

ANSYS Fluid Flow over a Bluff Body - Geometry

Author(s): Sebastian Vecchi, ANSYS Inc.

[Problem Specification](#)

[1. Start-Up](#)

[2. Geometry](#)

[3. Mesh](#)

[4. Physics Setup](#)

[5. Solution/Results](#)

[6. Verification & Validation](#)

Geometry

Import Geometry

Select and open the appropriate file named **FlameHolderSV**. This file can be downloaded [here](#).

Once successfully imported, [press the blue **Finish** button](#).

[Go to Step 3: Mesh](#)

[Go to all ANSYS AIM Learning Modules](#)