ANSYS Transonic Flow over a Wing - Results

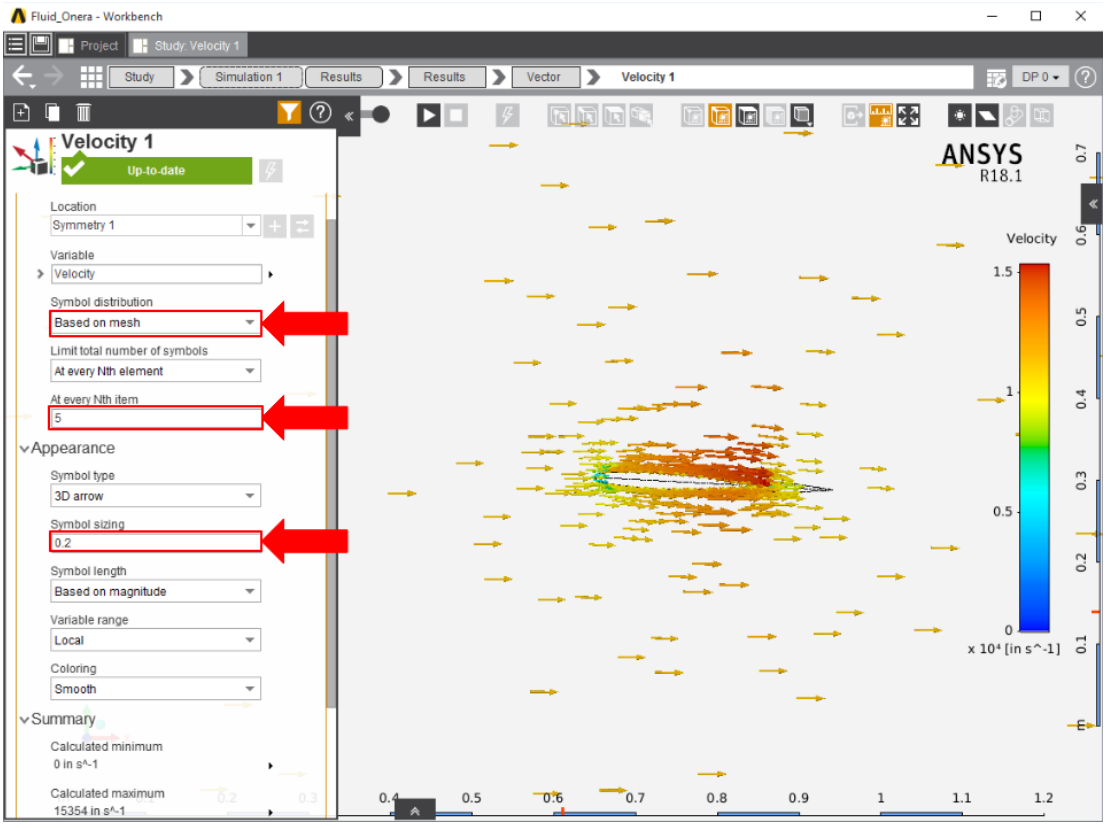
Author(s): Sebastian Vecchi, ANSYS Inc.

Problem Specification
1. Startup
2. Geometry
3. Mesh
4. Physics Setup
5. Solution/Results
6. Verification & Validation

Solution/Results

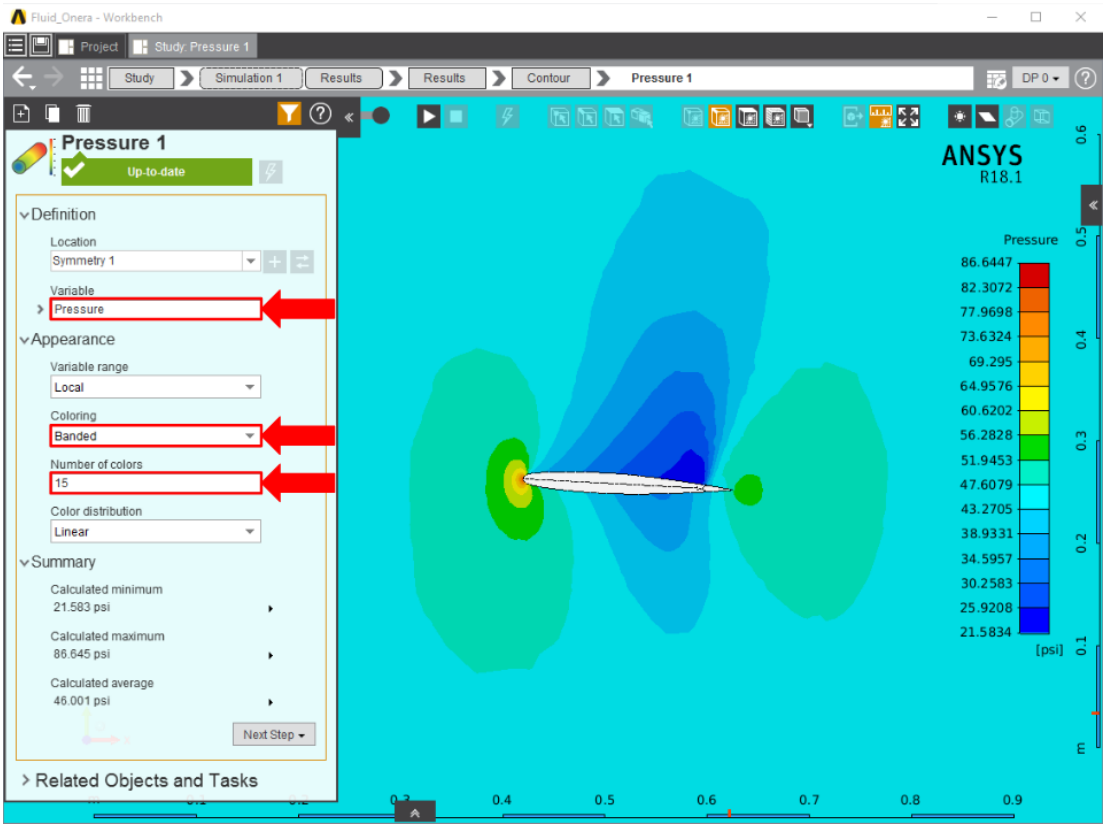
Press the **Results** button in the **Workflow** to extract information from the simulation. In order to find information that can be readily used, first press **Evaluate Results**. Once the evaluation is complete, AIM will automatically output a vector in the **Results** section under **Objects**. Due to the size of the flow volume, the evaluation may take some time to calculate.

Select the **Velocity Vector** to edit the settings with which the vectors are defined. Change the **Location** to **Symmetry 1**, the **Symbol distribution** to **Based on mesh**, and input 5 for **At every Nth item**. Change the **Symbol sizing** to 0.2 in the **Appearance** section to alter how big the arrows are. Press the **Play** button in the model window to see how these velocity vectors develop over time.



Pressure on the wing can be plotted by adding a **Contour** in the **Add** drop down menu. Use **Pressure** as the **Variable** and then select the faces of the wing.

A pressure contour on the symmetry plane can also be plotted by adding a **Contour** in the **Add** drop down menu. Use **Pressure** as the **Variable** and then select the symmetry face. Change the **Coloring** to **Banded** and update the **Number of colors** to 15.



Go to Step 6: Validation & Verification

Go to all ANSYS AIM Learning Modules