

PLEASE READ THIS PAGE FIRST

A statement of collaboration is mandatory and must be placed in the first page of the report. If no collaboration occurred, simply state "**No collaboration**". This certifies that the person submitting the report *has not helped anyone or received help from anyone* in the process of completing the assignment. Collaboration is allowed subject to the following rules.

(1) 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 the collaboration.**

(2) In a legitimate collaboration, each individual must make a meaningful 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 of collaboration.

(3) Even with collaboration, the write-up of the report must be done independently. Verbatim copying of a collaborator's report, in whole or in part, is not allowed.

The statement of collaboration

If no collaboration occurred, simply state "**No collaboration**". If collaboration occurred, provide the name of collaborator (only one allowed), a list of the task(s) on which collaboration occurred, and a description 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 determine time step size for the transient simulation</i>

The statement should be placed in the first page of the report.

A submission without the statement of collaboration will be rejected.

MAE 460/560 Applied CFD, Fall 2022 Project 1 - Internal Flow (27 points)

Please upload the report to Canvas as a single PDF file. Please follow the rules on collaboration described in the preceding page.

All tasks, except Task 1c, are for both MAE460 and MAE560. **Task 1c is for MAE560 only.**

Task 1: Internal flow with thermal convection (buoyancy effect)

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 45°C . The temperature of the water entering the inlet is fixed at 15°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 15°C . This task will use ANSYS-Fluent to compute the temperature of the outflow, 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. 1. (You may use a different definition of the directions, as long as consistency is maintained through the task.)

We recommend using the full geometry for this task.

Set the boundary conditions to **velocity inlet** for the inlet and **outflow** for the outlet. Set inlet velocity to a constant 0.06 m/s (for the velocity component normal to the opening of inlet) and inlet temperature to a constant 15°C . The system is filled with **water**. The viscosity, specific heat, and thermal conductivity of water are set to the default constant values from Fluent database, but density is set to **Boussinesq** to emulate the buoyancy effect due to variation of density with temperature. **Energy equation** is turned on, and turbulence **k -epsilon** model is chosen for the simulation. Use the default setting for the k -epsilon model; Use the default setting for turbulence parameters in the boundary conditions for the inlet. The detail on the initialization for turbulence parameters is given in Task 1a.

To set up the Boussinesq method, please choose the “mid-point” of temperature (i.e., the average of the maximum and minimum temperature in the system) as the **operating temperature**. The related settings for **operating density** and **thermal expansion coefficient** follow accordingly. Note that this might not be the optimal choice since the volume-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 so as to have a uniform basis for grading. The “NIST formula” for density of water as a function of temperature will be provided separately. Please use the formula to compute operating density and thermal expansion coefficient from operating temperature.

Recommendations on geometry and mesh: For this task, we recommend using the full geometry of the water tank. While we do not impose a strict specification for the mesh, the general recommendation given in HW1 still applies. For example, mesh resolution should be higher within the two side pipes compared to the interior of the main cylinder. It is also relevant to have enhanced resolution near the bottom plate; Apply local mesh refinement if necessary.

Task 1a: Steady solution with gravity turned on

Use the geometry and boundary conditions described in the preceding page. Turn **gravity** on and set it to -9.81 ms^{-2} in y -direction (see Fig. 1(a)). Set relevant parameters for **Boussinesq** approximation to allow buoyancy-driven thermal convection. Use **pressure-based solver** and seek **steady solution**.

Initialize the system with zero gauge pressure and uniform temperature of $T = 15^\circ\text{C}$ for the water in the tank. Set initial velocity to $(u, v, w) = (0.06, 0, 0) \text{ m/s}$. (Essentially, the initial velocity and temperature in the tank are the same as those imposed at inlet.) Initialize *turbulence kinetic energy* (k) to $1 \times 10^{-5} \text{ m}^2 \text{ s}^{-2}$, and *turbulence dissipation rate* (*epsilon*) to $1 \times 10^{-6} \text{ m}^2 \text{ s}^{-3}$. [Otherwise, leave "Compute from ..." blank when performing the initialization. DO NOT designate "compute from inlet".] We specify this particular way to initialize the system in order to have a uniform basis for grading on deliverable (D4).

To explain the deliverables, we 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 0.1°C over the span of 100 iterations. For example, if $T_{out} = 300.25^\circ\text{K}$ at the 1200th iteration and 300.34°K at the 1300th iteration, the solution at the 1300th iteration is considered acceptable. ***This requirement is the convergence criterion for this task. It supersedes the arbitrary default (residual-based) convergence criterion in Fluent.***

The deliverables are:

(D1) A plot of the mesh along the plane of symmetry, and a statement indicating the values of *operating temperature*, *operating density*, and *thermal expansion coefficient* used for the simulation.

(D2) A contour plot of the *y-velocity* (not to be confused with *velocity magnitude*) in the *plane of symmetry* for the steady solution. (See Fig. 1 for the definition of y -direction.)

(D3) A contour plot of *temperature* in the *plane of symmetry* for the steady solution. [Note: Since there is always a tight temperature gradient near the bottom, in order to clearly show the key feature of the "cool waterfall" coming down from the inlet it is recommended that the contour intervals be manually adjusted.]

(D4) The value of *outlet temperature* (T_{out} as defined in Eq. (1)) at the steady state. This number must be clearly written out. In addition, *a line plot of the outlet temperature, T_{out} , as a function of the number of iterations.*

Task 1b: Steady solution with gravity turned off

Repeat Task 1a except that gravity is turned off. This effectively suppresses the buoyancy effect. The

required convergence criterion remains the same as Task 1a. Namely, the calculation should be run until the variation of T_{out} is less than 0.1°C over the span of 100 iterations. The deliverables are:

(D5) A contour plot of the x -velocity (not to be confused with the y -velocity for D2) in the *plane of symmetry* for the steady solution.

(D6) A contour plot of *temperature* in the *plane of symmetry* for the steady solution. [See the remarks after D3 on proper adjustments of contour intervals.]

(D7) The value of *outlet temperature* (T_{out} as defined in Eq. (1)) at the steady state. This number must be clearly written out.

Task 1c Transient solution (MAE560 only)

Repeat the setting in Task 1a but now seek **transient** solution. Initialize the system at $t = 0$ in the same way as Task 1a. Namely, set initial velocity to $(u, v, w) = (0.06, 0, 0)$ m/s, temperature to 15°C , and gauge pressure to 0. Initialize *turbulence kinetic energy* to $1 \times 10^{-5} \text{m}^2 \text{s}^{-2}$, and *turbulence dissipation rate* to $1 \times 10^{-6} \text{m}^2 \text{s}^{-3}$. [Leave "Compute from..." blank when performing the initialization. DO NOT designate "compute from inlet".] Run the transient simulation to $t = 1$ hour.

The deliverables are

(D8) A statement that indicates your choice of (i) *time step size*, and (ii) *maximum number of iterations per time step*, for the transient simulation.

(D9) A plot of *outlet temperature* $T_{out}(t)$ (defined by Eq. 1) as a function of time, over the range of $0 \leq t \leq 1$ hour. The abscissa of this plot should be *time*, not "number of iterations".

(D10) Contour plots of y -velocity in the *plane of symmetry* at $t = 15$ minutes and $t = 1$ hour from the transient simulation.

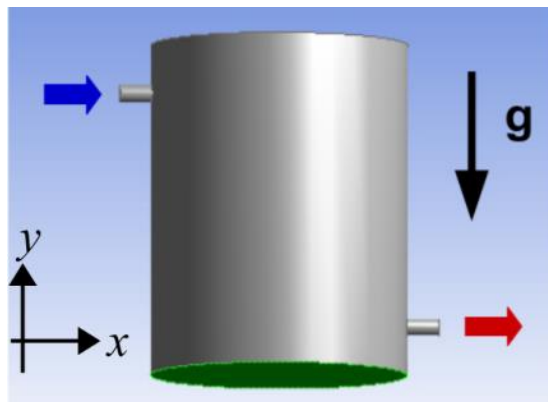


Fig. 1 The setting of the water tank used for Task 1. The bottom plate is in green. Inlet and outlet are indicated by blue and red arrows, respectively. Gravity vector is shown as the bold black arrow.

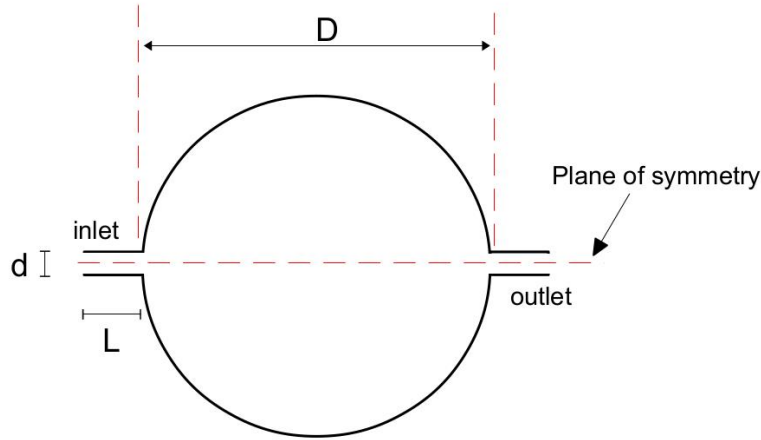


Fig. 2 Top view of the water tank system with a circular cross section. Key parameters: D is the diameter of the main cylinder; d is the diameter of both side pipes; L is the length of both side pipes.

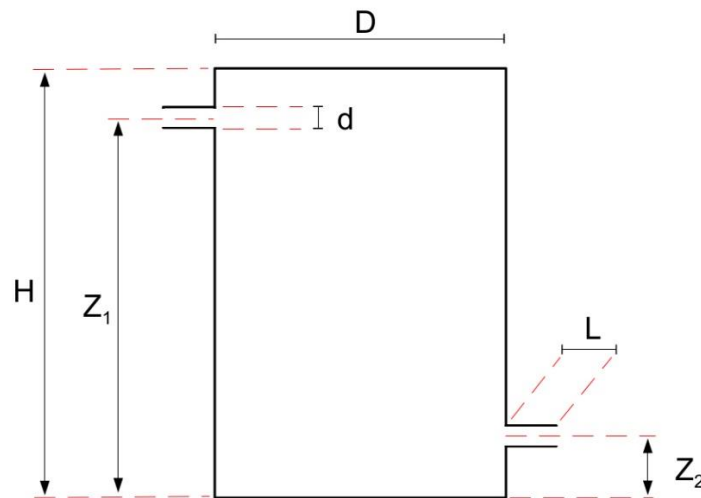


Fig. 3 The vertical cross section of the water tank along its plane of symmetry. Key parameters: H and D are the height and diameter of the main cylinder; Z_1 and Z_2 are the heights of the centers of the side pipes; L is the length of both side pipes; d is the diameter of both side pipes.

Task 2: Internal flow with heat source

Background: The water heater used in Task 1 is very primitive and inefficient. In practice, water heaters for household applications have very different designs. One of which 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 the flow through a prototype of the coiled-pipe system. Unlike Task 1, to keep the physical processes simple we revert the setting to *constant density* (therefore, gravity can also be turned off). The viscosity, specific heat, and thermal conductivity of the fluid are also set to constant, all using the default values from Fluent database.

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

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

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

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

where τ is a dummy parameter to help tracing the curve. For this task, the curve is traced from $\tau = 0$ to $\tau = 12\pi$ (radian), with $R = 0.3$ m and $C = 0.15/(2\pi)$ m. (The helix is raised by 0.15 m after completing a full circle.) The helical curve is shown in Fig. 4. The 3-D helical pipe (with the helical curve running through its center) with radius = 4 cm (diameter = 8 cm) is shown in Fig. 5.

Task 2a: Flow with heated wall

For this task, choose **water** as the fluid for the system. Set all physical parameters (density, specific heat, viscosity, thermal conductivity) of the fluid to constant using the values from Fluent database. 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 in this task, 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 700 Wm^{-2} at the wall.

Perform 4 runs with the inlet velocity set to 0.008, 0.016, 0.032, and 0.064 m/s, in the direction normal to the opening of the inlet. 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 are:

(D11) 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, make a plot of ΔT vs. inlet velocity and provide a brief interpretation of the relation between the two variables.

(D12) For the case with inlet velocity = 0.016 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.

Task 2b: Viscous heating

In general, viscous dissipation of kinetic energy will lead to an internal heating. This effect is negligible for the system considered in Task 2a. To demonstrate the effect, we keep the setting of Task 2a except the following:

(i) Turn off heating on the wall. In other words, change the heat flux on the wall from 700 Wm^{-2} back to 0. This will simply mean that the wall is thermally insulated without any external heat source/sink.

(ii) Use **engine oil** instead of **water** as the fluid in the pipe. Set all physical parameters for engine oil to constant using the values from Fluent database. (The reason of this setting is that engine oil has a much higher viscosity compared to water.)

(iii) Turn on **viscous heating** for the Laminar model. (Note that the default is “off”. This will be the only task with viscous heating turned on.)

With the setting described above, seek *steady solution* for 3 cases with inlet velocity set to 1, 2, and 4 m/s, respectively. The deliverables are

(D13) The values of ΔT for the 3 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, make a plot of ΔT vs. inlet velocity and provide a brief interpretation of the relation between the two variables.

(D14) For the case with inlet velocity = 2 m/s only, a contour plot (in panoramic view) of temperature over all exterior surfaces (wall, inlet, outlet) for the whole apparatus.

Remarks:

(i) Since the key deliverable of Task 2a and 2b is the outlet temperature, a sufficient number of iterations should be performed to ensure that T_{out} has reached an almost steady value. For example, a criterion similar to that required for Task 1 could be used to control the termination of iteration. Using the default (residual-based) termination criterion in Fluent will likely lead to an early termination of the iteration process, causing inaccuracies in the results.

(ii) In Fluent, *Parametric Design* can be used to automatically execute multiple runs with a varying parameter. (For this task, it is the inlet velocity.) You are not required to use Parametric Design for this task but please feel free to explore it.

(iii) The ΔT for Task 2a can be approximately estimated by a hand calculation of the 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 result of the CFD simulation makes sense.

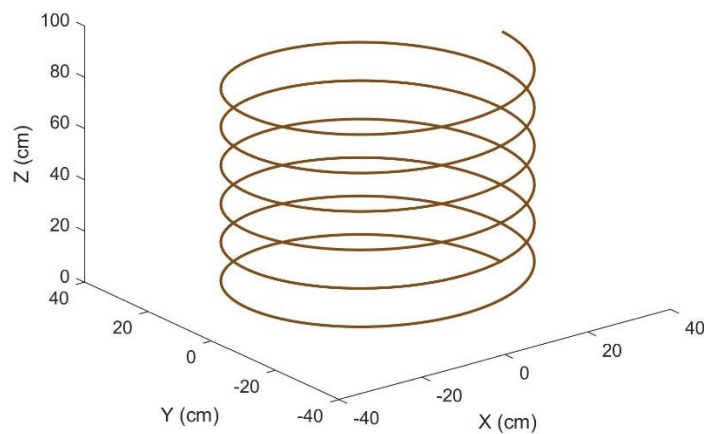


Fig. 4 The helical curve.

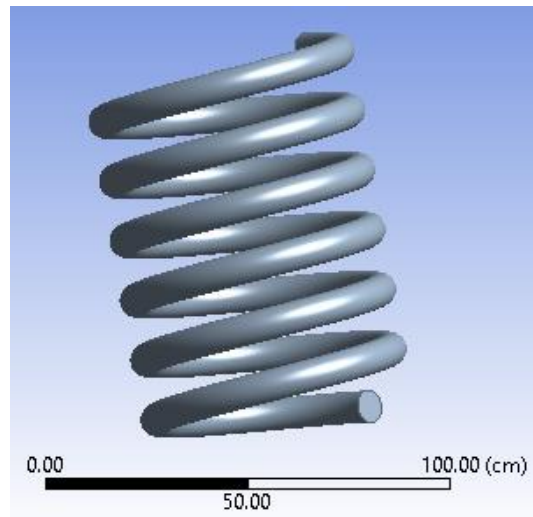


Fig. 5 The 3-D helical pipe.

Task 3: A simple compressible flow

In this task, we will demonstrate the solution for a generally compressible flow using Fluent. Consider the apparatus shown in Fig. 6 which consists of two cylindrical chambers (marked as “A” and “B”) connected by a cylindrical neck. The geometry of the cross section along a plane of symmetry is shown in Fig. 7. The device has only one opening which is marked as “outlet” in Figs. 6 and 7.

The device is filled with *air*. Unlike Task 1 and 2, here the density of air is set to **ideal gas** (such that the thermodynamic variables are related by the ideal gas law, $p = \rho RT$). Use **density-based solver** and seek **transient** solution. Use **k-omega** model with default setting. Crucially, **energy equation** needs to be turned on to capture the thermodynamic effect due to compressibility.

Set the boundary condition for the outlet as *pressure outlet* with total pressure = 10000 Pa, and backflow temperature = 300K. At $t = 0$, initialize the system with zero velocity and uniform temperature of $T = 300\text{K}$. Set the initial total pressure in chamber “A” (blue area in Fig. 7, excluding the neck) to 10200 Pa. Set the initial total pressure to 10000 Pa elsewhere (white area in Fig. 7) in the device. This makes “A” a pressurized chamber at the initial time. For $t > 0$, we anticipate a flow to be pushed out of chamber “A” into chamber “B” (and part of it eventually going out through the outlet). Initialize *turbulence kinetic energy* (k) = $1 \text{ m}^2 \text{ s}^{-2}$, and *specific dissipation rate* (ω) = 1 s^{-1} .

Important note: The pressure described above is the total value. Unlike Task 1 and 2 where only *gauge pressure* matters, in this task the total pressure is relevant (because of the introduction of ideal gas law and its related thermodynamic effect). In Fluent, the default setting of **operating pressure** is already 1 ATM (101325 Pa). The total pressure is **operating** pressure plus gauge pressure. (Thus, if we set gauge pressure to 0 then the total pressure is already 101325 Pa!) To avoid confusion, for this task the recommendation is to first set **operating pressure** to 0. After that, gauge pressure becomes total pressure. We can then initialize the gauge pressure as 10200 Pa for chamber “A”, and so on.

Run the transient simulation to $t = 0.05 \text{ s}$. The deliverables are

(D15) A statement that indicates your choice of (i) *time step size*, and (ii) *maximum number of iterations per time step*, for the transient simulation.

(D16) A plot of the mesh along the plane of symmetry. (We expect enhanced mesh resolution in the “neck”, etc.)

(D17) Contour plots of *temperature*, *density*, and *z-velocity* (not to be confused with *velocity magnitude*) along the plane of symmetry at two different times: $t = 0.025$ s and $t = 0.05$ s. (Three plots for each time. The z-direction is as indicated in Fig. 6.)

Note: Be careful in labeling the color scales for the contour plots in D17. For example, the variation of density may be very small such that one needs to adjust the number of digits to display to remove ambiguity in labeling.

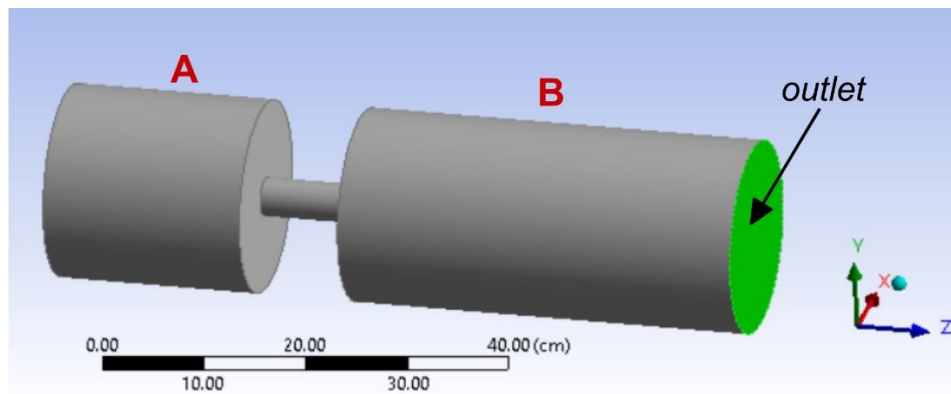


Fig. 6 The device used for Task 3.

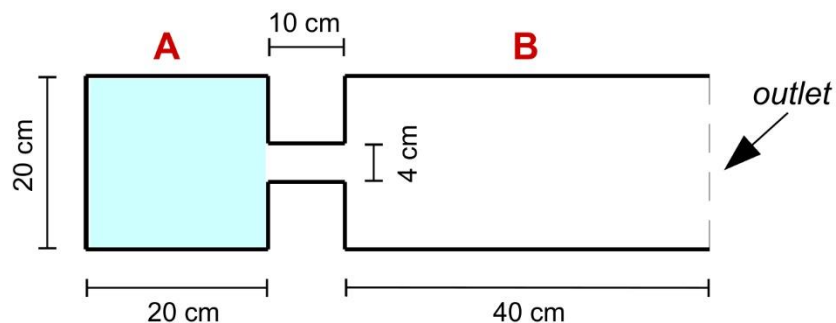


Fig. 7 Cross section (along the plane of symmetry) of the device used for Task 3.