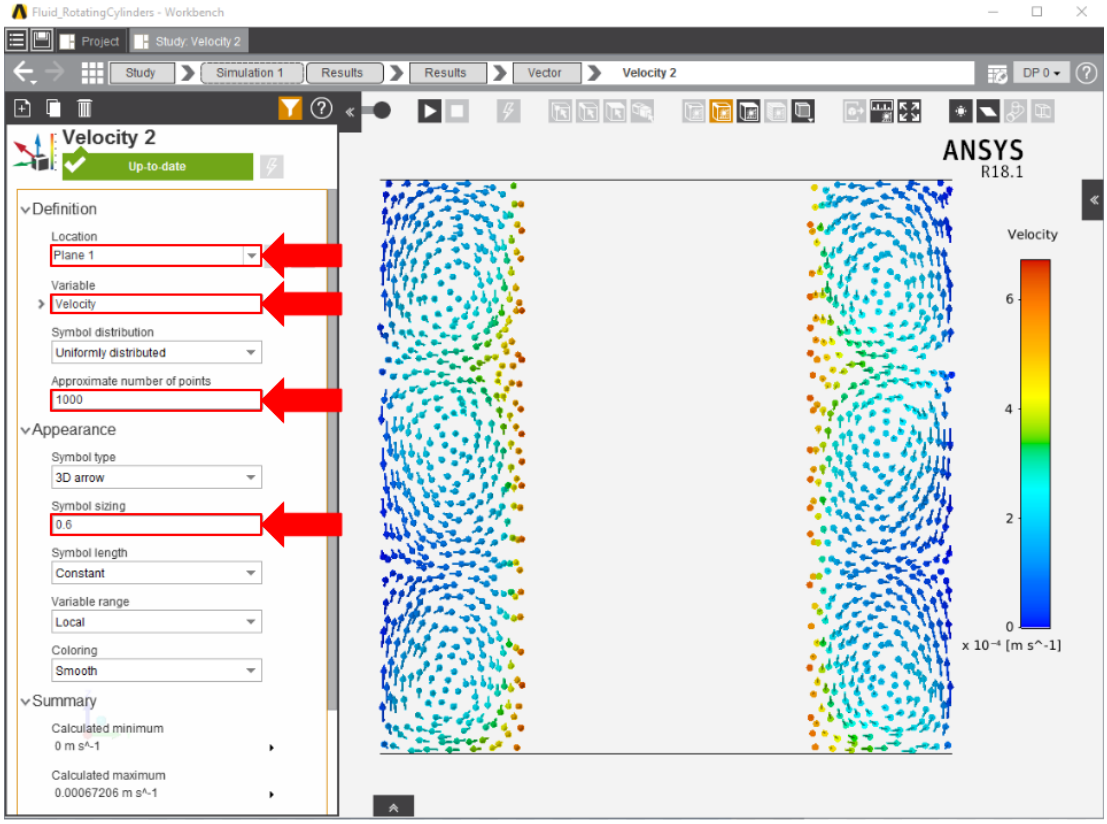
ANSYS Taylor-Couette Flow between Rotating Cylinders - Verification

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

Verification

In order to verify that the Taylor-Couette phenomenon is happening, velocity vectors need to be plotted that bisects the flow to show the recirculation. Select the outside face of the flow region and click the **Add Plane** button. Scroll down to **Axis 1 Orientation** and change **Axis** to **X Axis**. Right click the empty space and select **Add** > **Results** > **Vector**, change the **Location** to **Plane 1** and set the **Variable** to **Velocity**. Set the **Approximate number of points** to 1000 and **Symbol sizing** to 0.6.



It is evident that the Taylor-Couette recirculation regions are happening which means our results match the pre-analysis therefore the simulation is valid.

Reference

Dou, H.-S., Khoo, B.C., and Yeo, K.S., Instability of Taylor-Couette Flow between Concentric Rotating Cylinders, Inter. J. of Thermal Science, Vol.47, 2008, Vol.47, No.11, 1422-1435.

Go to all ANSYS AIM Learning Modules