ANSYS Fluid Flow over a Bluff Body - Results

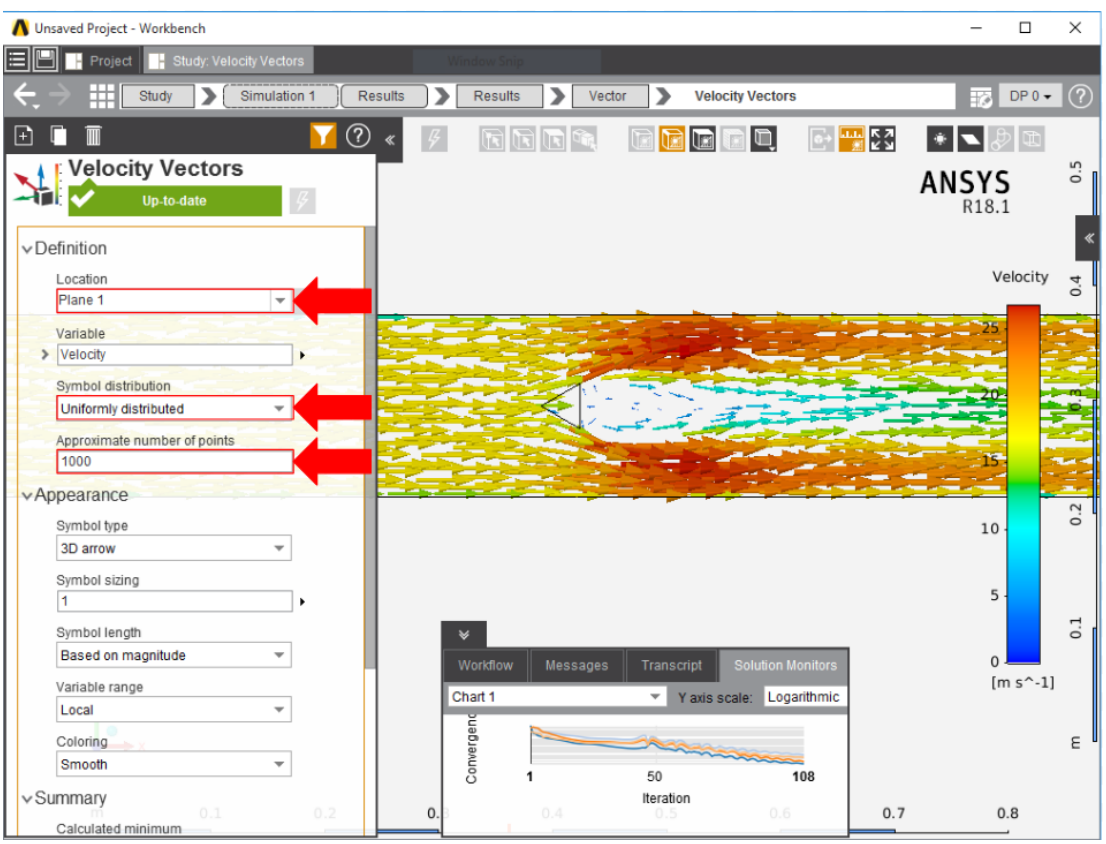
Author(s): Sebastian Vecchi, ANSYS Inc.

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