Compressible Flow in a Nozzle

Created using ANSYS 13.0. Updated to work with later versions.

This tutorial has videos. If you are in a computer lab, make sure to have head phones.

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

\[
A = 0.1 + x^2; \quad -0.5 < x < 0.5
\]

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

Go to all FLUENT Learning Modules