

Compressible Flow Over an Airfoil - Results

Author(s): Sebastian Vecchi, ANSYS Inc.

Problem Specification

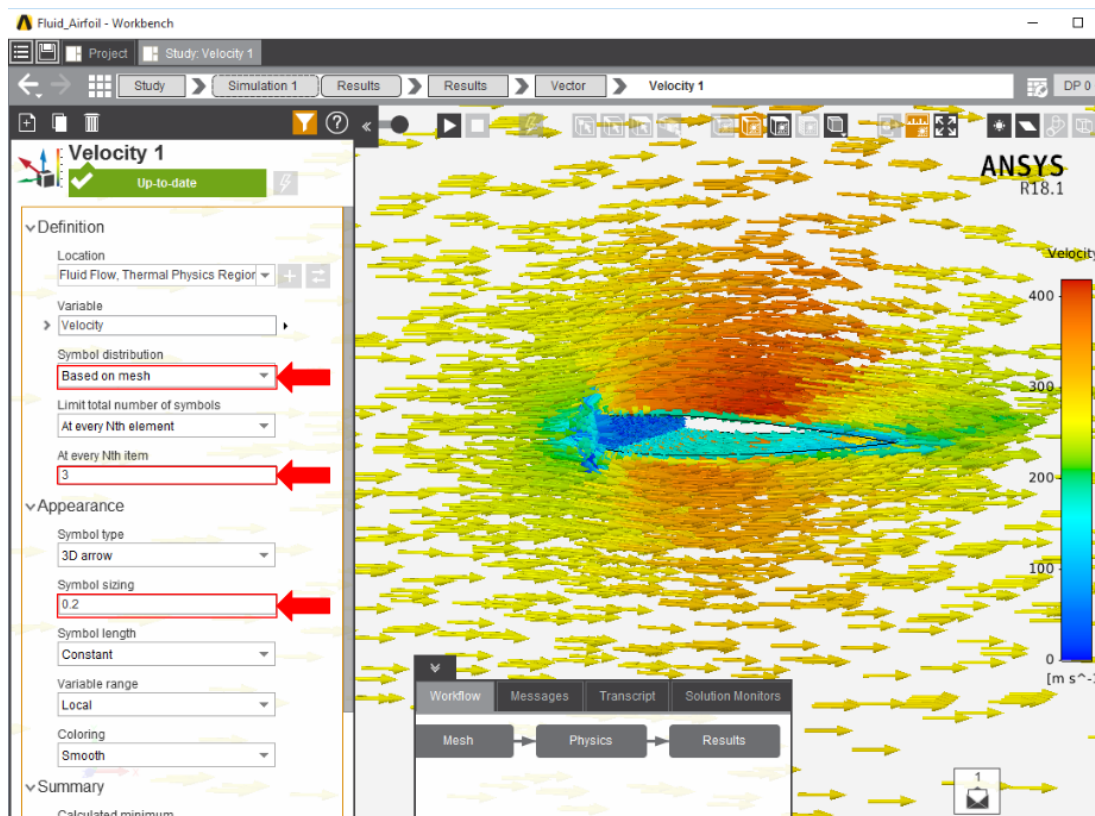
1. Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Results
6. Verification & Validation

Exercise

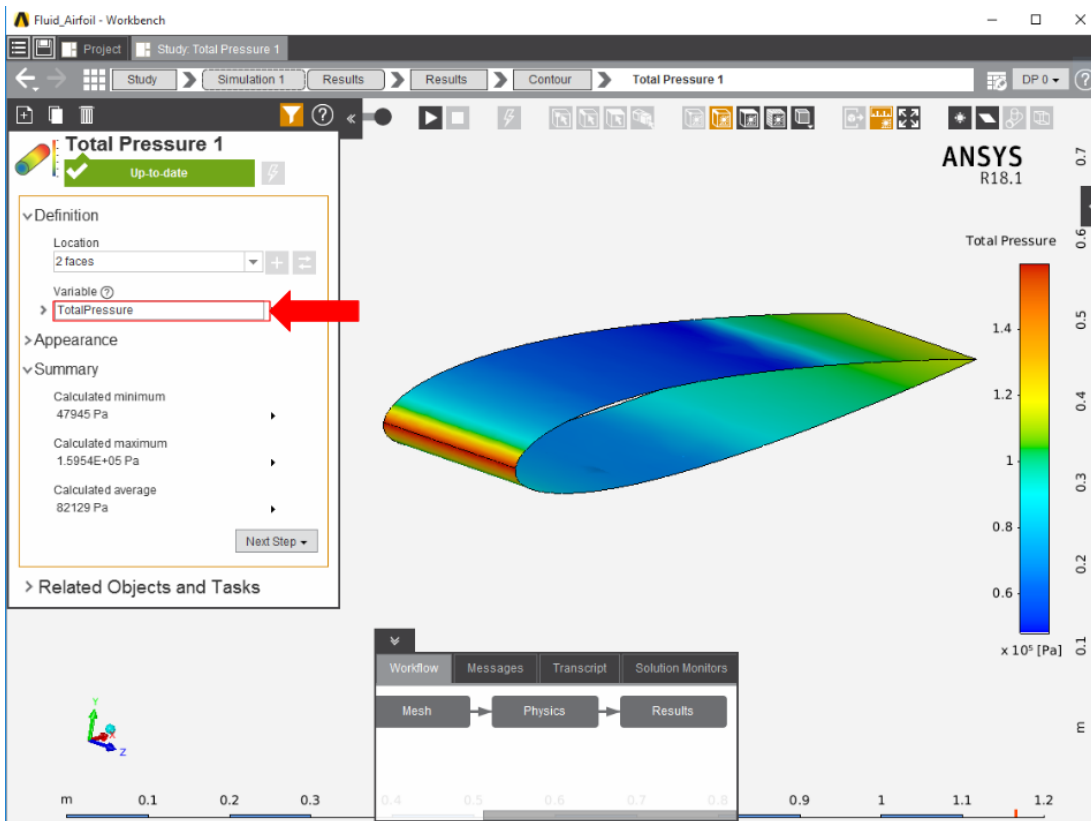
Results

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**. AIM will run the calculations and evaluate the results. Once the evaluation is complete, AIM will automatically output a velocity **Vector** in the **Results** section under **Objects**.

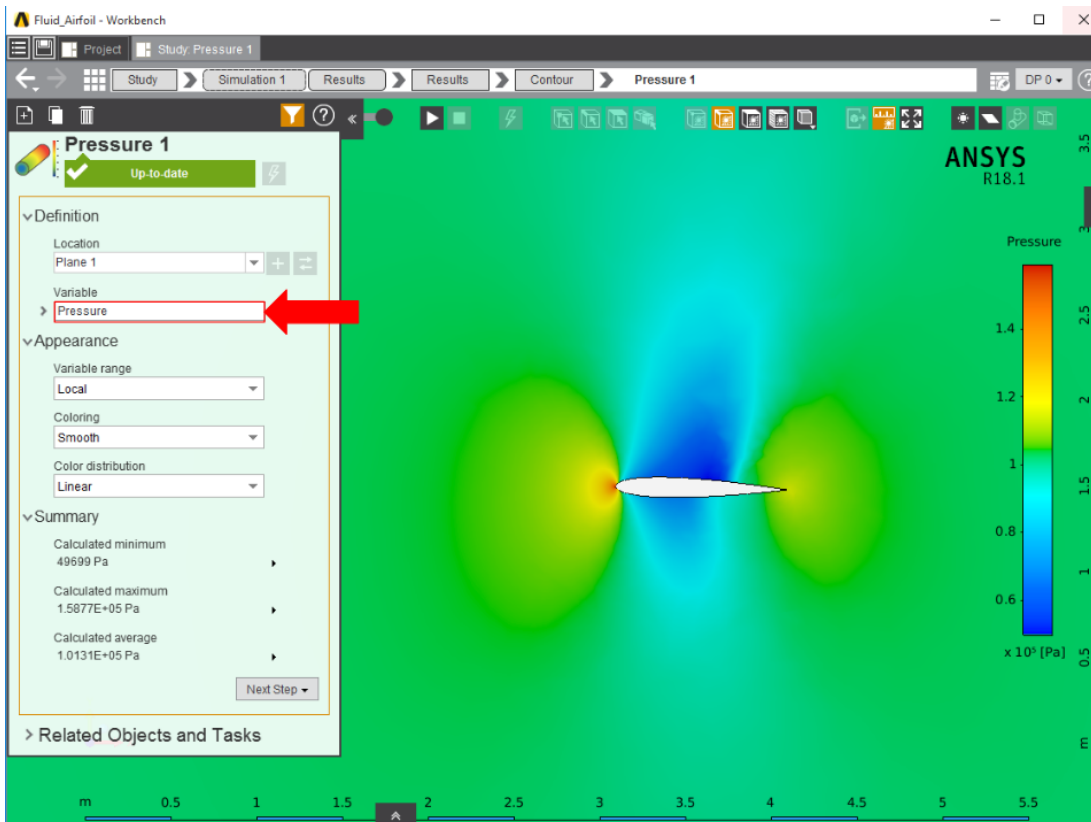
By default, the velocity vector arrows are too far from the airfoil to observe the flow characteristics. **Select the Velocity Vector** object to edit the settings with which the vectors are defined. **Change Symbol distribution to Based on mesh** and input 3 for **At every Nth item**. Press **Evaluate** to update the display. Under **Appearance**, change the **Symbol sizing** to 0.2 to shorten the arrows, then zoom in to see the arrows near the airfoil. Press the **Play** button in the model window to see how these velocity vectors develop over time.



Pressure on the airfoil can be plotted by adding a **Contour** in the **Add** drop down menu of the **Results** panel. Use **Total Pressure** as the **Variable** and then select the faces of the flow volume touching the faces of the airfoil.



To plot the pressure change, a contour on the side of the flow volume will accurately represent a 2D pressure contour of the flow. Select **Contour** in the **Ad** drop down menu, select one of the large faces, and assign the **Variable** to be **Pressure**.



[Go to Step 6: Verification & Validation](#)

[Go to all ANSYS AIM Learning Modules](#)