

# Compressible Flow in a Nozzle - Solution

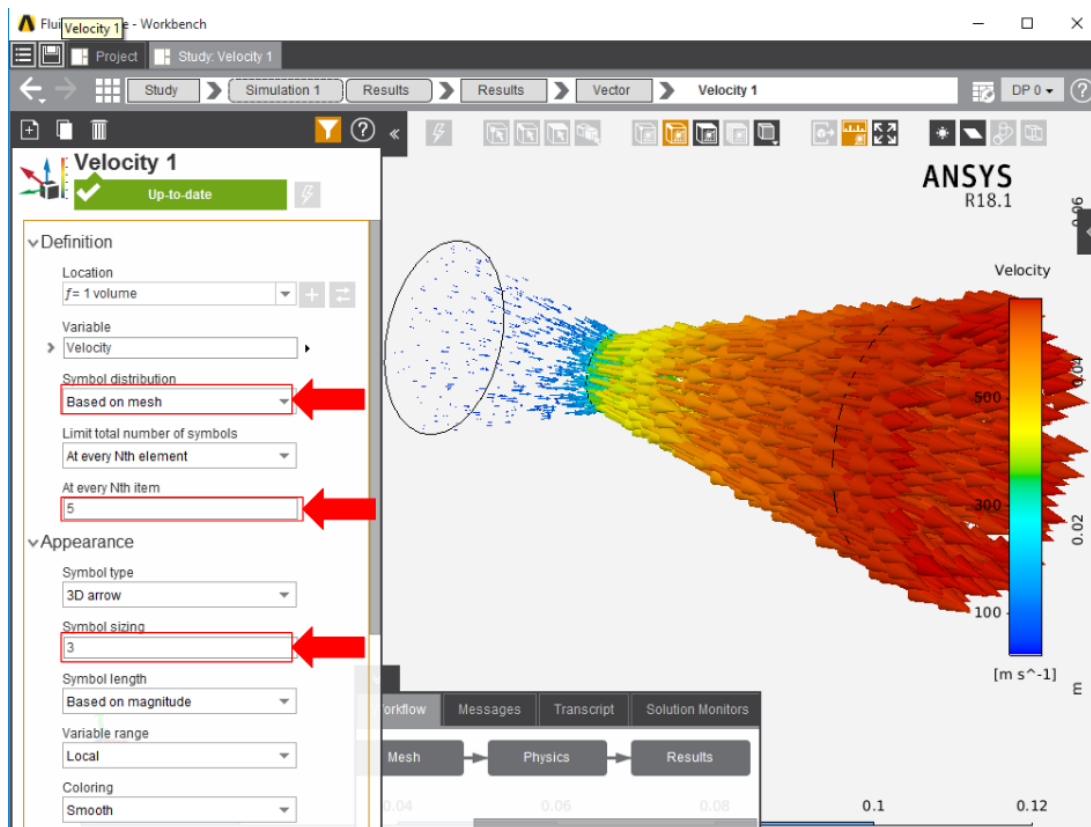
Author(s): Sebastian Vecchi, ANSYS Inc.

## Problem Specification

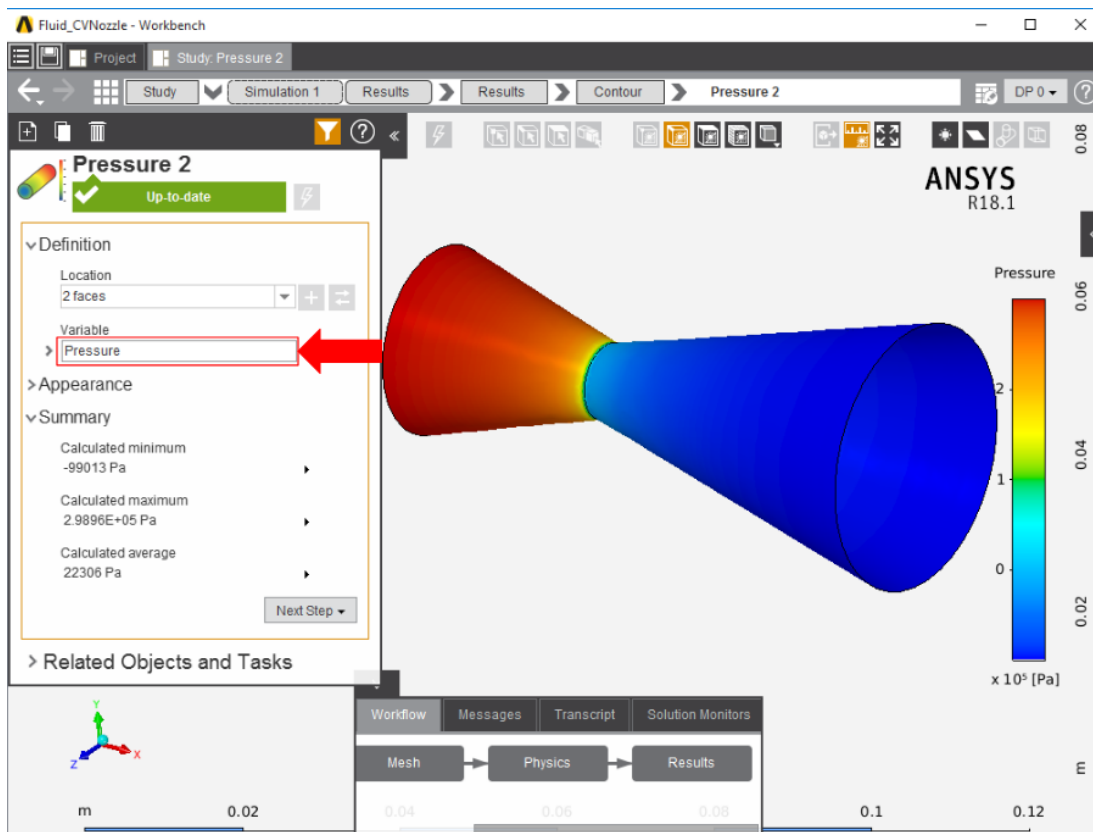
1. Pre-Analysis & Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Results
6. Verification & Validation

## 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](#)