MAE 494/598 Applied CFD, Fall 2018, 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 generally compressible flow in a 3-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

$$F(x) = 0.3 + 0.1\ [\ tanh(10\,x - 16) - tanh(10\,x - 8)\ ]\,, \quad 0 \le x \le 2\,, \qquad \text{Eq. (1)}$$

as shown in Fig. 2. The unit of x and $F(x)$ is meter. The 3-D nozzle can be created by revolving the profile described in Eq. (1) around the x-axis. (Be aware that $F(x)$ is not exactly 0.3 at $x = 0$ and 2.) 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) 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* = 98000 Pa
 (iii) *Total temperature* = 30°C.
(6) Set the right opening as a *pressure outlet.* Impose *gauge pressure* = 5000 Pa and *backflow temperature* = 30°C.
(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 simply set *Element size* (around 2 cm should be sufficient). For this project, the simulations (in Part (a) and (b)) can be performed using the full nozzle, or half nozzle by invoking symmetry. The deliverables are:

(i) Contour plots of *x-velocity* and *static temperature* along the plane of symmetry.
(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. The deliverables are:

(i) Contour plot of *x-velocity* along the plane of symmetry.
(ii) Line plot of *x-velocity* along the *x*-axis (i.e., the axis of symmetry of the nozzle).

[Note: For Task 1b, the setup is rather unphysical but we are mainly interested in using it to demonstrate the contrast between compressible and incompressible settings. For this simulation, the residuals might remain large for a relatively large number of iterations before eventually dropping down. Make sure that a sufficient number of iterations are performed. Otherwise, in the solution excessive backflow (with negative *x*-velocity) might appear near the outlet.]

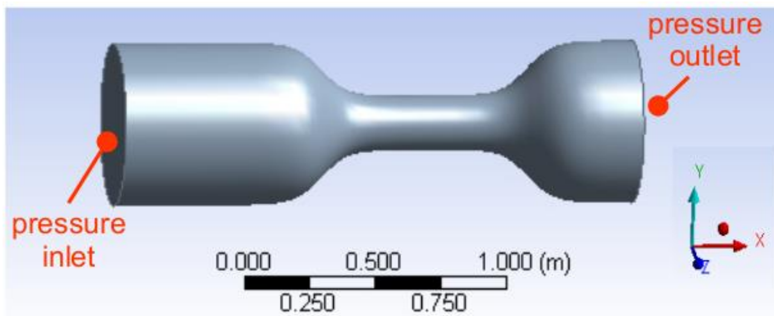

Fig. 1 The geometry of the 3-D nozzle.

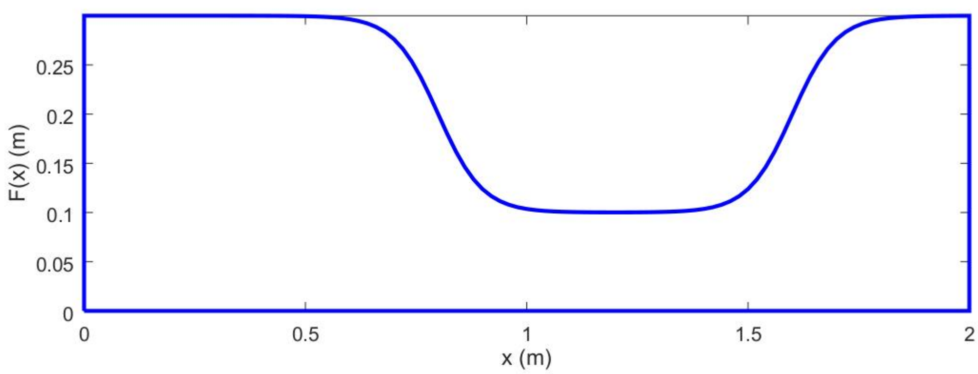

Fig. 2 The profile of the boundary of the nozzle as described by Eq. (1). The abscissa is the *x*-axis. The 3-D nozzle can be built by revolving this profile around the *x*-axis. The nozzle is 2 m long. (Note that the half-width of the nozzle is not exactly 0.3 at $x = 0$ and 2.)