ANSYS AIM - Flow in a S-Duct

Author(s): Sebastian Vecchi, ANSYS Inc.

Problem Specification
1. Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Results
6. Verification & Validation

Flow in a S-Duct

Created using ANSYS 18.1

Problem Specification

S-ducts, or serpentine inlets, are commonly found in jet engines due to their ability to allow shorter fins and rudders closer to the engine in the longitudinal direction. S-ducts are found in many other applications, from basic HVAC to complex F1 racing.

In this tutorial, the velocity vectors and pressure profile will be simulated in an S-duct. The flow volume has been provided. Flow will enter the large volume upstream of the S-duct and exit from the end protruding from the large volume. The pressure at the inlet will be 88744 [Pa] and the outlet will be supersonic.

Go to Step 1: Start-Up

Go to all ANSYS AIM Learning Modules