Compressible Flow in a Nozzle - Pre-Analysis & Start Up

Author(s): Sebastian Vecchi, ANSYS Inc.

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

Pre-Analysis

In quasi-one-dimensional flow for this converging diverging nozzle, we expect that the Mach number at the inlet will be subsonic and accelerating until, at the minimum nozzle area (the throat), the flow becomes sonic with M = 1. Since the nozzle starts to diverge, the fluid will continue accelerating until the exit, where it continues as supersonic. The Mach number at the end of the nozzle can be calculated in the equations below. The diameters of the end and throat area were converted to area using formula (1) and compared using formula (2) to find the Mach number using isentropic flow tables.

$$A = \frac{\pi}{4}D^2 \qquad (1)$$

$$\frac{A}{A^*} = \frac{A_e}{A_{th}} \qquad (2)$$

$$A_e = 962mm^2$$

$$A_{th} = 78.54mm^2$$

$$\frac{A}{A^*} = \frac{962mm^2}{78.54mm^2} = 12.25$$

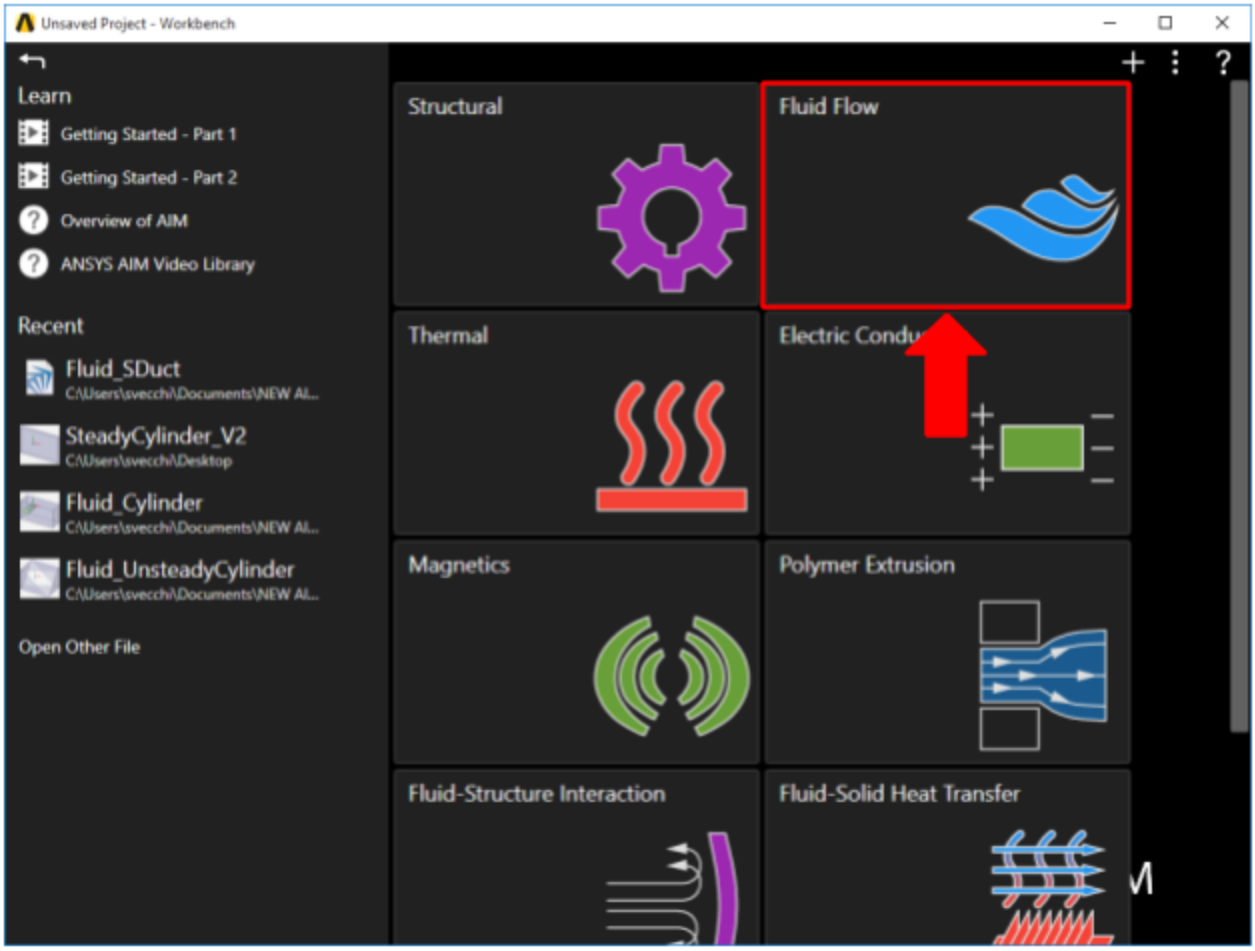
The resulting Mach number at the end of the nozzle is found to be 4.15, which confirms that the exit flow is supersonic.

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

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.



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