ANSYS Transonic Flow over a Wing - Start Up

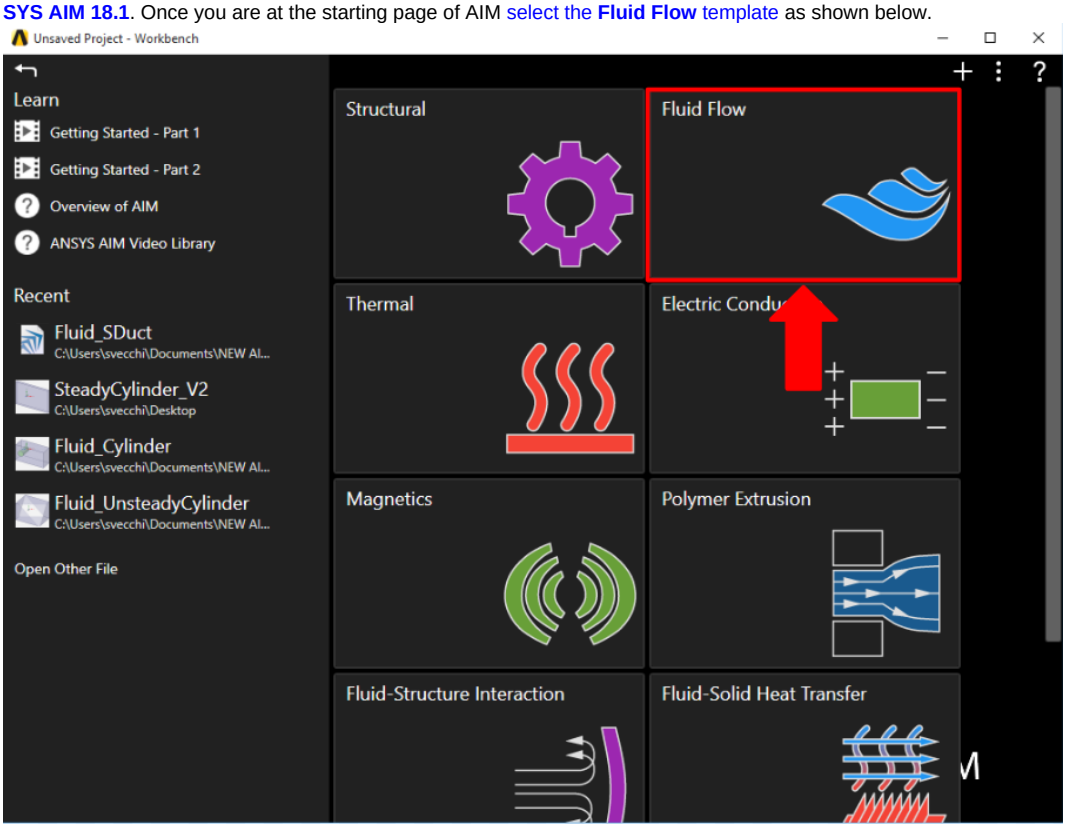
Author(s): Sebastian Vecchi, ANSYS Inc.

Startup

A few words on the formatting on the following instructions:

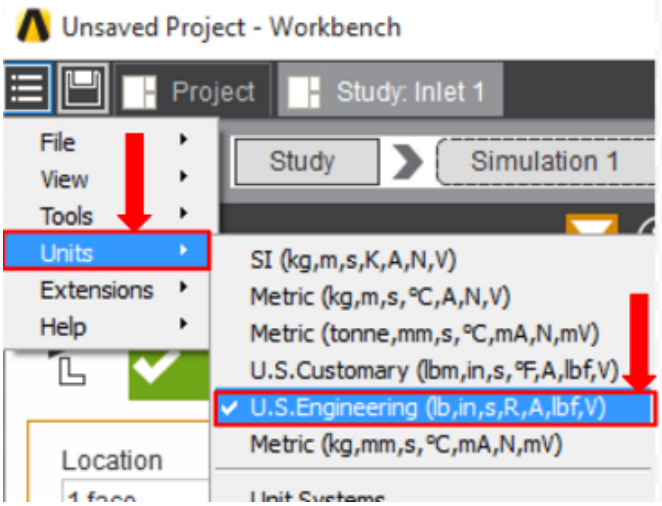
1. Notes that require you to perform an action are colored in blue
2. General information is colored in black, but does not require any action
3. Words that are **bolded** are labels for items found in ANSYS AIM
4. Most important notes are colored in red

Now that we have the pre-calculations, 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.



Units

In the upper left corner, select **Home** > **Units** > **U.S. Engineering**. The units will now match the ones given in the problem specification.



Go to Step 2: Geometry

Go to all ANSYS AIM Learning Modules