

PLEASE READ THIS PAGE FIRST

A statement on collaboration is required for all reports for the projects, including those that are done independently without any collaboration. See instruction below on how to prepare the statement. Please read the rules before forming any collaboration for the projects. 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 projects:

(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 assignment. See additional instruction below for the required content of the statement.

(2) For each assignment, 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 allowed. To certify that a collaboration is legitimate, the submitter's contribution to the collaborative effort must be documented in the statement on collaboration.

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 process of completing the 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 contribution by the submitter to the collaborative effort. Example:

Name of collaborator: <i>Joe Smith</i>	
Task(s), specific detail	Contribution to collaborative effort
<i>Task 1</i>	<i>Worked on local mesh refinement together</i>
<i>Task 3</i>	<i>Worked with collaborator to setup report files for the transient simulation</i>

MAE 494/598 Applied CFD, Fall 2019

Project 1 - Internal Flow with heat transfer (25 points)

Hard copy of report is due at 1:30 PM on the due date. Please follow the rules for collaboration as described in the first 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.**

Task 1

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 plate (colored in green in Fig. 1) of the main cylinder is externally maintained at a constant 55°C . The temperature of the water entering the inlet is fixed at 10°C . As cool water flows through the tank, it is warmed up by the hot plate at the bottom. Thus, the temperature of the outflow generally exceeds 10°C . This task will use ANSYS-Fluent to compute the temperature of the outflow at the steady state, taking into account the buoyancy effect due to density variation of water.

The key geometric parameters are defined in Figs. 2 and 3. Use $H = 0.8$ m, $D = 0.6$ m, $d = 0.06$ m, $L = 0.15$ m, $Z_1 = 0.6$ m, and $Z_2 = 0.2$ m. In all descriptions for this task, 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. Gravity is set to point in the negative y -direction, as shown in Fig. 1. (You may use a different definition of the directions, as long as consistency is maintained through the task.)

Set the boundary conditions to **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 opening of inlet) and inlet temperature is set to a constant 10°C . The system is filled with water. The viscosity, specific heat, and heat conductivity of water are set to the default constant values from Fluent database, but density is set to **Boussinesq** to allow variation of density with temperature. Energy equation is turned on, and turbulence k -epsilon model is chosen for the simulations. Use default for the settings of the model and boundary conditions related to turbulence, except that “full buoyancy effect” is turned on for the k -epsilon model. (See next page for the setup of initialization.)

For all the settings related to Boussinesq approximation, please choose the “mid-point” of temperature (i.e., the average of the maximum and minimum temperatures in the system) as the *operating temperature*. Other related settings (for *operating density*, *thermal expansion coefficient*, etc.) follow accordingly. Note that this might not be the optimal choice since the averaged temperature at steady state might deviate from the “mid-point” temperature. Nevertheless, we specifically require the use of mid-point temperature for the setup such that there is a uniform basis for grading.

Recommendations on geometry and mesh: For Task 1, 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.) We do not designate a specific configuration of mesh to use for the simulations. Nevertheless, it is essential for the mesh to have enhanced resolution near the wall, *including the bottom plate* (apply local mesh refinement if necessary), and within the two side pipes.

Task 1(a)

Use the geometry and boundary conditions as described in the preceding page. Turn **gravity** on and set it to -9.81 ms^{-2} in y -direction (see Fig. 1a). Follow the instructions given in the preceding page, set density of water to **Boussinesq** to allow buoyancy-driven thermal convection. Use **pressure-based solver** and seek **steady solution**.

Initialize the system with zero velocity, zero gauge pressure, and uniform temperature of $T = 10^\circ\text{C}$ for the water in the tank. Set the initial values of turbulence parameters to zero as well. [Note: Leave "Compute from ..." blank when performing the initialization. DO NOT designate "compute from inlet".] Since this task only seeks the steady solution, the detail of initialization would not affect the final steady state. Nevertheless, we specify the particular way to initialize the system such that there is a uniform basis for grading for deliverable (iii) as given below.

To explain the deliverables, let us first define the outlet temperature, T_{out} , as the area-weighted average of static temperature for the outlet,

$$T_{out} = \frac{1}{A} \iint_A T dA, \quad \text{Eq. (1)}$$

where A is the surface area of the outlet and the integration is performed over the surface of the outlet.

The calculation should be run until the variation of T_{out} is less than 0.05°C over the span of 100 iterations. For example, if $T_{out} = 300.25^\circ\text{K}$ at the 1200th iteration and 300.29°K at the 1300th iteration, the solution at the 1300th iteration is deemed acceptable. ***This requirement is the convergence criterion for this task and it supersedes the arbitrary default (residual-based) convergence criterion in Fluent.***

The deliverables are:

- (i) A contour plot of the y -velocity (not to be confused with *velocity magnitude*) in the *plane of symmetry* for the steady solution. Note that by our definition y is the direction parallel to the gravity vector.
- (ii) A contour plot of *temperature* in the *plane of symmetry* for the steady solution. [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) The value of *outlet temperature* at steady state. In addition, a *line plot of the outlet temperature, T_{out} , as a function of the number of iterations.*

Task 1(b)

Repeat Task 1(a) except that gravity is set to -1.62 ms^{-2} in y -direction. (This is the strength of gravity at the surface of the Moon.) All requirements and deliverables are the same as Task 1(a).

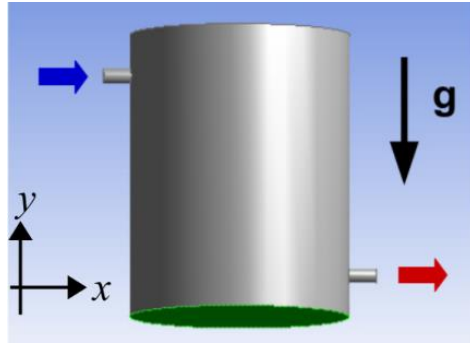


Fig. 1 The general setup of the prototypical water heater. The bottom plate is in green. Inlet and outlet are indicated by blue and red arrows, respectively.

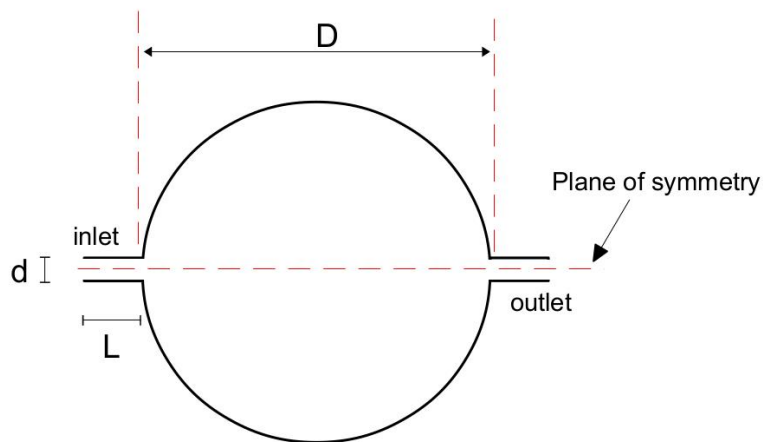


Fig. 2 Top view of the water tank system with a circular cross section. Key parameters: D is the diameter of the main cylinder; d is the diameter of both side pipes; L is the length of both side pipes.

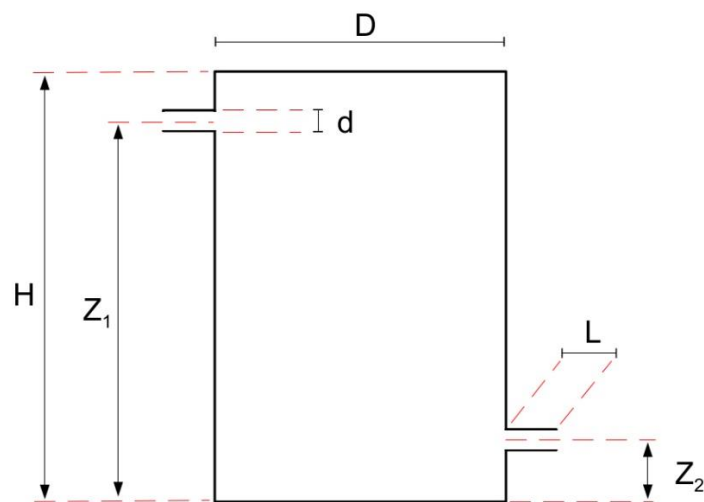


Fig. 3 The vertical cross section of the water tank along its plane of symmetry. Key parameters: H and D are the height and diameter of the main cylinder; Z_1 and Z_2 are the heights of the centers of the side pipes; L is the length of both side pipes; d is the diameter of both side pipes.

Task 2

Background: The water heater used in Task 1 is very primitive and inefficient. In practice, the water heaters for household uses have very different designs. One design is to run water through a coiled pipe with heated wall. This allows water to heat up quickly within limited space. We will use Fluent to simulate a prototype of the coiled-pipe system filled with *water*. Unlike Task 1, to keep the physical processes simple we revert the setting to *constant density* and no gravity. The viscosity and heat conductivity of water are also set to constant, all using the default values in Fluent database.

Consider a helical pipe with its center traced by the equation of a helical curve,

$$X(t) = R \cos(t) \quad \text{Eq. (2)}$$

$$Y(t) = R \sin(t) \quad \text{Eq. (3)}$$

$$Z(t) = C t, \quad \text{Eq. (4)}$$

where “ t ” is not time but a dummy parameter to help tracing the curve. For this task, the curve is traced from $t = 0$ to $t = 10\pi$ (radian), with $R = 0.3$ m and $C = 0.15/(2\pi)$ m. (The helix is “raised” by 0.15 m every revolution.) The helical curve is shown in Fig. 4. The 3-D helical pipe (with the helical curve as its center) with a circular cross section with radius = 4 cm (diameter = 8 cm) is shown in Fig. 5.

With the geometry as shown in Fig. 5, the openings of the inlet and outlet of the helical pipe are both perpendicular to the y -direction. The boundary conditions are *velocity inlet* for the inlet, and *outflow* for the outlet. For all runs for this task, we set the inlet temperature to $T = 300^\circ\text{K}$. This value is not important, as we will be interested in only the temperature *difference* between inlet and outlet. For all runs, use *Laminar* model and seek *steady solution*. In all cases, the wall of the helical pipe is heated by an external source that supplies a uniform energy input of 500 W m^{-2} at the wall.

Perform 4 runs with the inlet velocity set to 0.01, 0.02, 0.04, and 0.08 m/s, in the direction normal to the opening of the inlet (and going *into* the pipe). Other than varying the inlet velocity, all other conditions are identical for those runs. For this task, the detail of initialization is not critical. It suffices to use Hybrid Initialization.

The deliverables for this task are

(1) The values of ΔT for the 4 cases, where $\Delta T = T_{out} - T_{in}$ is the difference between outlet and inlet temperature at steady state. Note that $T_{in} = 300^\circ\text{K}$ while T_{out} is defined by Eq. (1) (i.e., it is the area-weighted average of outlet temperature). In addition, a plot of ΔT vs. inlet velocity and a brief interpretation of the relation between the two variables.

(2) For the case with inlet velocity = 0.04 m/s only, contour plots of *temperature* and *velocity magnitude* over the circular opening of outlet. Please indicate the inner and outer edges of the pipe in the contour plots. Adjust contour intervals as needed to ensure clarity of the plots.

[Remarks: (i) In Fluent, one can use *Parametric Design* to automatically execute multiple runs with a varying parameter. (For this task, it is the inlet velocity.) Using Parametric Design is not required for this task but please feel free to try it. (ii) The ΔT in this task can be estimated by a hand calculation of heat budget without invoking the complicated CFD simulation. While the theoretical estimate is not required for this task, it can be used to check whether the results of the CFD simulations make sense.]

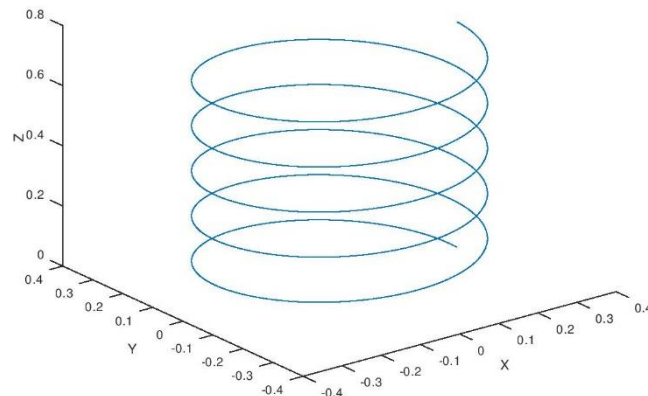


Fig. 4 The helical curve.

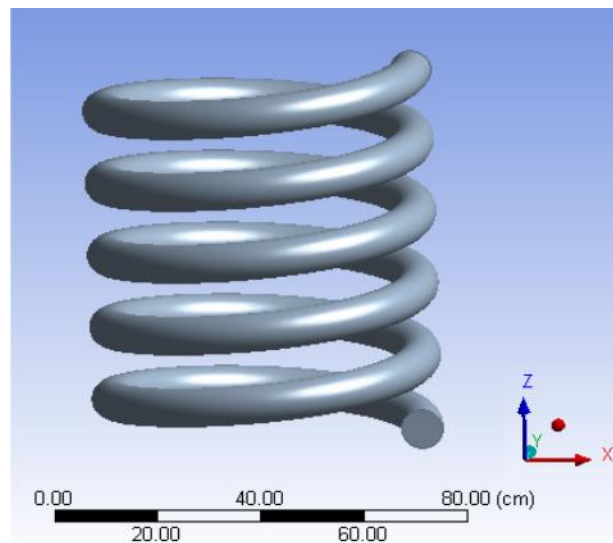


Fig. 5 The 3-D helical pipe.

Task 3

Background: In Task 1, the Boussinesq approximation retains the effect of density variation on buoyancy in the z-momentum equation but otherwise ignores volumetric expansion in the continuity and thermodynamic energy equations. This allows us to keep the pressure-based solver (normally for an incompressible flow) which simplifies the calculation. In this task, we consider a system in which the effect of volumetric expansion is important, which warrants the use of *density-based solver*.

Consider a simple cylindrical chamber, shown in Fig. 6, that is filled with air. The main cylinder has a height of 20 cm and diameter of 20 cm. A cylindrical side pipe has a length of 10 cm and diameter of 5 cm. It is inserted into the middle of the main cylinder. There is only one opening of the system at the end of the side pipe.

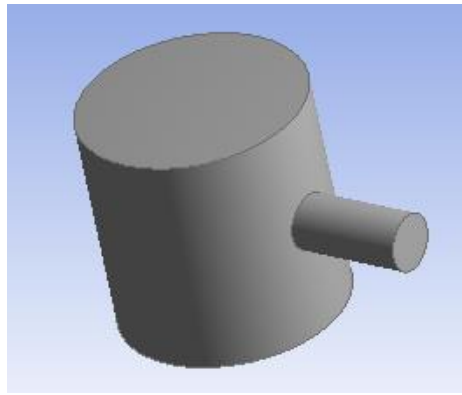


Fig. 6 The heated chamber for Task 3

The top and bottom plates of the main cylinder are heating panels, each supplying a uniform energy input of 10 W m^{-2} . Starting from an initial state with zero velocity inside the chamber, as the air warms up, it expands and forces a flow out of the chamber. The area-weighted average of the velocity at the outlet is defined as

$$V_{out} = \frac{1}{A} \iint_A v \, dA, \quad \text{Eq. (5)}$$

where v is the *outward* velocity in the direction normal to the opening of the side pipe, and A is the surface of the opening. We expect V_{out} to be positive as the chamber warms up.

For the key simulation, choose *density-based solver*, ignore gravity, and set the density of air to ***ideal gas***. Perform a ***transient*** simulation from $t = 0$ to $t = 10 \text{ s}$ with the initial state (at $t = 0$) set to a uniform $T = 20^\circ\text{C}$, zero velocity, and zero gauge pressure inside the chamber. (This will require Standard Initialization; Leave “compute from...” blank. DO NOT designate “compute from inlet/outlet”.) The outlet is set as a *pressure outlet* with zero gauge pressure and backflow temperature of 20°C . It suffices to use *Laminar* model for the simulation. Although the system is symmetric between top and bottom, and left and right, for this task please *use the full geometry*.

The deliverables are

- (i) A line plot of V_{out} vs. time from $t = 0$ to $t = 10 \text{ s}$.
- (ii) Contour plots of pressure, velocity magnitude, and temperature along the “vertical” plane of symmetry. This is the plane that splits the system into left and right halves and cuts a rectangular cross section through the main cylinder. (The other plane of symmetry would cut a circular cross section through the main cylinder. Do not choose that plane for the plots.)

[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 iterations 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.

The system used in Task 3 has two-fold symmetry between top and bottom, and left and right. The two planes of symmetry are described in deliverable (i) of Task 3. Given so, by invoking symmetry one could run the simulation using only the top (or bottom) half, or only the right (or left) half. Given the two-fold symmetry, one could push the idea further and run a simulation using only **a quarter** of the system (e.g., the bottom right quarter) by *declaring two planes of symmetry*. Try to use this setup and repeat the run in Task 3. **The deliverables are the same as (i) and (ii) given in Task 3 (please comment on whether the new run recovers the results in Task 3), but this task has an additional requirement:**

(iii) Show the geometry/mesh of the system (which should consist of only a quarter of the original system in Task 3) for the new simulation.