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

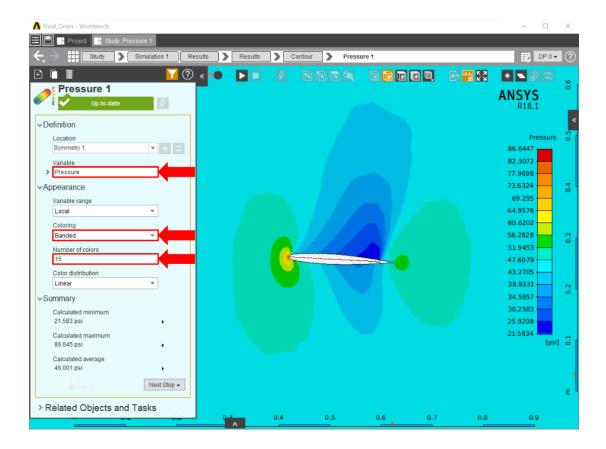
Press the **Results** button in the **Workflow** to extract information from the simulation. In order to find information that can be readily used, first press **Evalua te Results**. Once the evaluation is complete, AIM will automatically output a vector in the **Results** section under **Objects**. 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 Base d 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