ANSYS Taylor-Couette between Cylinders - Startup

Author(s): Sebastian Vecchi, ANSYS Inc

Pre-Analysis & Start Up

Simplified Governing Equations

In the paper "Instability of Taylor-Couette Flow between Concentric Rotating Cylinder" by Hua-Shu Dou, Boo Cheong Khoo, and Khoon Seng Yeo the equation for the critical condition of primary instability is simplified for a concentric rotating cylinder. It is given and solved in the equations below. In order to incite Taylor-Couette flow, K_{max} must be between 370-389 so a value of 380 was chosen as K_{max} for the calculation.

$$K_{max} = \frac{\omega_1 R_1 (R_2 - R_1)}{\nu} = 380$$

$$\nu = \frac{\mu}{\rho} = 8.93 * 10^{-7}$$

$$R_1 = 0.5[m]$$

$$R_2 = 1[m]$$

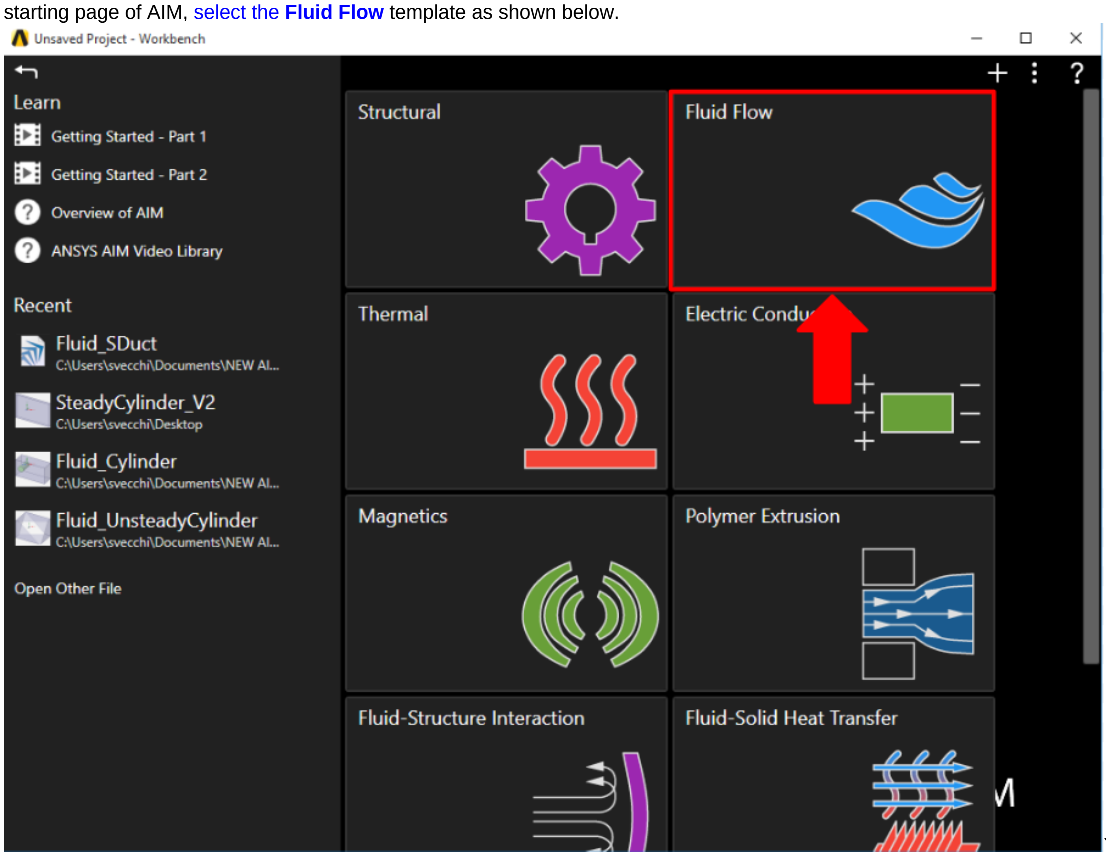
$$\omega_1 = \frac{K_{max}\nu}{R_1(R_2 - R_1)} = 0.00135736\ [rad/s]$$

Start-Up

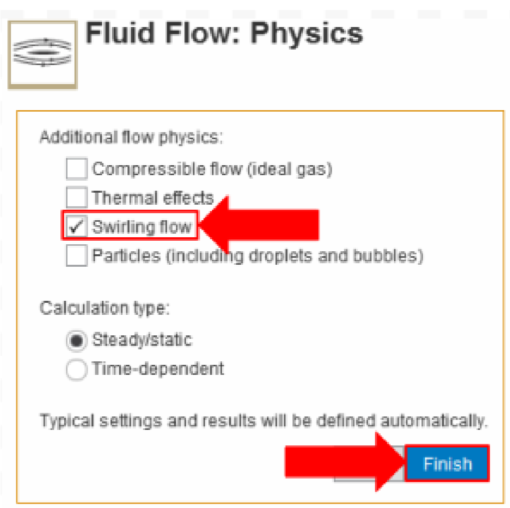
A few words on the formatting in the following instructions:

1. Notes that require you to perform an action are colored in blue
2. General information is colored in black, but does not require any action
3. Words that are **bolded** are labels for items found in ANSYS AIM
4. Most important notes are colored in red

We are ready begin simulating in ANSYS AIM. Open ANSYS AIM by going to **Start** > **All Apps** > **ANSYS 18.1** > **ANSYS AIM 18.1**. Once you are at the starting page of AIM, select the **Fluid Flow** template as shown below.



You will be prompted by the **Fluid Flow** template to either **Define new geometry**, **Import geometry file**, or **Connect to active CAD session**. Select **Define new geometry** and press **Next**. Finally, check the **Swirling flow** box in the **Additional flow physics** and press **Finish**.



Go to Step 2: Geometry

Go to all ANSYS AIM Learning Modules