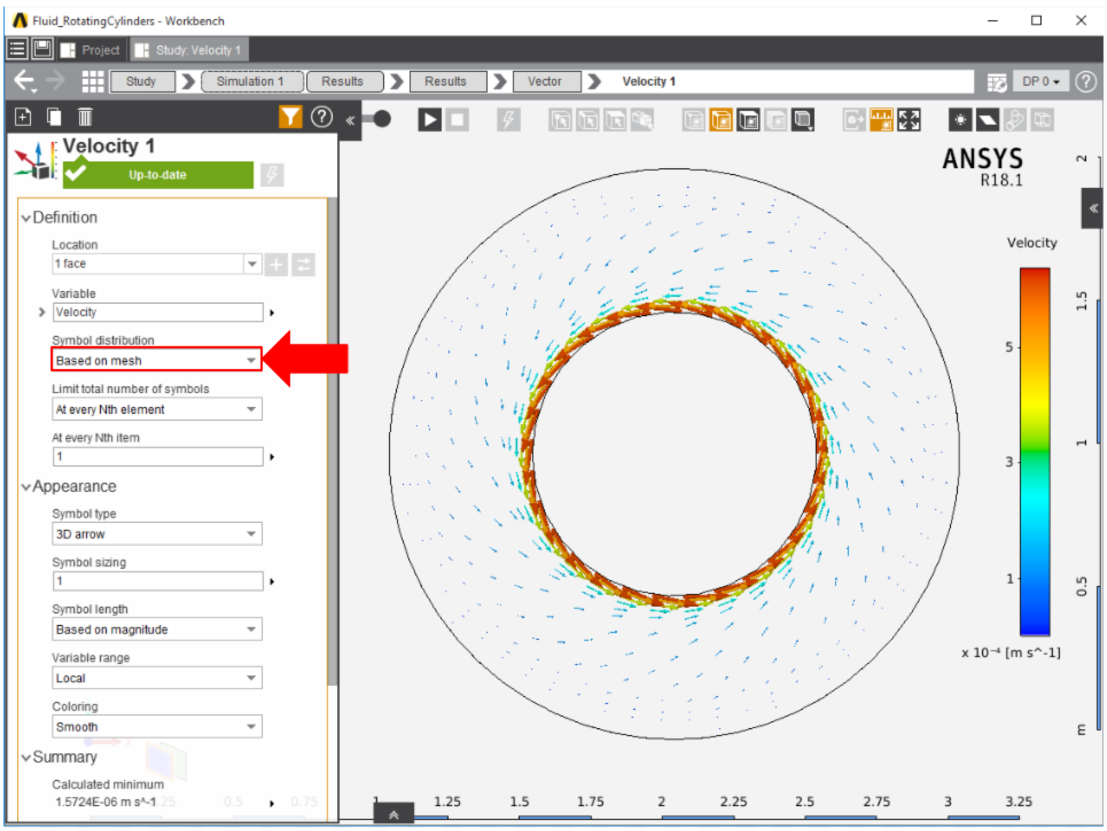
ANSYS Taylor-Couette Flow between Rotating Cylinders - Results

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 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