## **ANSYS Fluid Flow over a Bluff Body - Results**

Author(s): Sebastian Vecchi, ANSYS Inc.

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

## Solution/Results

Press the **Results** button in the **Workflow** to extract information from the simulation. In order to find information that can be readily used, first press **Evalua te Results**. Once the evaluation is complete, AIM will automatically output a vector in the **Results** section under **Objects**. Most of the time, the default setting for these vector plots use a vector appearance that is too big, making it difficult to analyze. A good technique is to make a plane that bisects the fluid flow. Add a plane by selecting the two symmetry walls and pressing the **Add plane** button near the top right corner of the window.



Select the Velocity Vector to edit the settings with which the vectors are defined. Select the new plane as the Location. Retain Symbol distribution as U niformly distributed and input a value between 1000-2000 for Approximate number of points. If desired, change the Symbol sizing in the Appearance section to alter the arrows. Press the Play button in the model window to see how these velocity vectors develop over time.



## Go to Step 6: Verification & Validation

Go to all ANSYS AIM Learning Modules