Abstract

The aim of this project is to simulate flow through a convergent-divergent nozzle using OpenFOAM. The governing equation for flow through a convergent-divergent nozzle can be easily derived by assuming quasi-one-dimensional flow The project investigates results such as shock location, pressure distribution etc. along the length of the nozzle, obtained from the simulation, with the analytical results.

Problem Statement

This case involves steady, inviscid, non-heat-conducting flow through a convergent-divergent nozzle. The nozzle cross-section varies as

$$A(x) = \begin{cases} 1.75 - 0.75\cos(0.2x - 1)\pi, & 0 < x \leq 5 \\ 1.25 - 0.25\cos(0.2x - 1)\pi, & 5 < x \leq 10 \end{cases}$$

The nozzle geometry is shown in fig. 1.

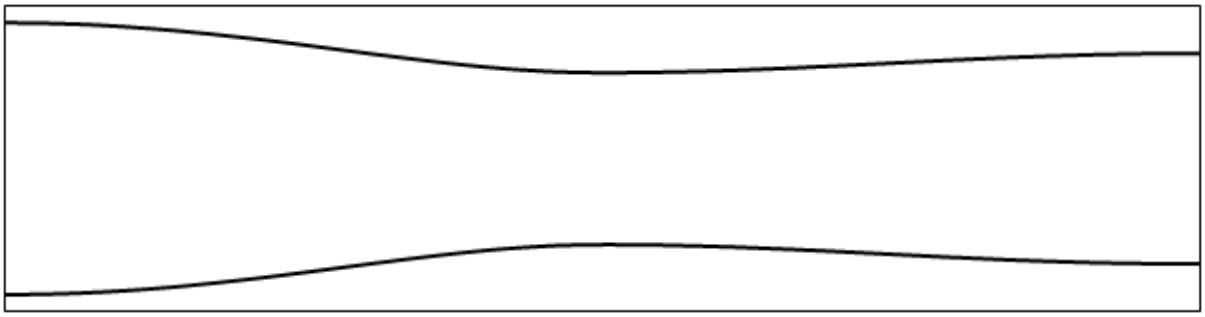

Figure 1. The configuration of flow through a convergent-divergent nozzle.

The nature of the flow is determined by the exit static pressure. Three values of exit static pressure are examined which result in three types of flows:

1. subsonic, isentropic flow ($p_{exit}/p_{in} = 0.89$)
2. supersonic flow with a normal shock in the diffusing section ($p_{exit}/p_{in} = 0.75$)
3. supersonic, isentropic flow (p_{exit}/p_{in} to be determined)