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

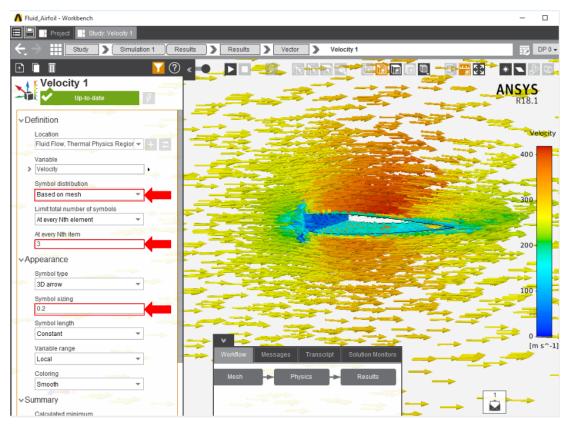
6. Verification & Validation

Exercise

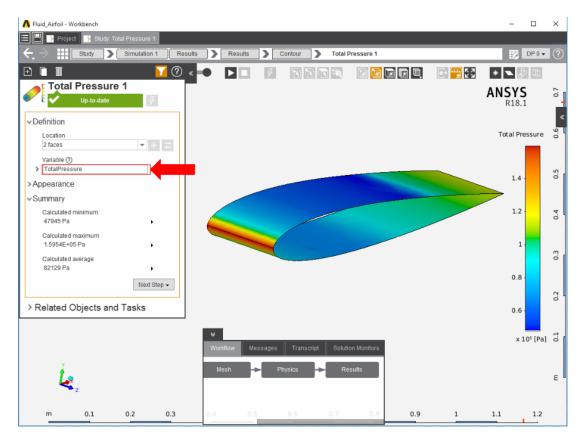
Results

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

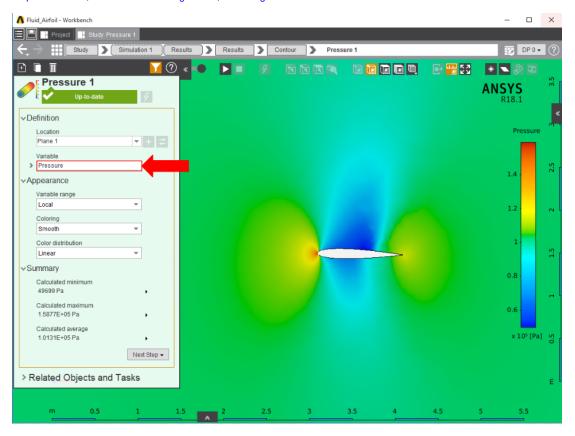
By default, the velocity vector arrows are too far from the airfoil to observe the flow characteristics. Select the **Velocity Vector** object to edit the settings with which the vectors are defined. Change **Symbol distribution** to **Based on mesh** and input 3 for **At every Nth item**. Press **Evaluate** 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**c 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