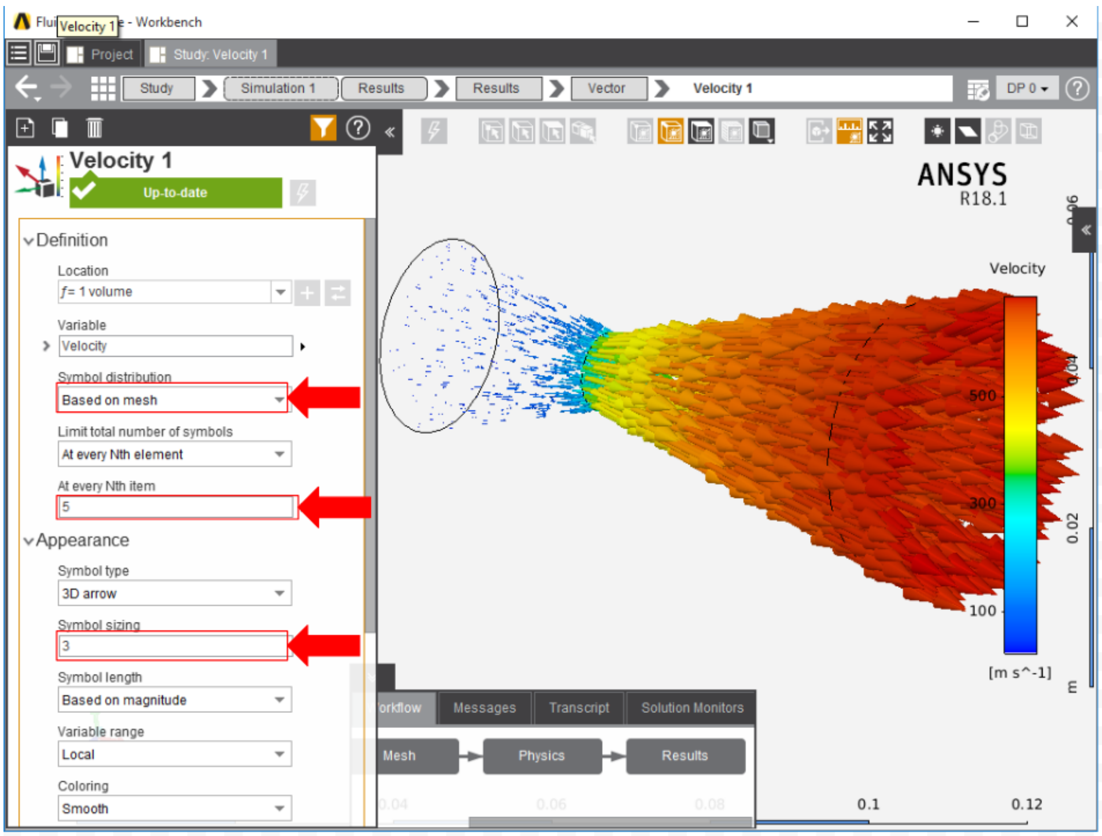
Compressible Flow in a Nozzle - Solution

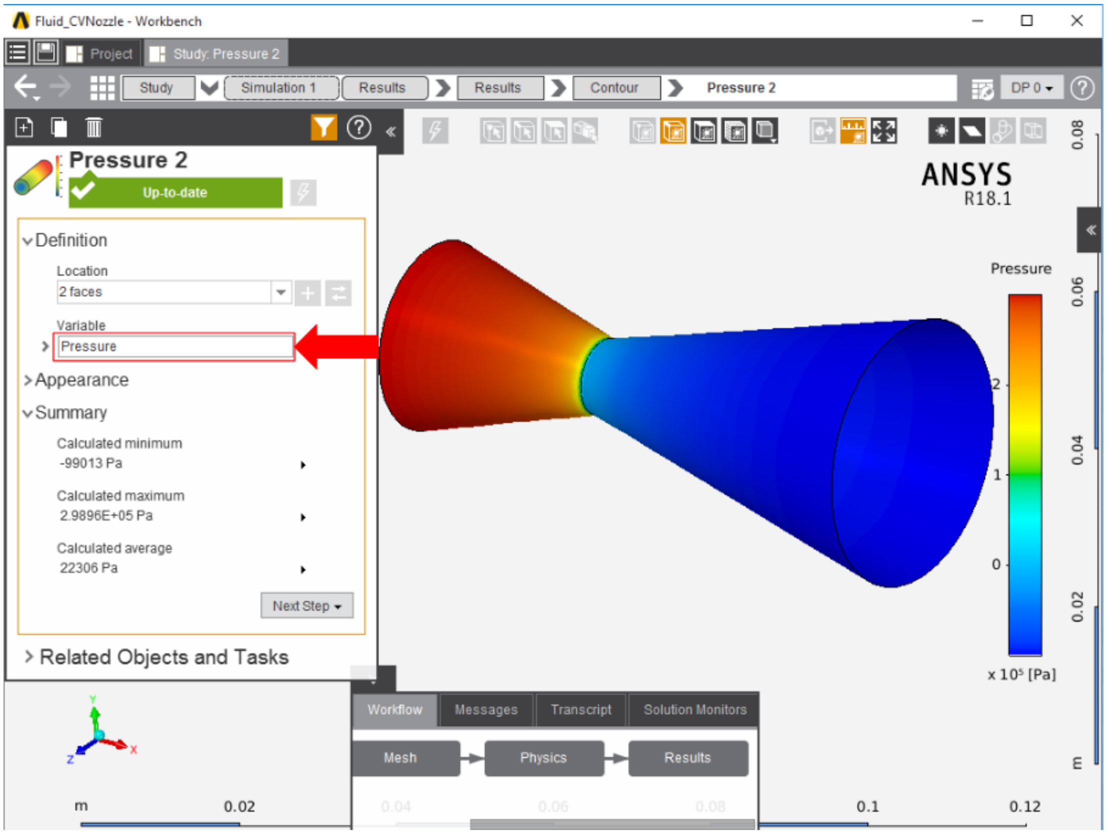
Author(s): Sebastian Vecchi, ANSYS Inc.

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**. The vectors show air velocity, but may be hard to see. Select the **Velocity vector** to edit the settings with which the vectors are defined. Change the **Symbol distribution** to **Based on mesh** and **At every Nth item** to 5. Under **Appearance**, change **Symbol sizing** to 3. Press **Evaluate** to update the vectors. Press the **Play** button in the model window to see how these velocity vectors develop over time.



To find the total pressure on the walls of the nozzle, select **Contour** from the **Results Add** dropdown menu. Select all of the outside faces of the flow volume (except the inlet and outlet) as the **Location** and change the **Variable** to **Pressure**.



The Mach number distribution inside the flow field can be displayed by adding a plane that evenly cuts through the nozzle. In the top right corner of the model window, click the **Add plane** button. By default, the plane is oriented to bisect the nozzle. Right click on the plane and select **Add** > **Results** > **Contour**, then change the **Variable** to **Mach Number**. Press **Evaluate** to see the contour.

Go to Step 6: Verification

Go to all ANSYS AIM Learning Modules