

ANSYS Flow Through U-Duct - Verification

Author(s): Sebastian Vecchi, ANSYS Inc.

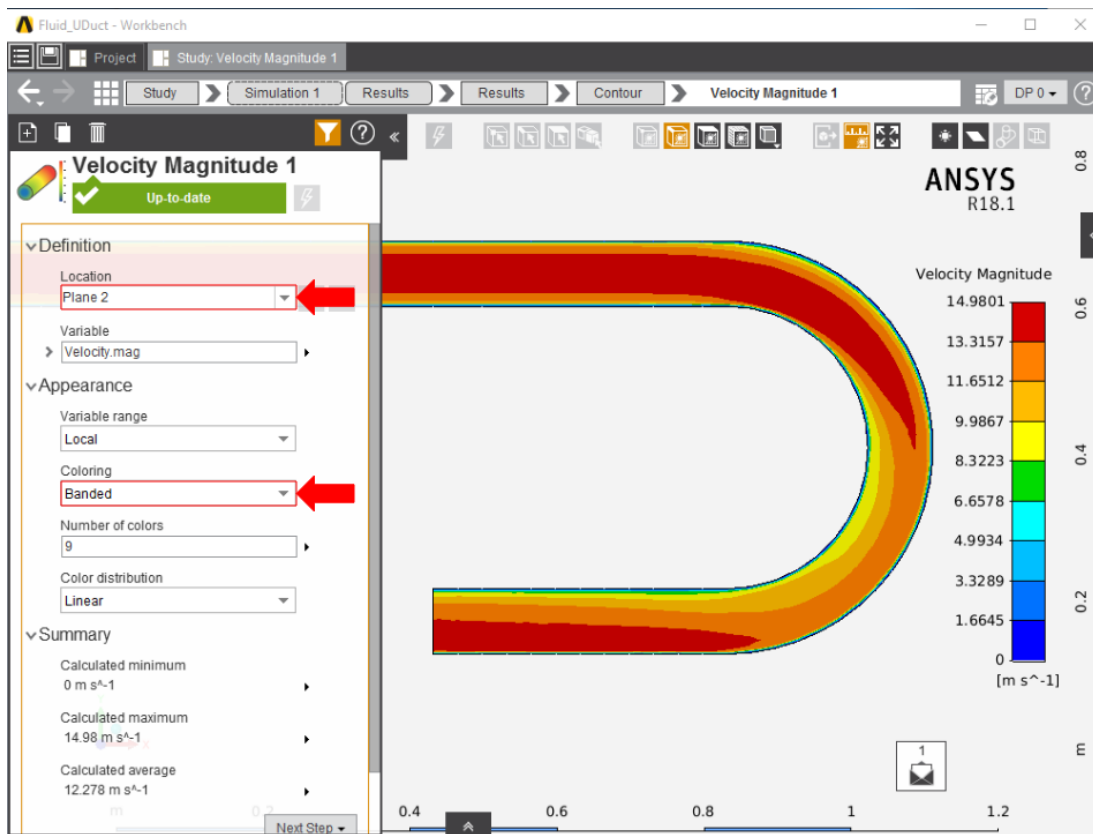
Problem Specification

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

Verification & Validation

A good way of validating the results from the simulation are by comparing them to an outside source which has computed results for the same scenario. In this tutorial, "Flow Through a 180 Degree U Bend" by Azore CFD Simulation Software will be compared to the simulation flow. Below is the flow visualization from the simulation from Azore.

This can be compared to our simulation by creating a velocity contour at the midplane created in the solution section previous. Add a **contour** at the plane and change the **variable** to **velocity magnitude**. In order for the results to be visualized the same, change the **Coloring** to **Banded** in the **Appearance** section. The contour is shown below and it validates our simulation. The velocity contours are extremely similar in both flow shape and velocity.



Reference

"Flow Through a 180 Degree U Bend." Flow Through a 180 Degree U Bend. Azore CFD Simulation Software, n.d. Web. 18 July 2017.

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