FLUENT - Compressible Flow in a Nozzle

Author: Benjamin Mullen, Cornell University

Problem Specification
1. Pre-Analysis & Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Numerical Solution
6. Numerical Results
7. Verification & Validation
Exercises
Comments

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

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

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