ANSYS Flow Through an Aortic Aneurysm - 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, select the **Vector** in the **Results** section under **Objects**. To show more vectors, change **Symbol distribution** to **Based on mesh** and input 2 for **At every Nth item**, then press **Evaluate**. To make the vectors easier to see, change the **Symbol sizing** in the **Appearance** section to 0.05 and the **Symbol length** to **Constant**. Press the **Play** button in the model window to see how these velocity vectors develop over time.
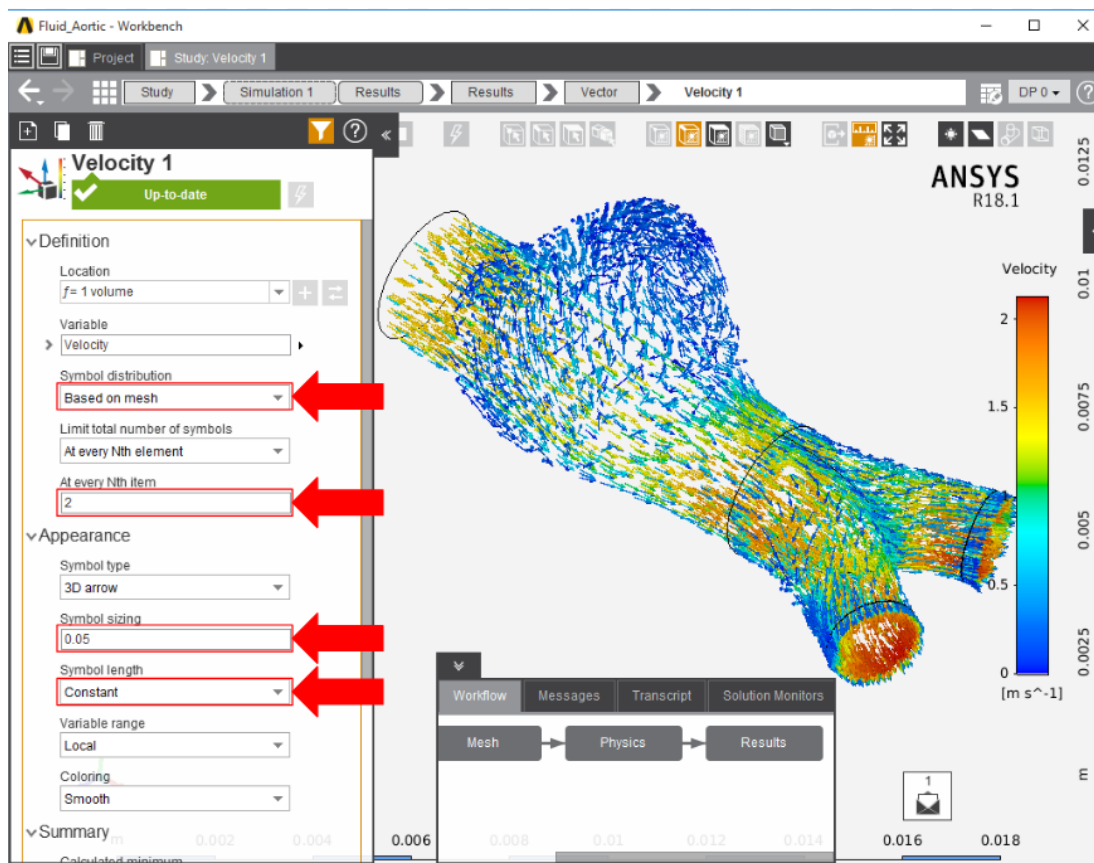

Streamlines are frequently used to display flow patterns. Return to the **Results** panel and **Add** and **Streamline** object. Select the **Inlet** as the **Seed location** and press **Evaluate**. Improve the display by changing **Distribution** to **Based on mesh** and set **N** to 2. Press the **Play** button to animate the streamlines. Experiment with the settings under **Appearance** to see how they affect the display.

To plot the pressure change, create a **Contour** on the walls of the flow volume. Select **Contour** in the **Add** drop down menu, select the **Wall** as the **Location**, and assign the **Variable** to be **Pressure**.

When velocity vectors are arranged in a 3D space, it often becomes difficult to see patterns without moving the object around. A simple way of seeing the patterns inside the flow is to create a plane which cuts across the volume showing the vectors in a 2D space. Select the **Plane** button in the upper right corner to create the construction geometry. Drag the arrows on the compass tool to orient the plane, dividing the aorta down the middle as shown below.
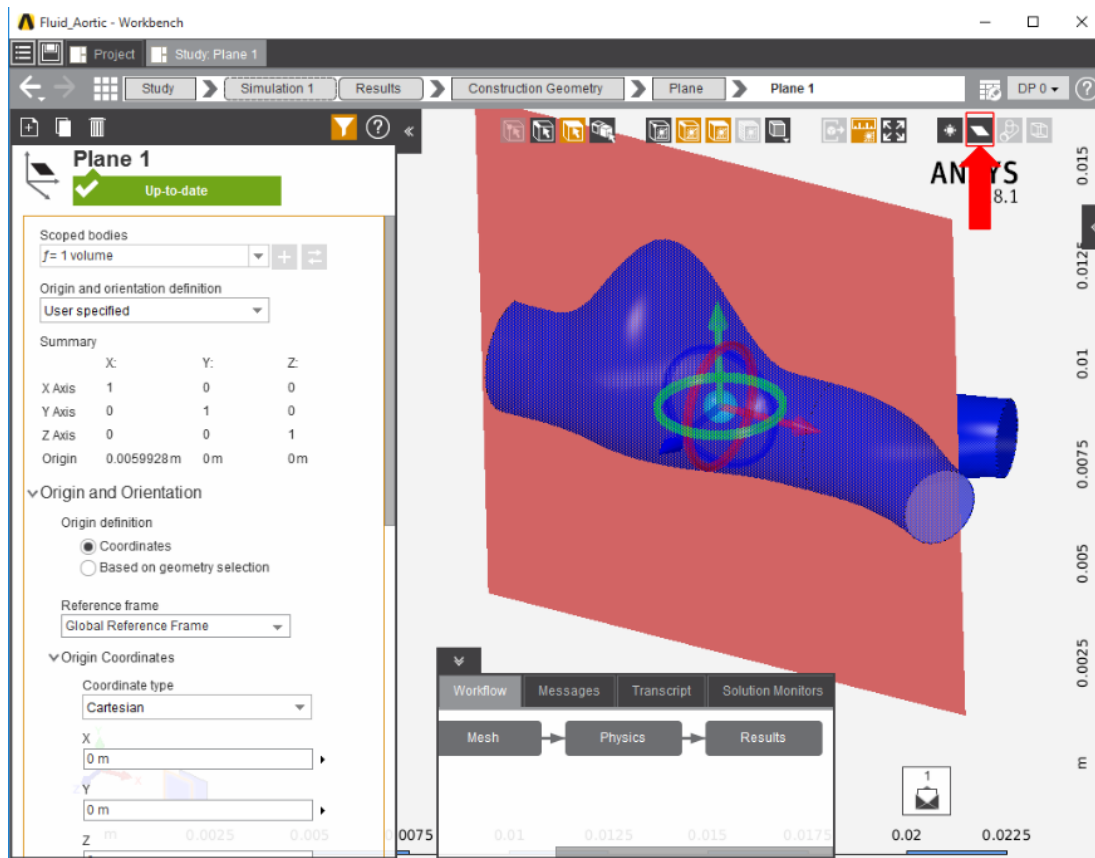
Return to the **Results** panel and select **Vector** from the **Add** dropdown menu. Select **Plane 1** as the **Location**, choose **Velocity** as the **Variable**, and change the **Approximate number of points** to 400. Under **Appearance**, change the **Symbol sizing** to 1 and the **Symbol length** to **Constant**.
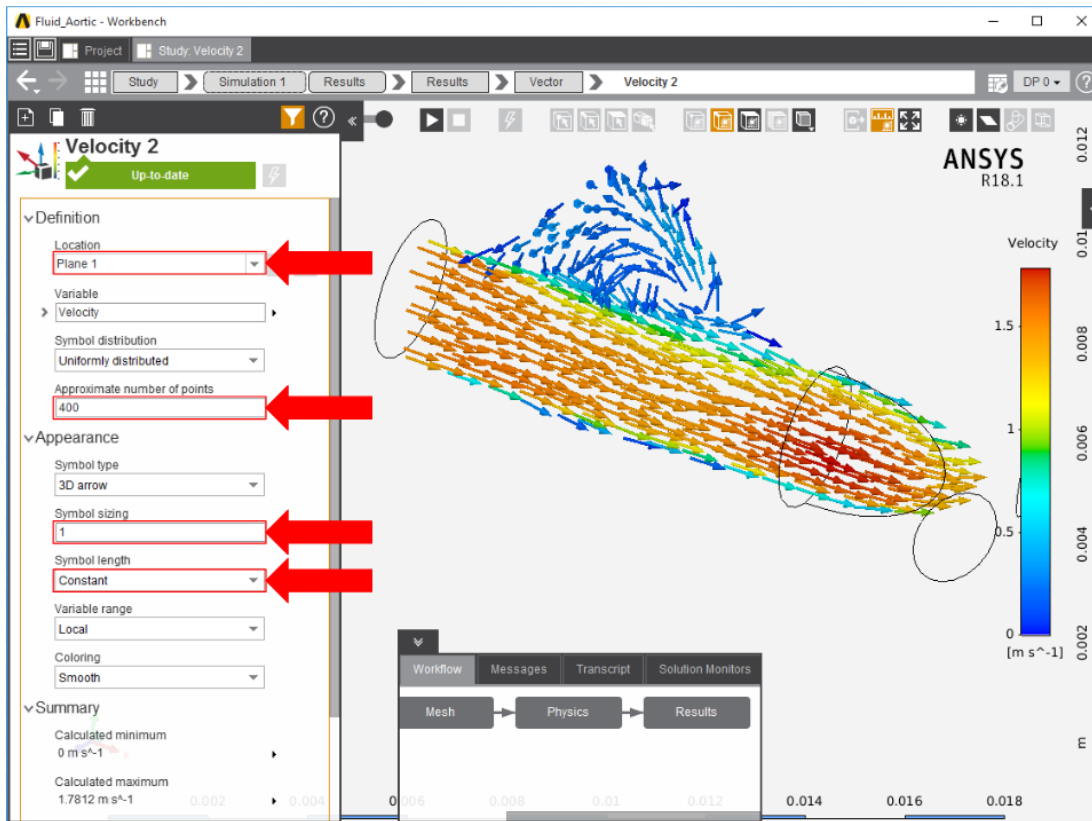
Go to Step 6: Verification & Validation

Go to all ANSYS AIM Learning Modules