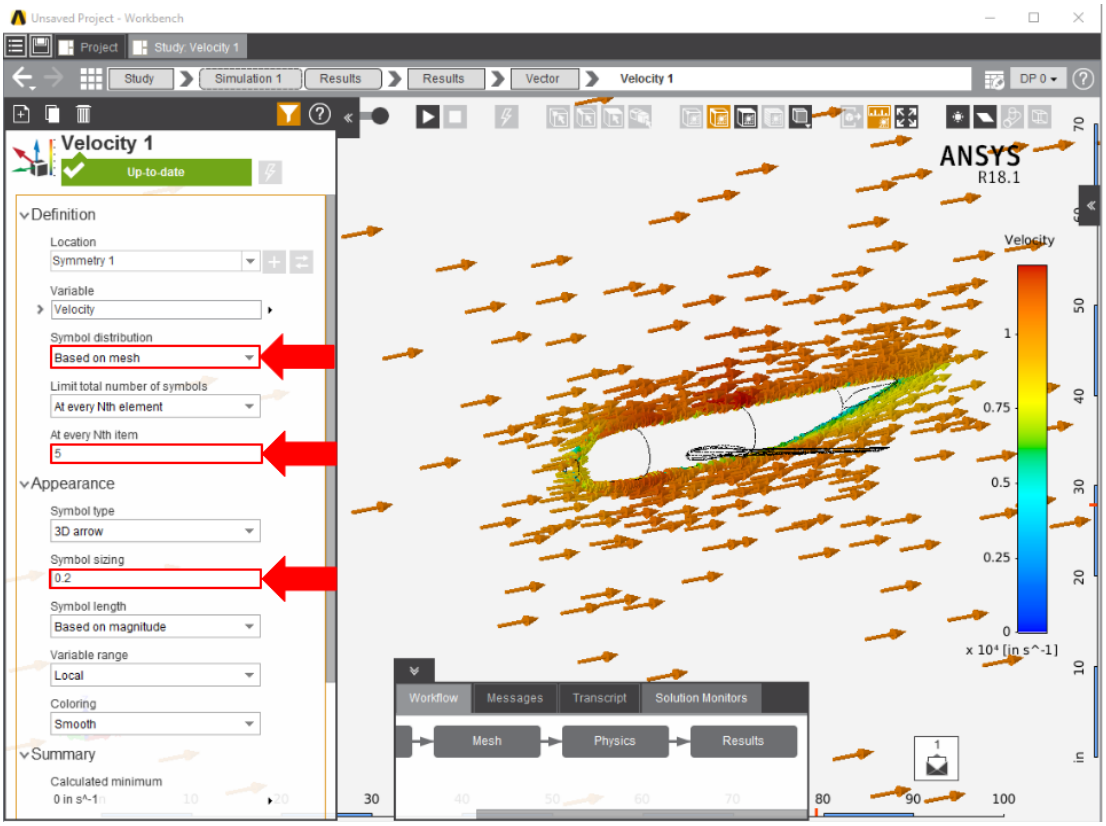
ANSYS Compressible Flow over a Wing-Body Junction - Solution/Results

Author(s): Sebastian Vecchi, ANSYS Inc.

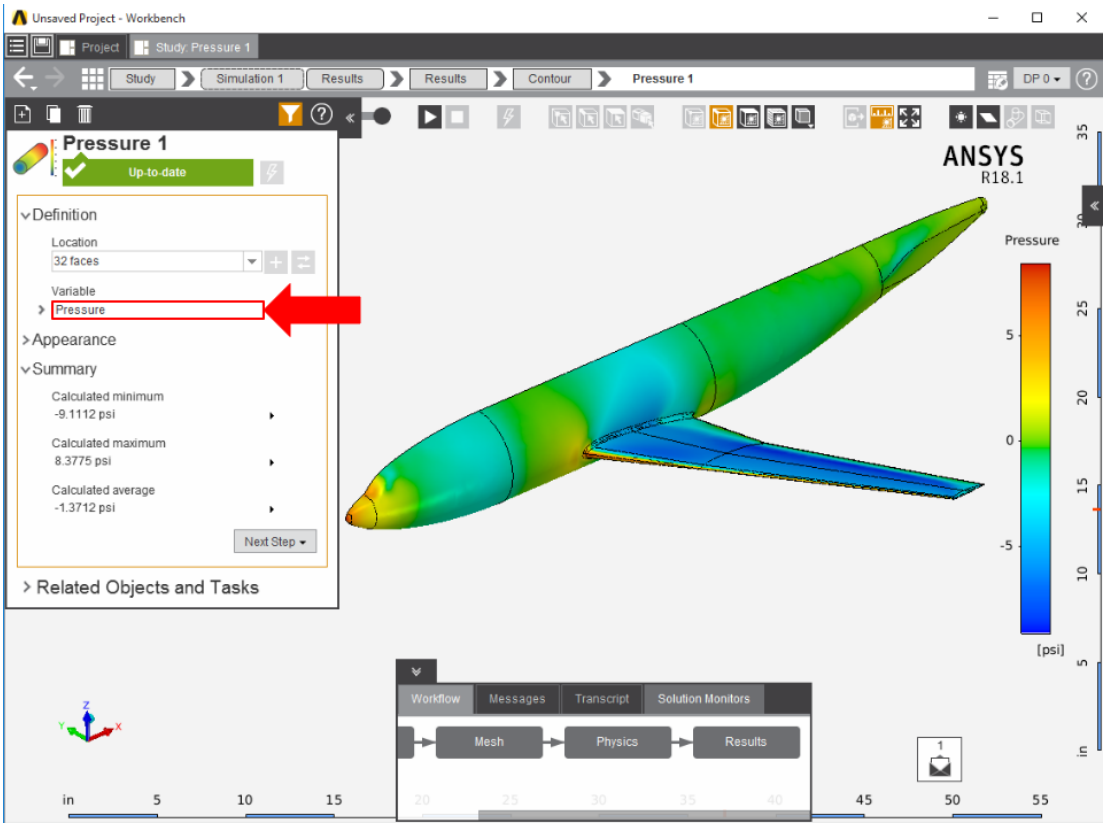
Solution/Results

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**.

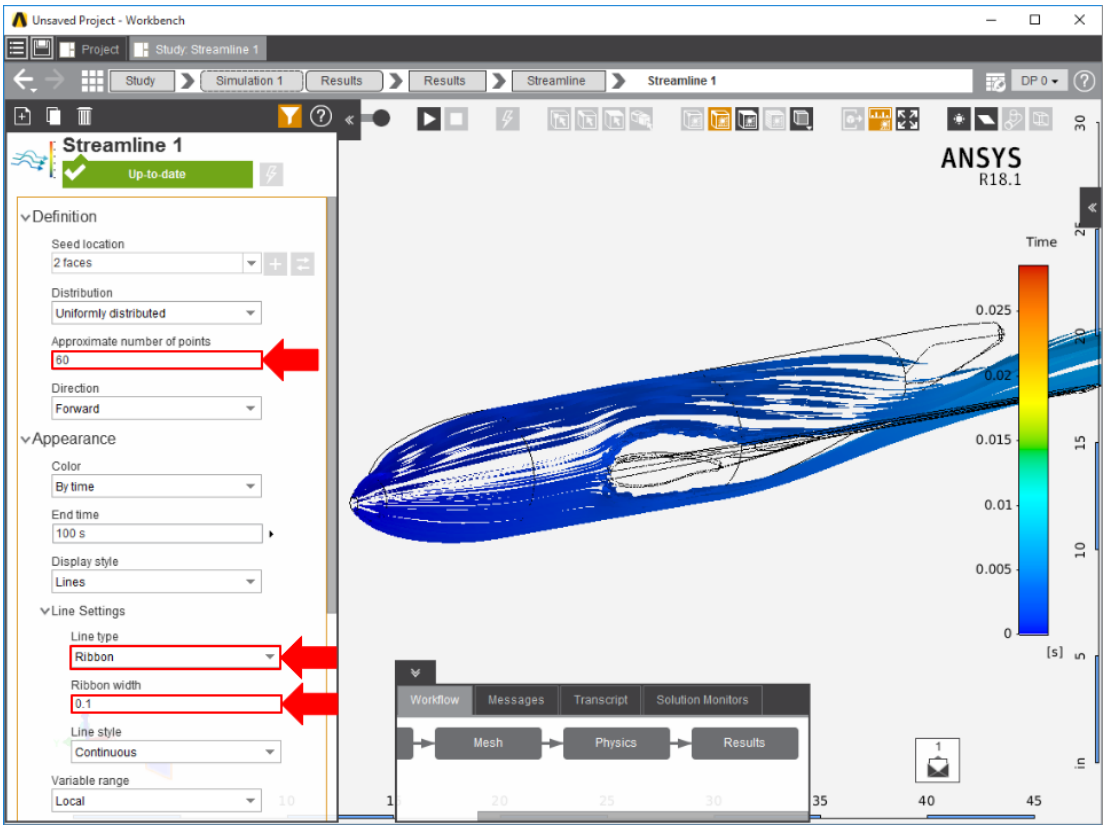
Select the **Velocity Vector** to edit the settings with which the vectors are defined. Update the **Location** to **Symmetry 1**, change **Symbol distribution** to **Based on mesh** and input 5 for **At every Nth element**. Change the **Symbol sizing** in the **Appearance** section to alter how big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



Pressure on the body can be plotted by adding a **Contour** in the **Add** dropdown menu. Use **Pressure** as the **Variable** and then select the faces of the airplane body.



Streamlines can be computed by picking the **Streamline** option in the **Add** drop down menu near the **Results** category. Select the 2 faces at the nose of the airplane as the **Seed location**. Then, input 60 for **Approximate number of points**. Under **Line settings** in the **Appearance** section, change the **Line type** to **Ribbon** and the **Ribbon width** to 0.1. Press the **Play** button in the model window to see how these streamlines develop over time. Using **Ribbon** as the **Line type** allows us to see how the flow twists.



A pressure contour can be applied to the streamline to show the pressure change as the flow goes over the airplane. Right click the empty model area and select **Add** > **Results** > **Contour**. The **Location** should automatically be updated to **Streamline 1**. Change the **Variable** to **Pressure** and press **Evaluate**.

Go to all ANSYS AIM Learning Modules