ANSYS Compressible Flow over a Wing-Body Junction - Startup

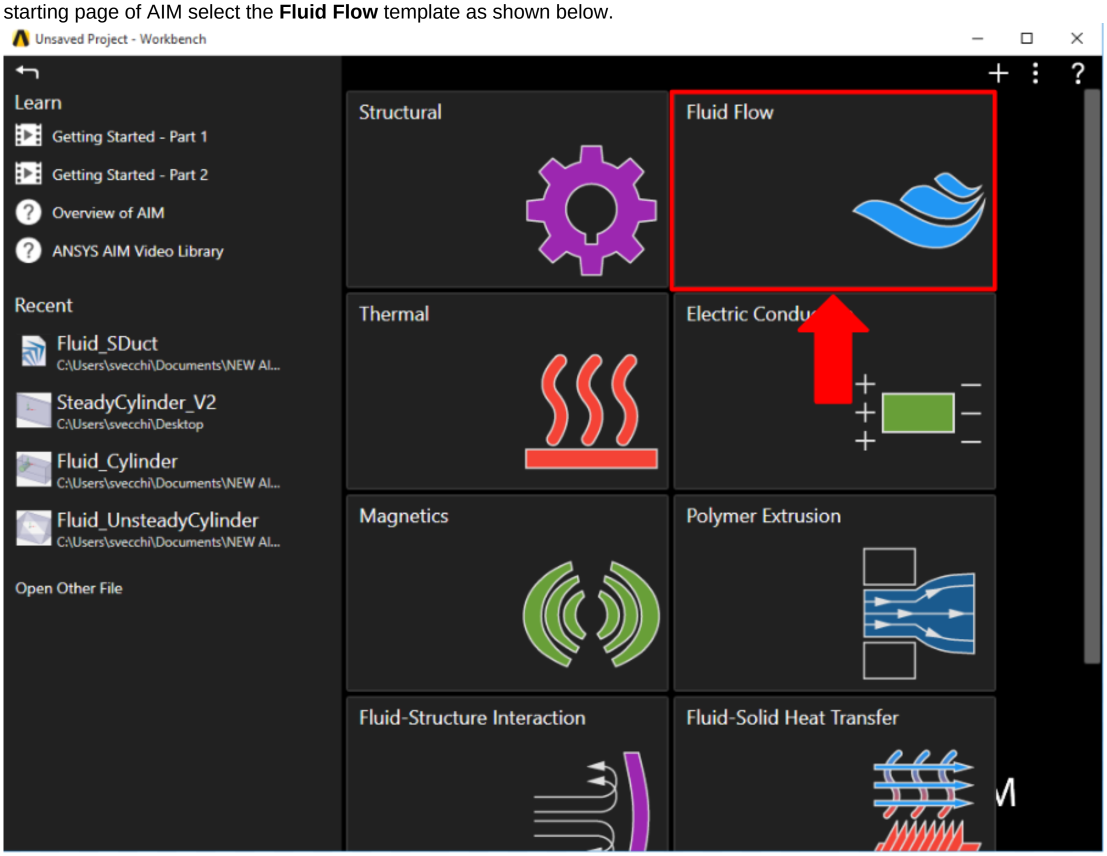
Author(s): Sebastian Vecchi, ANSYS Inc.

Start-Up

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

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 **Import geometry file** and press **Next**.

Go to Step 2: Geometry

Go to all ANSYS AIM Learning Modules