For example, if by your design the side pipe for the inlet is parallel to the *x*-axis, the contour maps in (iii) should be those for the *u*-velocity.

Use the contour maps generated for this task to interpret the outcome of Task 1. For example, if a particular choice of (Z_1, Z_2) yields the highest outlet temperature, explain why. (This discussion is important. Do not treat it as an afterthought.)

Task 3

For the case from Task 1 with the highest outlet temperature, repeat the simulation but with the inlet velocity set to 0.025 m/s, 0.1 m/s and 0.2 m/s. Plot the steady-state outlet temperature as a function of inlet velocity for the four cases (including the original one from Task 1). For the case with 0.2 m/s inlet velocity, make contour plots of temperature and velocity for the vertical cross section in the same fashion as Task 2 ((ii) and (iii) only; no need to show the horizontal cross sections).

Task 4

For the case from Task 1 with the highest outlet temperature, change the "turbulence k-epsilon" model to "laminar" model and redo the simulation. Calculate the steady-state outlet temperature. Is the outlet temperature sensitive to the choice of the model? Does the behavior of numerical convergence (based on the diagram of "residual" vs. "number of iteration" provided by Fluent) change when you switch the model from turbulence to laminar?

▲ Task 5

Consider the case from Task 1 with the highest outlet temperature.

(i) Based on the difference between the heat fluxes that go through the outlet and inlet, estimate the average heat flux that comes out of the bottom boundary into the main water tank.

(ii) Replace the boundary condition, "temperature = 70°C", for the bottom boundary by a boundary condition with an imposed constant *heat flux* using the estimate from Part (i). Redo the simulation. Can you recover the outlet temperature from the original simulation with an imposed constant temperature at the bottom? If not, try to explain why.