

PLEASE READ THIS PAGE FIRST

A statement of collaboration is required for all reports, including those that are produced independently. See instruction below on how to prepare the statement. Please read the rules before forming any collaboration. 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)-(4) and unless the extent of collaboration is properly disclosed in a statement in the first page of the report. 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 the 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 of collaboration.

(4) 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

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>

There are no specific requirements on the format of the statement. Please do always put it in the first page of the report.

MAE 460/560 Applied CFD, Fall 2021 Project 1 - Internal Flow (25 points)

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

All tasks, except Task 2b, are for both MAE460 and MAE560. **Task 2b is for MAE560 only.**

Task 1: Internal flow with thermal convection

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 40°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. 1. (You may use a different definition of the directions, as long as consistency is maintained through the task.) Task 1a and 1b will use different settings for gravity, as shown in Figs. 1(a)-(b). ***We recommend using the full geometry for this 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 thermal conductivity of water are set to the default constant values from Fluent database, but density is set to ***Boussinesq*** to emulate the effect of variation of density with temperature. Energy equation is turned on, and turbulence *k-epsilon* model is chosen for the simulations. Use the default setting for the *k-epsilon* model. Also, use the default setting for turbulence parameters in the boundary conditions for the inlet. Further details on the initialization for turbulence parameters are given under Task 1a.

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 so as to have a uniform basis for grading. A reference for the “NIST formula” of density of water as a function of temperature will be posted to class website separately.

Recommendations on geometry and mesh: For this task, we recommend using the full geometry for the simulations. We do not have a strict specification of the mesh to use. Nevertheless, the recommendations on mesh for HW1 still apply here. (For example, the mesh resolution should be higher within the two side pipes compared to the main cylinder.) It is also relevant to have enhanced resolution near the bottom plate; Apply local mesh refinement if necessary.

Task 1a: Vertically oriented tank

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. 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 velocity, zero gauge pressure, and uniform temperature of $T = 10^\circ\text{C}$ for the water in the tank. Initialize *turbulence kinetic energy* to $1 \times 10^{-5} \text{ m}^2 \text{ s}^{-2}$, and *turbulence dissipation rate* to $1 \times 10^{-6} \text{ m}^2 \text{ s}^{-3}$. [Note: 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, 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 0.1°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 and 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: Horizontally oriented tank

Repeat Task 1a except that gravity is set to $+9.81 \text{ ms}^{-2}$ in x -direction, as illustrated in Fig. 1(b). (This is equivalent to orienting the water tank horizontally while keeping gravity in the vertical direction.) 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 *velocity magnitude* or y -velocity) 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 the contour interval.]

(D7) 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*.

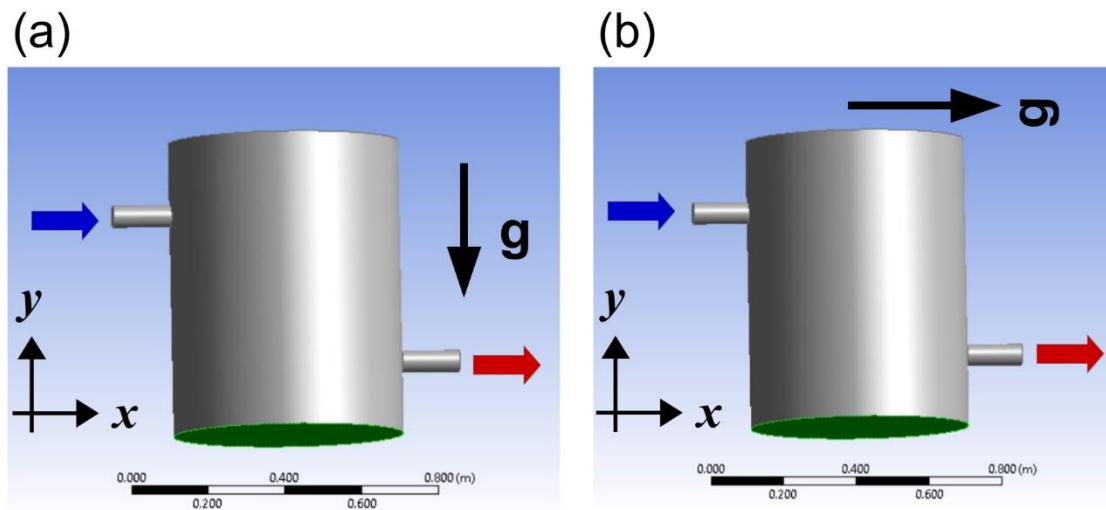


Fig. 1 (a) The setting of the water tank used in Task 1a. The bottom plate is in green. Inlet and outlet are indicated by blue and red arrows, respectively. Gravity vector is shown as the bold black arrow.

(b) Same as (a) but for the setting in Task 1b.

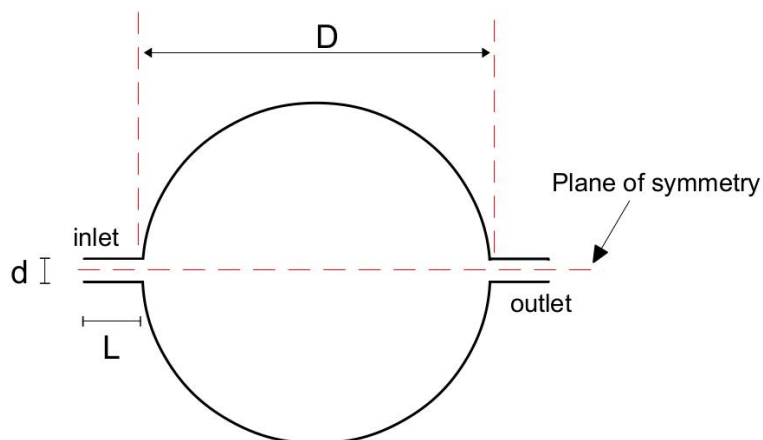


Fig. 2 Top view of the water tank system with a circular cross section. Key parameters: D is the diameter of the main cylinder; d is the diameter of both side pipes; L is the length of both side pipes.

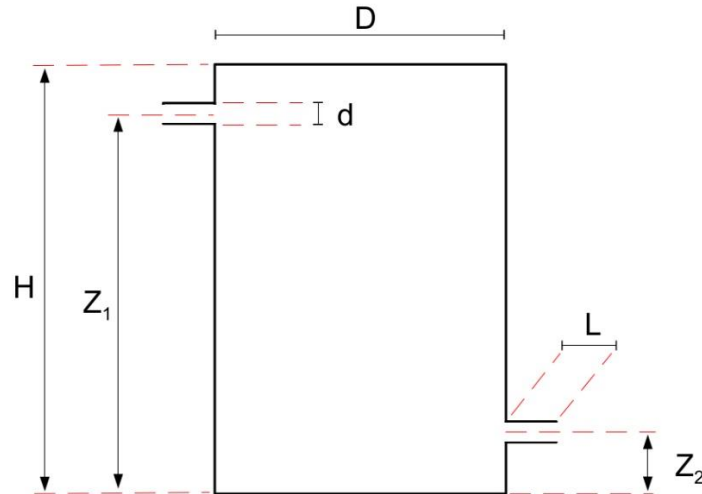


Fig. 3 The vertical cross section of the water tank along its plane of symmetry. Key parameters: H and D are the height and diameter of the main cylinder; Z_1 and Z_2 are the heights of the centers of the side pipes; L is the length of both side pipes; d is the diameter of both side pipes.

Task 2: Internal flow with heat transfer

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 those 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* (therefore, gravity can be turned off). The viscosity, specific heat, and thermal 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,

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

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

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

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 as its center) with a circular cross section with radius = 4 cm (diameter = 8 cm) is shown in Fig. 5.

Task 2a: Steady solution

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, 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 1000 Wm^{-2} at the wall.

Perform 4 runs with the inlet velocity set to 0.005, 0.01, 0.02, and 0.04 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:

(D8) 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.

(D9) For the case with inlet velocity = 0.02 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) Since the key deliverable of this task is the outlet temperature, a sufficient number of iterations should be performed to ensure that T_{out} has reached an almost steady value. In other words, a criterion similar to that required in Task 1 should be used to control the termination of iteration. The default (and arbitrary) convergence criterion based on scaled residual should not be used, as it might prompt the termination of iteration prematurely before T_{out} reaches a near-steady value.

(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 try it.

(iii) The ΔT in this task can be approximately 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 result of the CFD simulation makes sense.

Task 2b: Transient solution (for MAE560 only)

Retaining the setting in Task 2a for the case with inlet velocity = 0.02 m/s, we now consider the transient solution for the system. Initialize the system with a uniform temperature of 300 °K at $t = 0$. Run a transient simulation to $t = 20$ minutes (1200 seconds). To explain deliverable (D10), we define the volume-averaged temperature as

$$T_{vol} = \frac{1}{V} \iiint_V T dV, \quad \text{Eq. (5)}$$

where V is the volume of water for the entire system. The deliverables are:

(D10) A plot of T_{out} and T_{vol} as a function of time for the transient solution over $0 \leq t \leq 20$ minutes. Here, T_{out} is the averaged outlet temperature as given in Eq. (1), and T_{vol} is defined in Eq. (5). The two curves should be put in the same plot.

(D11) A statement that indicates your setup of (i) *time step size*, and (ii) *number of iterations per time step*, for the transient simulation.

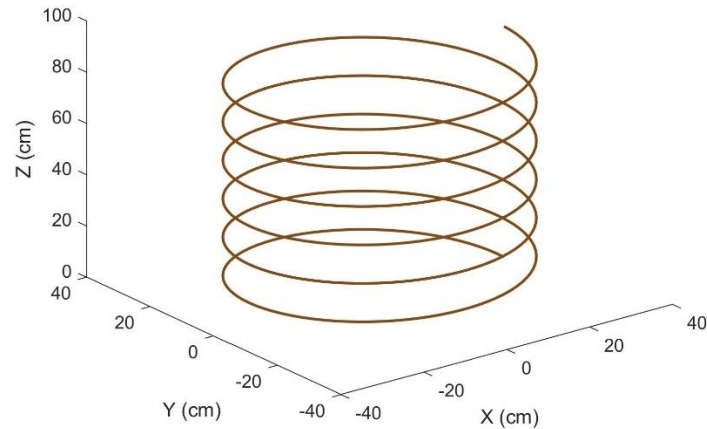


Fig. 4 The helical curve.

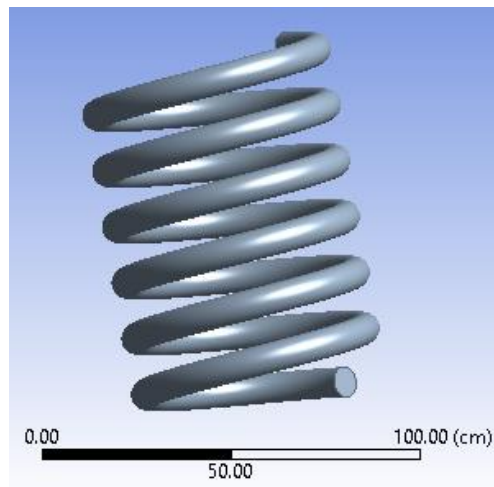


Fig. 5 The 3-D helical pipe.

Task 3: Compressible flow

This task is essentially a tutorial for an example that requires the use of *density-based solver* in Fluent.

Task 3a: High-speed flow in a 2D nozzle

Consider a flow in a 2-D nozzle as illustrated in Fig. 6 (not drawn to scale). The system is symmetric with respect to the x -axis. The profile of the wall of the nozzle is given as

$$F(x) = 0.3 + 0.1 [\tanh(10x - 13) - \tanh(10x - 7)], \quad 0 \leq x \leq 2, \quad \text{Eq. (6)}$$

as shown in Fig. 7. The unit of x and $F(x)$ is meter. The system is filled with air. A higher pressure is imposed at the left opening and a lower pressure at the right opening. The pressure difference drives the flow through the nozzle. The anticipated flow velocity for this system is high enough to push Mach number to order 1, which requires the setup for a compressible flow:

- (1) Select *Density based* solver and seek *steady* solution.
- (2) Set the density of air to *ideal gas* to allow density to vary significantly with pressure and temperature. Turn *Energy equation* on.
- (3) Select *Inviscid* model.
- (4) Set *Operating pressure* to 0. (This means the values of pressure imposed at the inlet and outlet are those of absolute pressure.)
- (5) Set the left opening as a *pressure inlet* and impose
 - (i) *Gauge total (stagnation) pressure* = 101360 Pa
 - (ii) *Supersonic/Initial gauge pressure* = 98910 Pa
 - (iii) *Total temperature* = 500°K.
- (6) Set the right opening as a *pressure outlet* and impose *Gauge pressure* = 5000 Pa. Set the temperature of backflow to 200°K.
- (7) For *Reference values*, choose "*Compute from pressure inlet*".
- (8) For *Solution initialization*, choose "*Standard*" and "*Compute from pressure inlet*".

Since this is an inviscid simulation, there is no need to place highly concentrated mesh near the wall. In other words, the nodal points can be distributed more uniformly. (One way to do it is to choose *Face Meshing* and set a uniform *Element size*.) You may use full-nozzle or half-nozzle geometry, the latter by invoking symmetry.

From the steady solution, the deliverables are:

- (D12) Contour plots of *x-velocity* and *static temperature*.
- (D13) Line plots of *x-velocity* and *Mach number* along the *x-axis* (i.e., the axis of symmetry of the nozzle).

Task 3b: Demonstrating the effect of compressibility

To demonstrate the importance of fully resolving the compressibility effect in Task 3a, let's keep the setting of that task but artificially set density of air to constant (using the default value in Fluent database). This setting is entirely artificial such that the steady solution will not converge. (With the inviscid model, the total kinetic energy will keep increasing indefinitely.) Nevertheless, for illustrative purpose, take this setting and run the steady solution to 1000 iterations. The deliverables are:

- (D14) A line plot of *x-velocity* along the *x-axis* (i.e., the axis of symmetry of the nozzle). In addition, a brief discussion on the difference in the velocity between the line plots in (D13) and (D14). Briefly explain why suppressing the effect of compressibility leads to the erroneous structure of velocity in (D14).

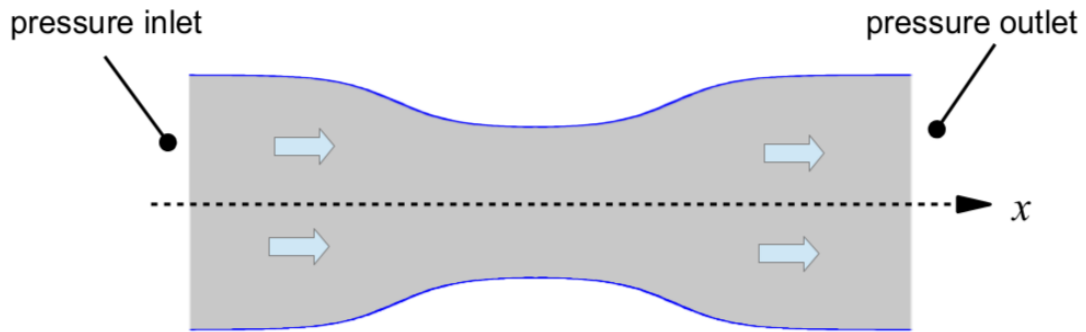


Fig. 6 A sketch of the 2-D nozzle considered in Task 3. Not drawn to scale.

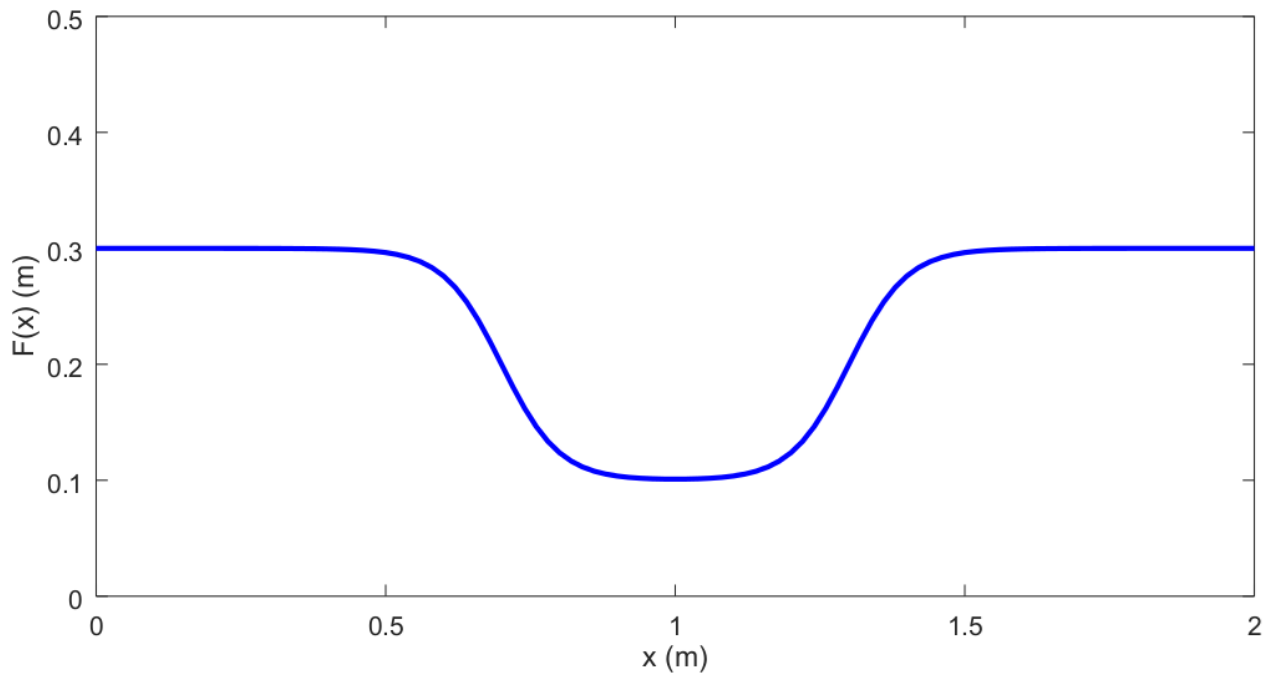


Fig. 7 The profile of the wall of the nozzle. The abscissa is the x -axis. The nozzle is 2 m long. [Note that the half-width of the nozzle is not exactly 0.3 at $x = 0$ and 2. Please use the formula in Eq. (6) to ensure that there are no gaps in the geometry.]