MAE 598/494 Applied CFD, Fall 2016 Project #4 (10 points)

Task 1 (Regular task for both MAE598 and MAE494)

This project considers a generally compressible flow in a simple 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

$$F(x) = 0.3 + 0.06 \left[\tanh(10\,x - 9) - \tanh(10\,x - 4) \right], \ 0 \le x \le 1.4 ,$$

as shown in Fig. 2. <u>The unit of x and $F(x)$ is meter</u>. This case differs from those in previous projects in that we will have a *pressure inlet*. The system is filled with air as an *ideal gas* to allow its density to vary significantly with pressure and temperature. The flow velocity is high enough that the Mach number of the system is no longer smaller than order 1, justifying the setup for a compressible flow. Consider the following key setups for Fluent:

(1) Choose *Mapped Face Meshing* and set *Element size* to 0.02 m under "Sizing" to generate mesh.
(2) Select *Inviscid* model.
(3) Turn *Energy equation* on.
(4) Select *Density based* solver and seek *steady* solution.
(5) Choose *air* as the material and set the density of air to *ideal gas*.
(6) Set *Operating pressure* to 0. (This means the values of pressure imposed at the inlet and outlet are those of absolute pressure.)
(7) For the boundary condition at pressure inlet, impose
  (i) *Gauge total (stagnation) pressure* = 101360 Pa
  (ii) *Supersonic/Initial gauge pressure* = 98910 Pa
  (iii) *Total temperature* = 30°C.
(8) For the boundary condition at pressure outlet, impose *Gauge pressure* = 5000 Pa.
(9) For *Reference values*, choose "Compute from *pressure inlet*".
(10) For *Solution initialization*, choose "Standard" and "Compute from *pressure inlet*".

Run the simulation and make the following plots from the solution: (i) Contour plot of density, (ii) Contour plot of static pressure, (iii) Contour plot of x-velocity (not to be confused with velocity magnitude), (iv) Line plot of Mach number along the x-axis. In this plot, the abscissa should be the position along the x-axis and ordinate the Mach number. (v) In addition, make a plot of the mesh. For this project, you may run the case using the full nozzle, or using a half of the nozzle by invoking symmetry.
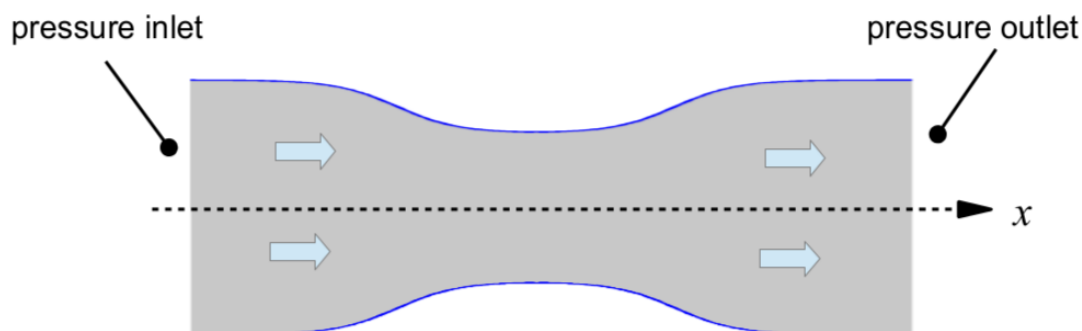

Fig. 1 The geometry of the 2-D nozzle considered in this project. Not drawn to scale.
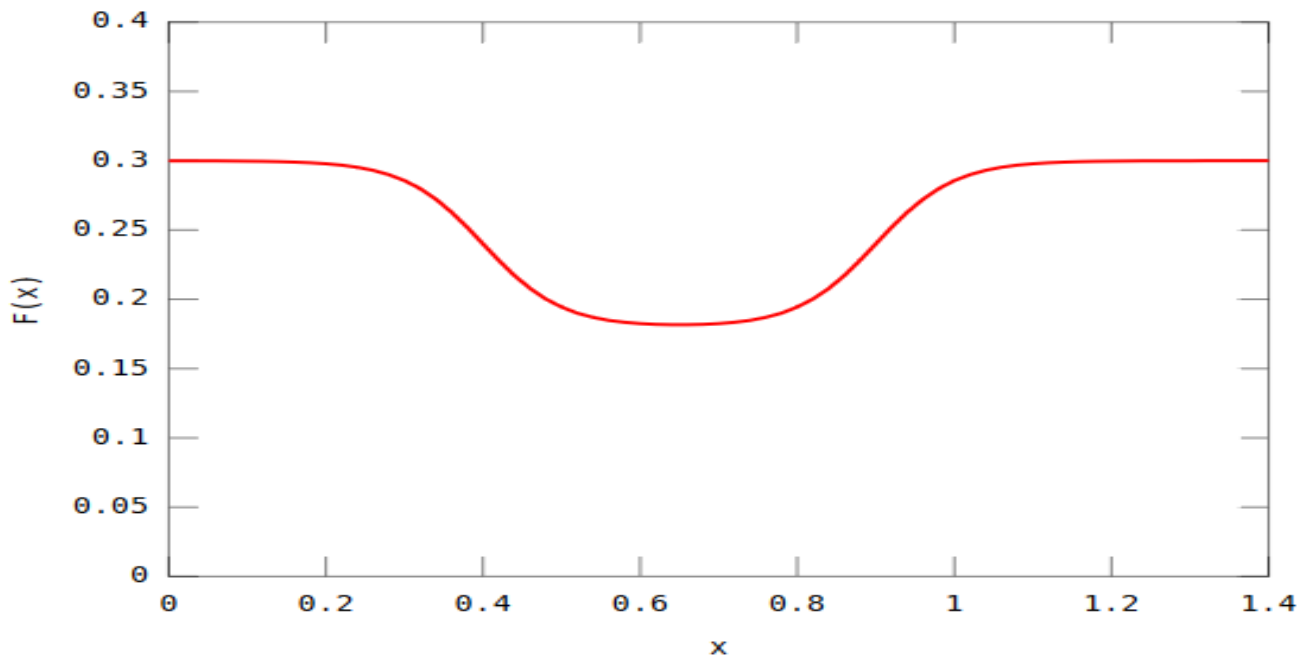
Fig. 2 The profile of the wall of the nozzle. The abscissa is the x-axis. The nozzle is 1.4 m long.