

ANSYS Fluid Flow over a Bluff Body - Startup

Author(s): Sebastian Vecchi, ANSYS Inc.

Problem Specification

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

Start Up

A few words on the formatting on the following instructions:

1. Notes that require you to perform an action are colored in blue
2. General information is colored in black, but does not require any action
3. Words that are **bolded** are labels for items found in ANSYS AIM
4. Most important notes are colored in red

We are now ready to begin simulating in ANSYS AIM. Open ANSYS AIM by going to [Start > All Apps > ANSYS 18.1 > ANSYS AIM 18.1](#). Once you are at the starting page of AIM select the **Fluid Flow** template as shown below.

You will be prompted by the **Fluid Flow** template to either **Define new geometry**, **Import geometry file**, or **Connect to active CAD session**. [Select Import geometry file](#) and press **Next**.

[Go to Step 2: Geometry](#)

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