ANSYS AIM - Fluid Flow over a Bluff Body

Author(s): Sebastian Vecchi, ANSYS Inc.

Fluid Flow over a Bluff Body

Created using ANSYS 18.1

Problem Specification

Simulate fluid flow over a triangular bluff body under isothermal flow conditions. The following image specifies the problem geometry, while the inlet velocity is 17.3 m/s.

Go to Step 1: Start-Up

Go to all ANSYS AIM Learning Modules