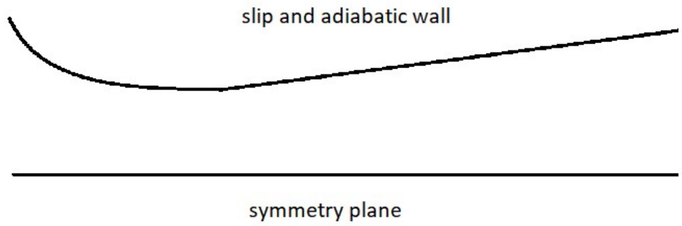
7th assignment - Solution of compressible isentropic flow in a nozzle

The assignment is about the solution of a compressible flow in a convergent-divergent nozzle. A scheme of the geometry and boundary conditions can be seen below.

$$p_o = p\left(1+\frac{\gamma-1}{2}M^2\right)^{\gamma/(\gamma-1)} = 100000\,Pa$$

$$T = 300\,K$$



$$p = p_{exit}$$

$$\frac{\partial \vec{v}}{\partial n}=0 \qquad \frac{\partial T}{\partial n}=0$$

1) Choose a directory to save the case and the case name.

2) Import the mesh. The mesh used in the assignment is in the GAMBIT format and is named "nozzle.neu".

3) After importing the mesh use the "Mesh" menu to specify the "empty" boundary conditions, so that the flow is in the xy plane. You should specify also the "wall" and "patch" and "symmetry" boundary conditions. Notice that a selected boundary acquires a soft red color.

4) In the "Setup" menu you should choose the solver. Select a solver for steady, compressible and turbulent flow. The rhoSimpleFoam solver is going to be used for the simulation.

5) In the "Turbulence" menu select laminar flow.

6) In the "Thermo" menu, under "material database", select the air. Check that air is selected as a perfect (ideal) gas. Change the viscosity value to $\mu = 1.0\times10^{-20}\ N.s/m^2$. The purpose of this value is that we want to make a simulation without irreversibilities.

7) In the "Discretization" menu there are four submenus, for temporal and spatial discretization (Convection), gradients and interpolations calculation. In the temporal discretization option the flow should be automatically selected as steady. In spatial discretization, change the convective momentum scheme from upwind to "limited Linear 1". Check the "vector specific" box.

The "limited Linear α" scheme is a scheme that limits an interpolation of centered differences, so that:

$$\psi(r) = MAX(MIN(2r/\alpha, 1), 0)$$

For α=1 we have a TVD scheme:

$$\psi(r) = MAX(MIN(2r, 1), 0)$$

8) In the solution menu there are four submenus, "Solvers", "Simple", "Residuals" and "Relaxation". Do not change anything in the "Solvers" menu. In "Simple" check the "consistent" and "transonic" boxes. In "residuals" change the value of the target residuals, lowering them all to 10-5. In relaxation, use 0.5 for pressure and 0.1 for the other variables.

9) Ignore the "*Operating Conditions*", "*cell zones*" and"*passive scalars*" menus.

10) In the "Boundary Conditions" menu set the conditions for velocities and pressures at the boundaries that have been categorized as "patch". At the inlets, typically for compressible flows the total pressure and the static temperature are stipulated, and the velocity is calculated using the "pressure directed inlet" boundary condition. At the outlets typically you should specify static pressure, and velocity and temperature have a "zeroGradient" condition. Use a total pressure of 100000 Pa at the inlet, coupled with a inlet temperature (use "fixedValue", ie static temperature; do not use total temperature) of 300K. Set the (1,0,0) direction for the velocity. At the outlet set a static pressure of 60000 Pa.

Check that the wall is a slip wall and is adiabatic $(\partial T/\partial n = 0)$.

11) In the "Initial Conditions" menu, set the initial estimate for pressure, temperature and velocity in the domain. Use a pressure equal to the outlet pressure, and a temperature of 300 K. Set a velocity of 100m/s in the x direction.

12) In "Run", in the "time control" submenu, set a maximum number of iteration equal to 100000. In the "output" option, set how often you want to save the solution. Save your results every100 iterations, saving a maximum of 2 files.

14) Save the case using the floppy disk button and run.

16) After convergence, post-process the results.

17) Go to "postProcessing" and click on the paraView button.

18) In the upper right corner, under "time", choose the last saved iteration.

19) Click on the "apply" button.

20) Click the calculator button and calculate the Mach number in the domain. This is done using the formula $mag(U)/sqrt(1.4*287*T)$, where we have the magnitude of velocity divided by the velocity of sound. Remember that for air $\gamma = 1.4$ and the gas constant is $R = 287$ m^2/(s^2 K).

20) In the "filters" menu, under "alphabetical", choose "plot over line".

21) Choose a line between the points (-5,0,0) and (10,0,0). Plot the Mach numbers and pressure.

22) Repeat the simulations for outlet pressures of 70000 Pa, 80000 Pa and 90000 Pa. Every time you change the outlet pressure, change also the initial domain pressure accordingly. Plot the position of the shock as a function of the outlet pressure and draw your conclusions (the location where the shock occurs should change).