Abstract

The aim of this project is to simulate pressure driven flow through a nozzle connected to a reservoir using OpenFOAM. The governing equation for flow through a nozzle can be easily derived by assuming quasi-one-dimensional flow. The nozzle geometry is fixed and the reservoir conditions are also given. Calculations need to be done to determine the outlet boundary conditions for observing a shock in the nozzle. The project investigates results such as shock location, pressure distribution, variation of Mach number etc. along the length of the nozzle, obtained from the simulation, and compare the same with the analytical results.

Problem Statement

This case involves steady, inviscid, non-heat-conducting flow through a nozzle connected to a reservoir.

The geometry used is shown in fig. 1.

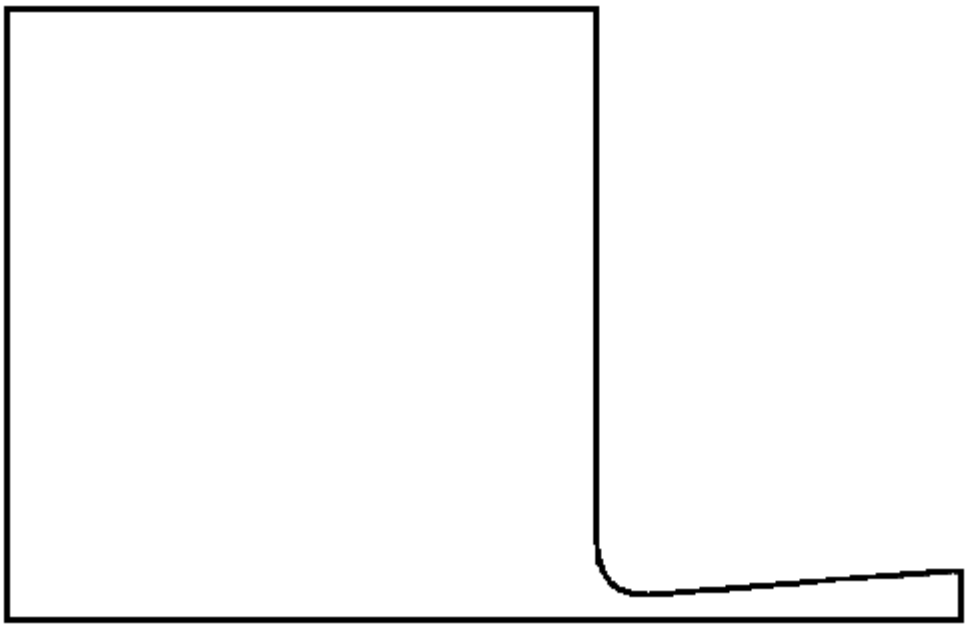

Figure 1. The configuration of pressure flow through a nozzle connected to a reservoir.

The nature of the flow is determined by the exit static pressure. The inlet conditions (reservoir) are fixed. For the given nozzle geometry, the exit boundary conditions are calculated such that a shock is generated in the nozzle.