Project #5 Compressible flow

MAE 598/494 Applied CFD, Fall 2017 Project #5 (5 points)

Task 1 (for both MAE598 and MAE494)

(a) This project considers a generally compressible 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

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

as shown in Fig. 2. <u>The unit of *x* and *F(x)* is meter</u>. 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, justifying the setup for a compressible flow. Consider the following key setups for Ansys-Fluent:

(1) Set the density of air to *ideal gas* to allow density to vary significantly with pressure and temperature.
(2) Select *Inviscid* model and turn *Energy equation* on.
(3) Select *Density based* solver and seek *steady* solution.
(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* = 30°C.
(6) Set the right opening as a *pressure outlet* and impose *Gauge pressure* = 5000 Pa.
(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. Instead, we will try to distribute the nodal points more uniformly by choosing *Face Meshing* and set a uniform *Element size* under *Sizing*. The recommended element size is 0.02 m. For this project, you may run the case using the full nozzle, or half nozzle by invoking symmetry.

Run the simulation and make the following plots from the solution:

(i) Contour plots of *x-velocity*, *density*, 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. Run the simulation and produce the following plots:

(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).
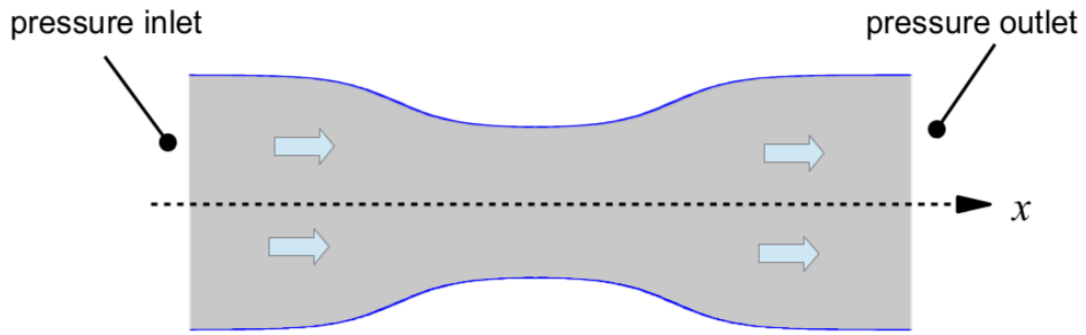
Fig. 1 The geometry of the 2-D nozzle considered in this project. Not drawn to scale.
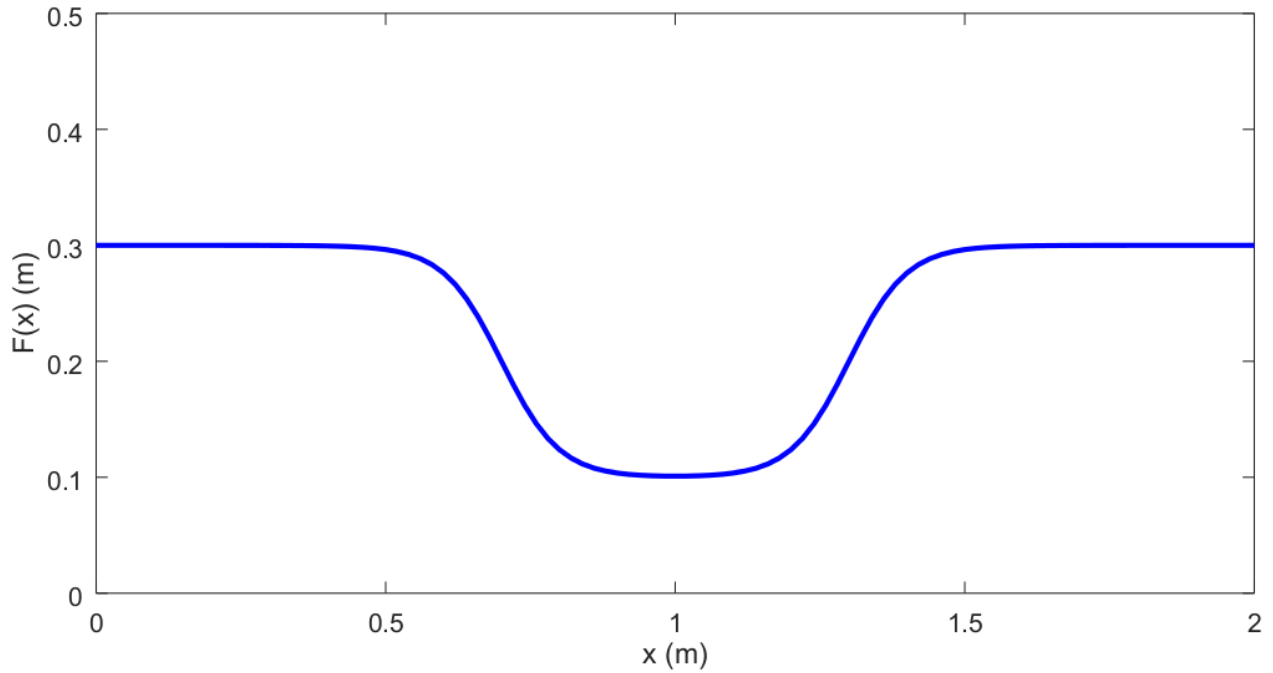

Fig. 2 The profile of the wall of the nozzle. The abscissa is the x-axis. The nozzle is 2 m long. (Note that the half-width of the nozzle is not exactly 0.3 at $x = 0$ and 2.)