AIM Lid-Driven Cavity - Pre-Analysis

Author(s): Sebastian Vecchi, ANSYS Inc.

Pre-Analysis & Start Up

Governing Equation

The following nondimensional equations govern conservation of mass and momentum.

$$\frac{\delta u}{\delta x} + \frac{\delta v}{\delta y} = 0$$

$$\frac{\delta(u^2)}{\delta x} + \frac{\delta(uv)}{\delta y} = \frac{-\delta \rho}{\delta x} + \frac{1}{Re}\left(\frac{\delta^2 u}{\delta x^2} + \frac{\delta^2 u}{\delta y^2}\right)$$

$$\frac{\delta(uv)}{\delta x} + \frac{\delta(v^2)}{\delta y} = \frac{-\delta \rho}{\delta y} + \frac{1}{Re}\left(\frac{\delta^2 v}{\delta x^2} + \frac{\delta^2 v}{\delta y^2}\right)$$

These equations can be combined in order to create a governing equation that will dictate the flow in our lid-driven box. Below is the equation which was created when combining the above.

$$\frac{\delta(u\varphi)}{\delta x} + \frac{\delta(v\varphi)}{\delta y} = \frac{\delta}{\delta x}\left(\Gamma^{\varphi}\frac{\delta \varphi}{\delta x}\right) + \frac{\delta}{\delta xy}\left(\Gamma^{\varphi}\frac{\delta \varphi}{\delta y}\right) + S^{\varphi}$$

The terms on the left govern the net convection flow in the volume while the terms on the right govern net diffusion, with the exception of S which represents source generation of the flow.

Reynold's Number

Using the equation below, the Reynolds number can be calculated for the problem. This will allow us to predict how the flow will behave inside the box before running the simulation. It is specified in the problem that the inside length of the box is 0.1 m and the speed of the lid is 1 m/s. Since the fluid inside of the box is water, the density is 997.05 kg/m^3 and the viscosity is 0.00089002 N*s/m^2. These values can be found online or later on in the Material properties section of the Physics template.

$$Re = \frac{\rho u L}{\mu} = 112,025$$

The resulting value from the Reynolds number calculation indicates that the flow will be turbulent and will produce multiple eddies.

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 will be colored in black, but do not require any action
3. Words that are **bolded** are labels for items found in ANSYS AIM
4. Most important notes will be colored in red

Start-Up

Now that the pre-calculations are finished, we are ready to begin the simulation in ANSYS AIM. Open ANSYS AIM by going to **Start** > **All Apps** > **ANSYS 18.1** > **ANSYS AIM 18.1**. Once the starting page has opened, 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**, press **Next**, then press **Finish**.

Go to Step 2: Geometry

Go to all ANSYS AIM Learning Modules