

# ANSYS Taylor-Couette Flow between Rotating Cylinders - Results

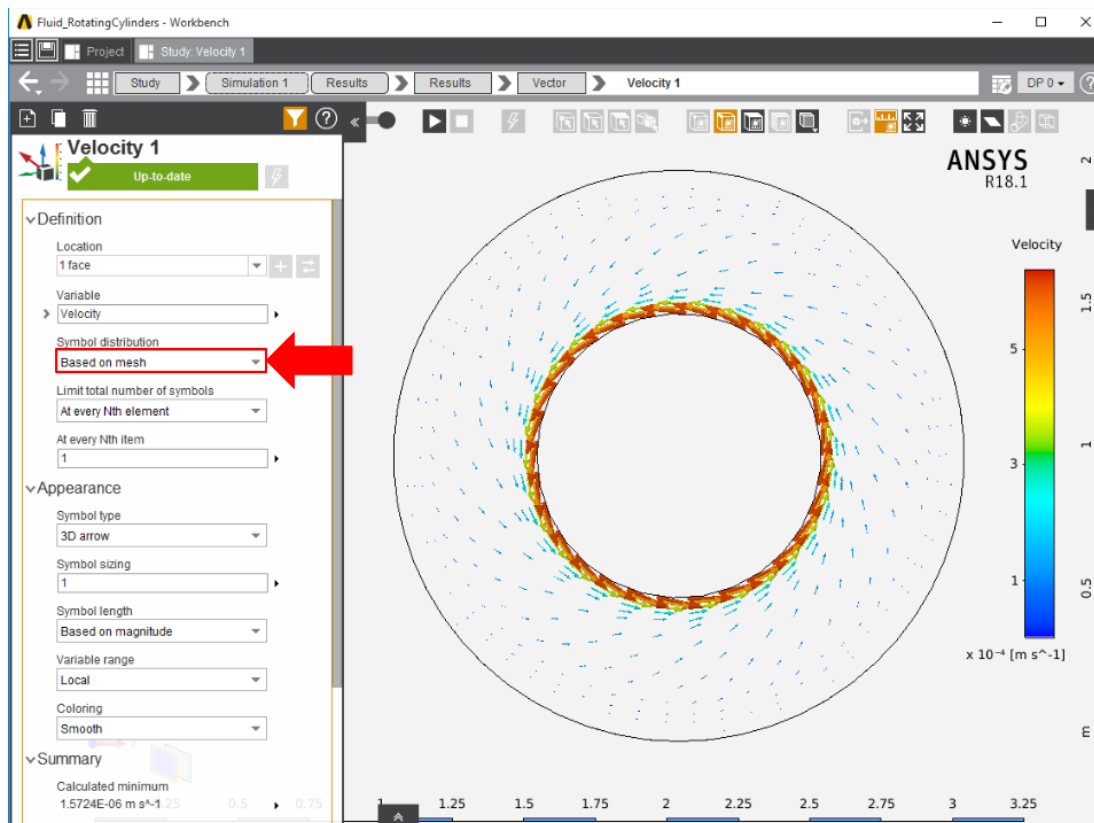
Author(s): Sebastian Vecchi, ANSYS Inc

## Problem Specification

1. Pre-Analysis & Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Solution/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 Velocity Vector in the Results section under Objects. **Select the Velocity Vector** to edit the settings with which the vectors are defined and update the **Location** to the top face of the flow. **Change Symbol distribution to Based on mesh** and then press **Evaluate**. Press the **Play** button in the model window to see how these velocity vectors develop over time.



The pressure as a function of distance and radius can be plotted by adding a contour to the top and bottom faces of the flow volume. **Return to the Results panel**, then **select the top and bottom faces of the flow volume** and **click on Contour** from the **Add** drop down menu. A **Contour** panel will appear with the location specified. **Change the Variable to Pressure** and **press Evaluate**. By selecting both ends of the flow volume, it can be observed that the flow is uniform throughout.

**Go to Step 6: Verification & Validation**

[Go to all ANSYS AIM Learning Modules](#)