

ANSYS Fluid Flow over a Bluff Body - Results

Author(s): Sebastian Vecchi, ANSYS Inc.

Problem Specification

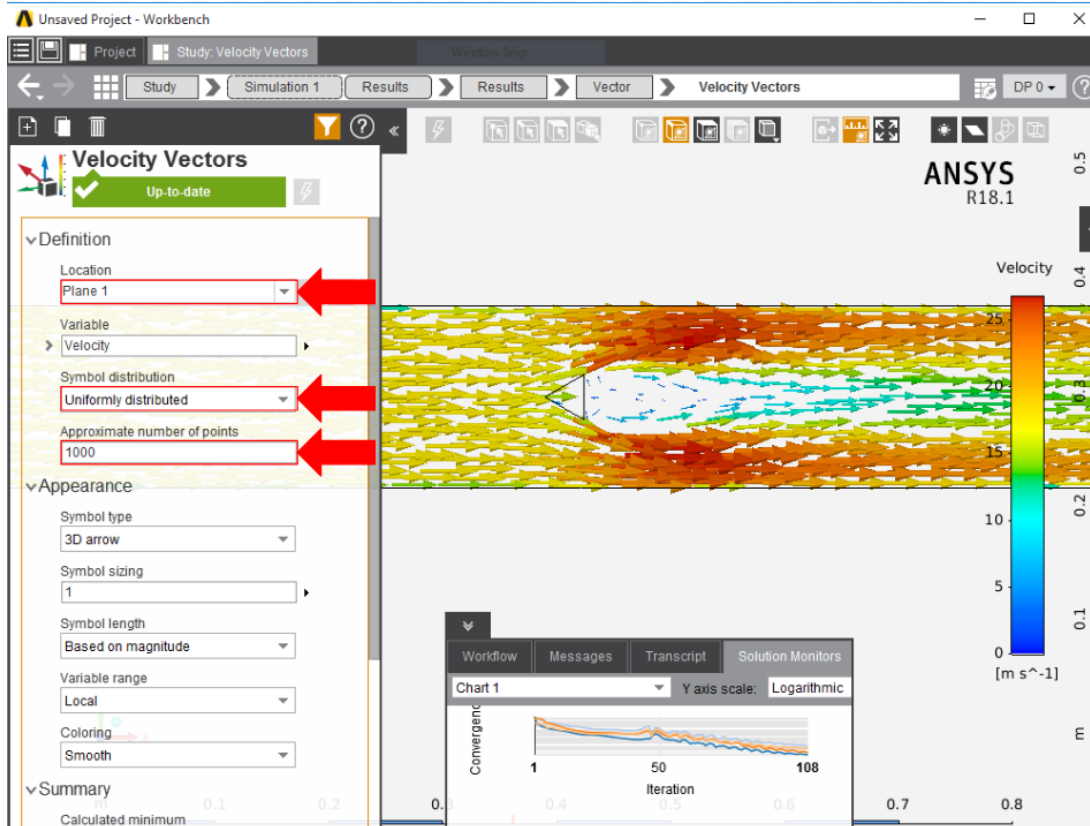
1. Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Solution/Results
6. Verification & Validation

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 Evaluate 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 **Uniformly 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](#)