

Compressible Flow Over an Airfoil - Start-Up

Author(s): Sebastian Vecchi, ANSYS Inc.

Problem Specification

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

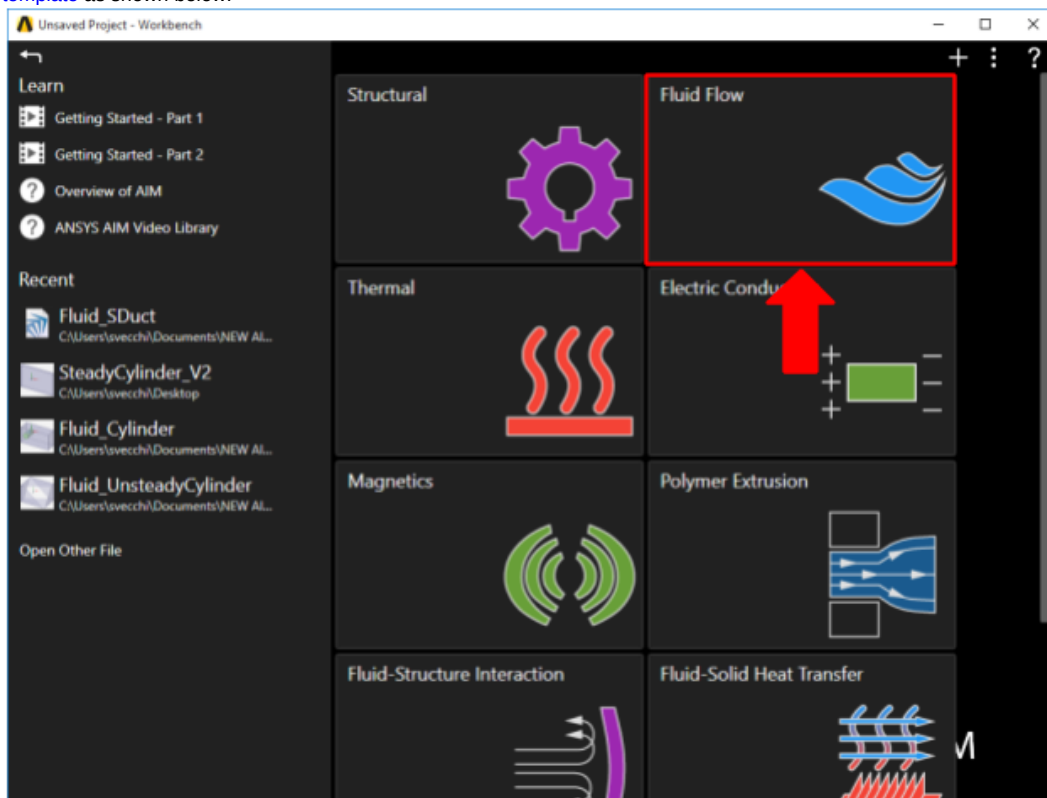
Exercise

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**

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](#)