ANSYS AIM - Flow through U-Duct

Author(s): Sebastian Vecchi, ANSYS Inc.

Flow through U-Duct

Created using ANSYS 18.1

Problem Specification

In this tutorial, the velocity vectors, pressure profile and streamlines will be simulated in a 180 degree U-bend square duct. The material passing through the duct will be air with an intake velocity will be 12 m/s which is comparable to the speed of air in an office HVAC system. The geometry which we will be using is shown below, the inlet will be at the end of the longer section at the top while the outlet is the at the end of the shorter duct on the bottom.

Find the velocity vectors and pressure contour of the entire duct.

Go to Step 1: Start-Up

Go to all ANSYS AIM Learning Modules