

ANSYS AIM - Fluid Flow over a Bluff Body

Author(s): Sebastian Vecchi, ANSYS Inc.

Problem Specification

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

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](#)