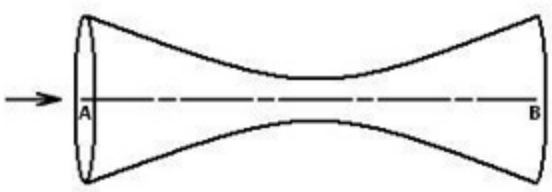
FLUENT - Compressible Flow in a Nozzle- Problem Specification

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