

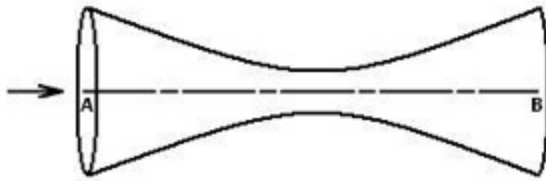
FLUENT - Compressible Flow in a Nozzle- Problem Specification

Author: Rajesh Bhaskaran, Cornell University

Problem Specification

1. [Pre-Analysis & Start-up](#)
 2. [Geometry](#)
 3. [Mesh](#)
 4. [Setup \(Physics\)](#)
 5. [Solution](#)
 6. [Results](#)
 7. [Verification & Validation](#)
- [Problem 1](#)
[Problem 2](#)

Problem Specification



Consider air flowing at high-speed through a convergent-divergent nozzle having a circular cross-sectional area, A , that varies with axial distance from the throat, x , according to the formula

$$A = 0.1 + x^2; -0.5 < x < 0.5$$

where A is in square meters and x is in meters. The stagnation pressure p_o at the inlet is 101,325 Pa. The stagnation temperature T_o at the inlet is 300 K. The static pressure p at the exit is 3,738.9 Pa. We will calculate the Mach number, pressure and temperature distribution in the nozzle using FLUENT and compare the solution to quasi-1D nozzle flow results. The Reynolds number for this high-speed flow is large. So we expect viscous effects to be confined to a small region close to the wall. So it is reasonable to model the flow as inviscid.

Go to [Step 1: Pre-Analysis & Start-up](#)

[See and rate the complete Learning Module](#)

[Go to all FLUENT Learning Modules](#)