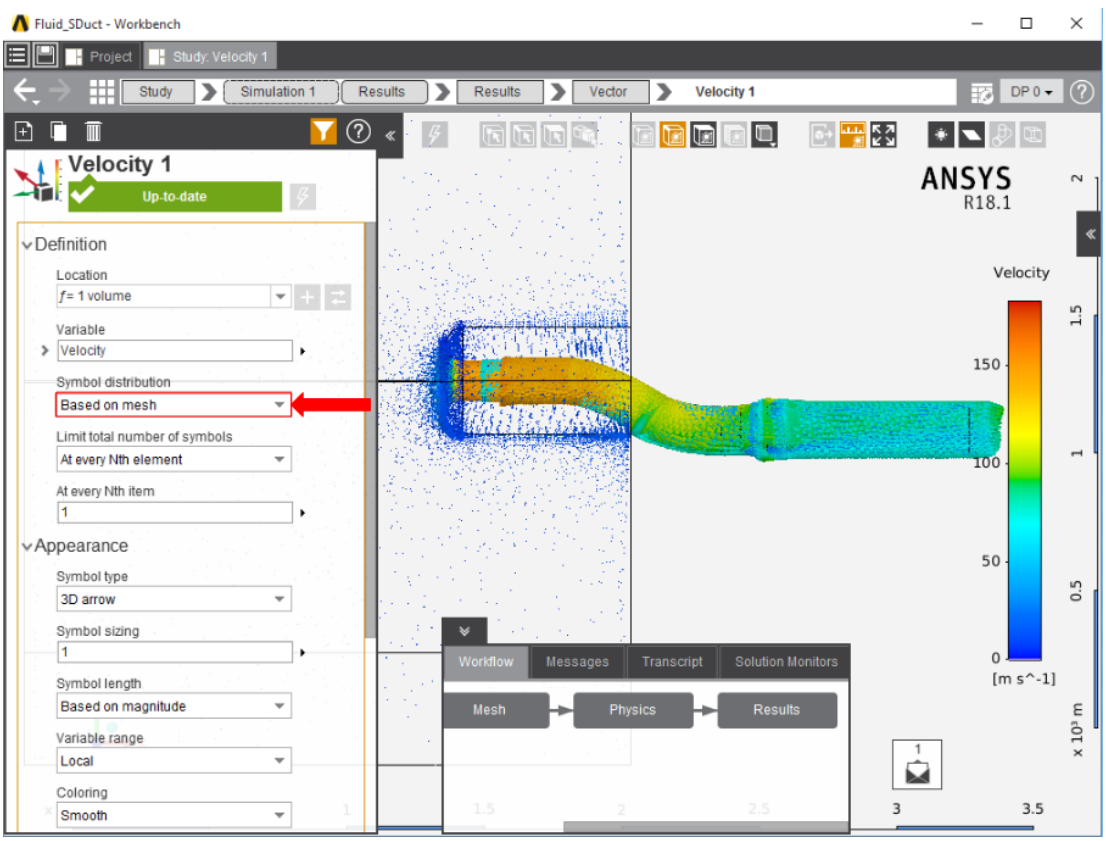
ANSYS Flow in a S-Duct - Results

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. The vectors will show air velocity, but may be difficult to see.

Select the **Velocity Vector** to edit the settings with which the vectors are defined. Change the **Symbol distribution** to **Based on mesh**, then press **Evaluate**. Press the **Play** button in the model window to see how these velocity vectors develop over time.



To create a contour of the velocity inside the fluid volume, create a **Plane** to bisect the flow volume. The initial orientation of the plane should be as intended. Once the plane has been created, use the **Add** drop down menu in the **Results** panel to create a **Contour** whose **Location** is the plane and **Variable** is **Velocity Magnitude**.

Go to Step 6: Validation

Go to all ANSYS AIM Learning Modules