

MAE 598/494 Applied CFD, Fall 2019 Project 5 – Compressible flow (5 points)

Hard copy of report is due **at the start of final exam**. Please follow the rules for collaboration as described in the cover page of the document for Project 1. A statement on collaboration is mandatory for all. All tasks in this project are for both MAE598 and MAE494.

Task 1

(a) Consider a flow in a 2-D nozzle as illustrated in Fig. 1. The system is symmetric with respect to the x -axis. The profile of the wall of the nozzle is given as

$$F(x) = 0.3 + 0.1 [\tanh(10x - 13) - \tanh(10x - 7)], \quad 0 \leq x \leq 2, \quad \text{Eq. (1)}$$

as shown in Fig. 2. 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.
- (3) Select *Inviscid* model and turn *Energy equation* on.
- (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 put 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 run the case using the full nozzle, or half nozzle by invoking symmetry.

From the steady solution, the deliverables are:

- (i) Contour plots of x -velocity and *static temperature*.
 - (ii) Line plots of x -velocity and *Mach number* along the x -axis (i.e., the axis of symmetry of the nozzle).
- (b) Repeat Task 1a by retaining all of the setups except that the density of air is set to constant (using the default value from Fluent database) to artificially suppress the effect of compressibility on velocity. (If the run is slow to converge, it is acceptable to stop at 1000 iterations.) The deliverables are
- (i) Contour plot of x -velocity
 - (ii) Line plot of x -velocity along the x -axis (i.e., the axis of symmetry of the nozzle).

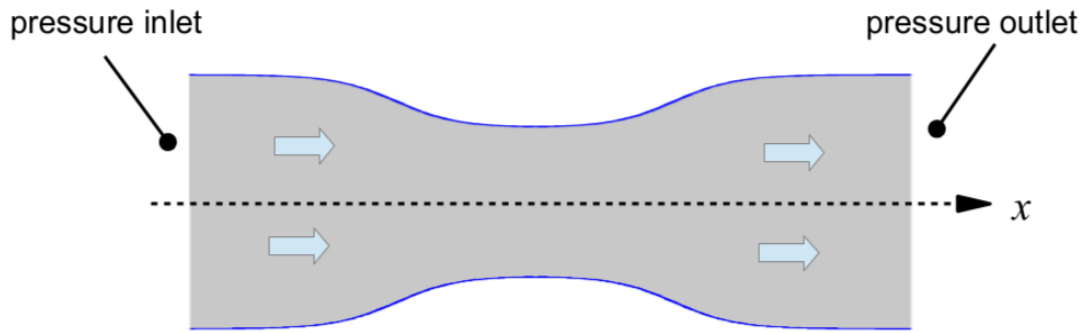


Fig. 1 A sketch of the 2-D nozzle considered in this project. Not drawn to scale.

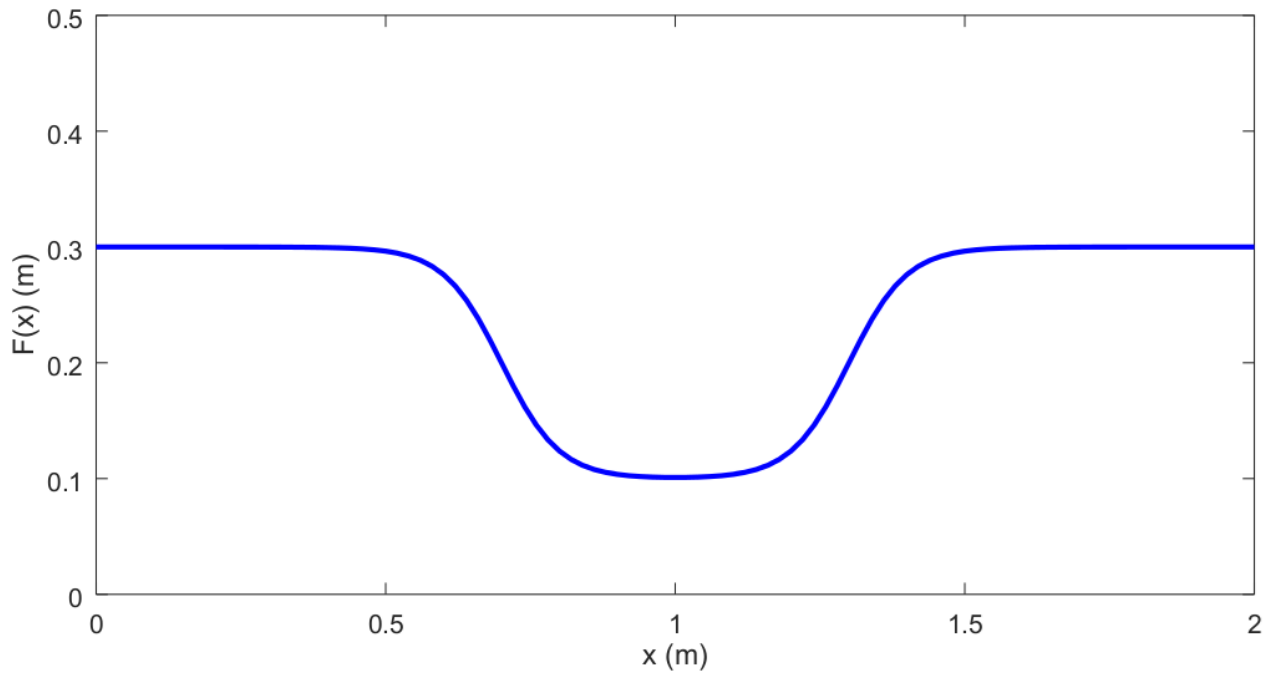


Fig. 2 The profile of the wall of the nozzle. The abscissa is the x -axis. The nozzle is 2 m long. (Note that the half-width of the nozzle is not exactly 0.3 at $x = 0$ and 2. Please use the formula in Eq. 1 to make that there are no gaps in the geometry.)