

# ANSYS Flow Through an Aortic Aneurysm - Results

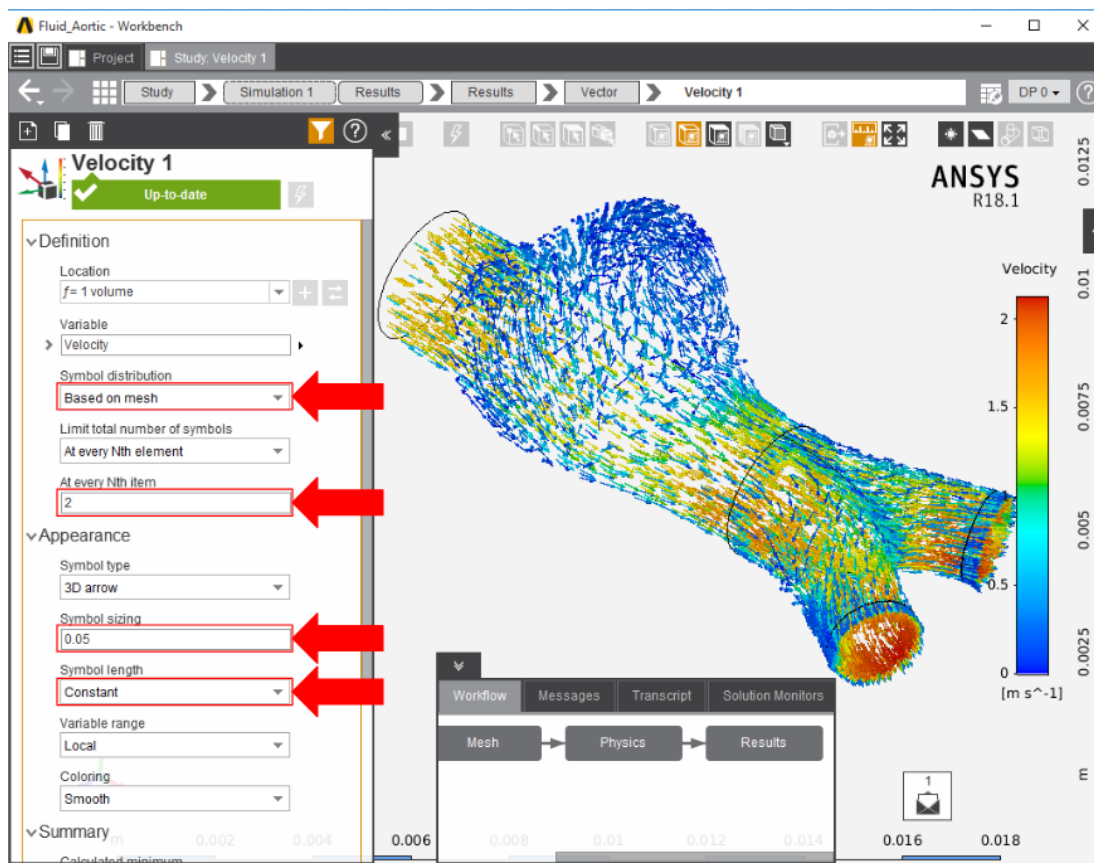
Author(s): Sebastian Vecchi, ANSYS Inc.

## Problem Specification

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

## Solution/Results

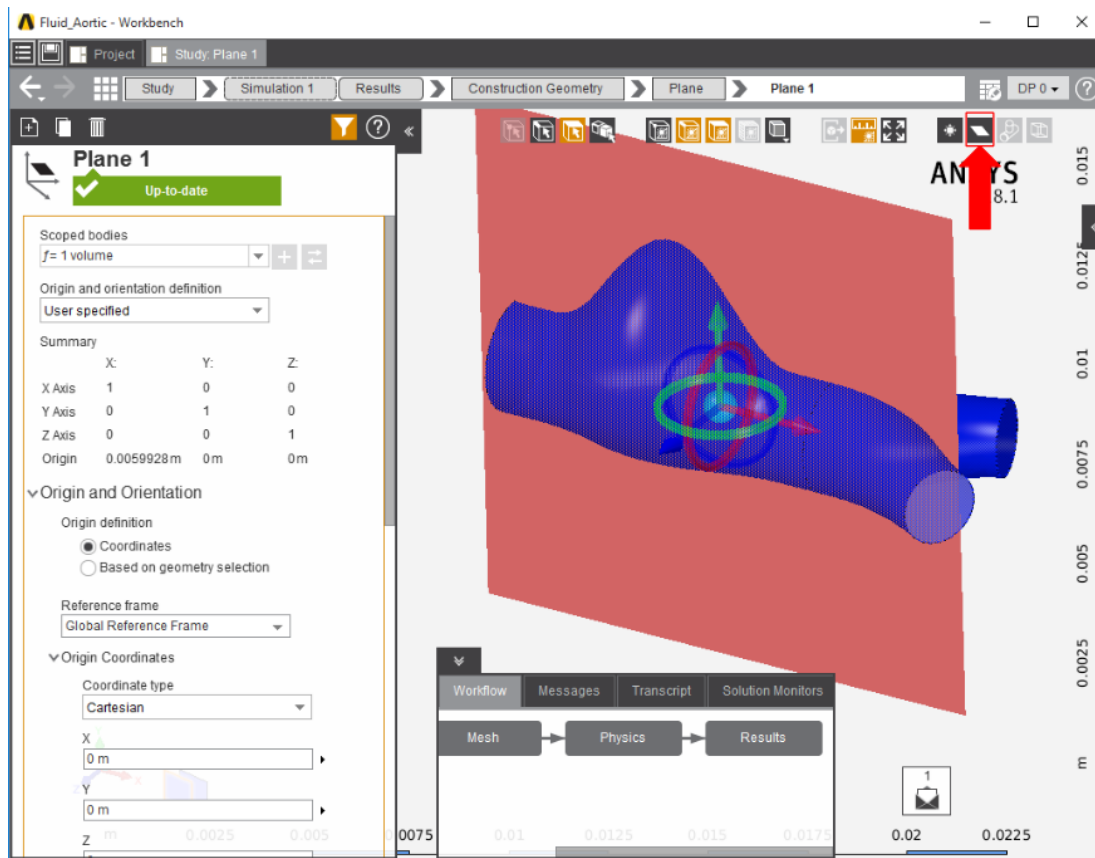
Press the **Results** button in the **Workflow** to extract information from the simulation. In order to find information that can be readily used, first press **Evaluate Results**. Once the evaluation is complete, select the **Vector** in the **Results** section under **Objects**. To show more vectors, 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.



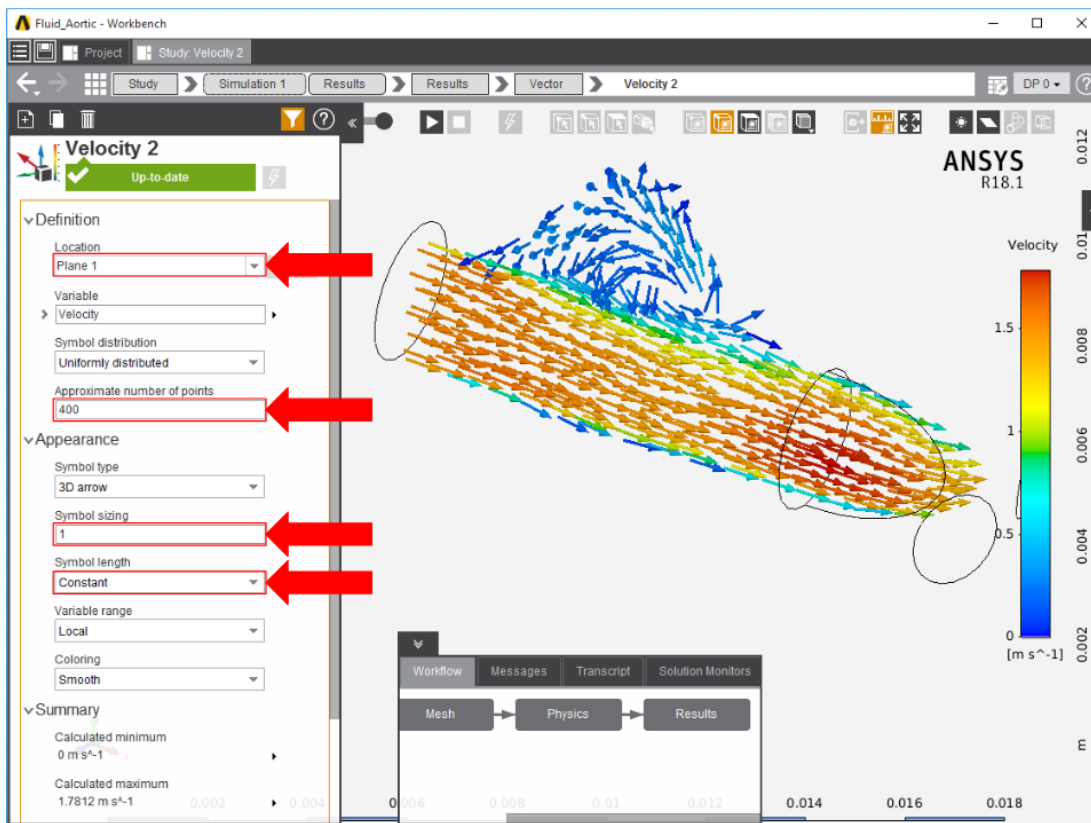
Streamlines are frequently used to display flow patterns. Return to the **Results** panel and **Add** and **Streamline** object. Select the **Inlet** as the **Seed location** and press **Evaluate**. Improve the display by changing **Distribution** to **Based on mesh** and set **N** to 2. Press the **Play** 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](#)