

ANSYS Flow in a S-Duct - Results

Author(s): Sebastian Vecchi, ANSYS Inc.

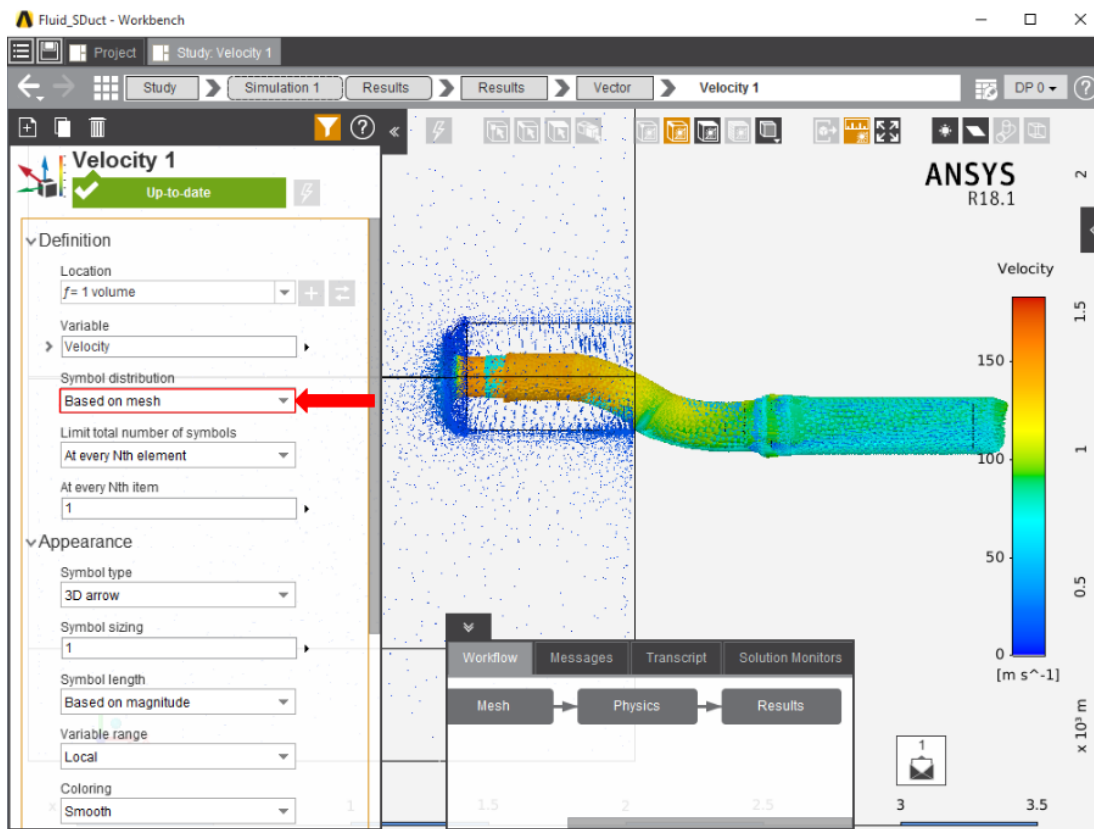
Problem Specification

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

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](#)