

AIM Lid-Driven Cavity - Geometry

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

Geometry

Draw Geometry

There are two ways of creating the flow volume which will be tested. The first is to make a cube and the second is to make a hollow box and extract the volume. The first option would be very straight forward, but the second is more realistic in the physical world. This tutorial will explore the latter option because the first can be done by using this tutorial and simply not creating the hollow box.

First, we will begin by creating the outside boundary of the cube. Since we are interested in the volume inside the cube, the outside boundary is obsolete. In order to simplify meshing, a wall thickness of 0.02m was picked; therefore, the outside width of the cube will be 0.12m. [Select the Rectangle sketch tool, check the Define rectangle from center box, and drag in the model window to make a square with edges 0.12m long.](#) (While dragging to create the rectangle, you can press the spacebar, then enter 0.12m for the first dimension, then press the tab key, then enter 0.12m for the second dimension.)

Once the square drawing has been created, use the [Pull tool in the Edit section of the Design tab](#). In order to maintain the inside space while considering the wall thickness, [the drawing was pulled 0.12m in the positive Y direction by clicking on the surface and dragging it out.](#)

[Click on the Rectangle tool and then select the top surface of the box.](#) Then, [create another square using the rectangle sketch tool that has 0.1m length.](#) Make sure the Define rectangle from center option is selected to create a symmetrical design.

[Use the Pull tool on the sketch, move it 0.1m in the negative Y direction to create the 0.1x0.1x0.1 cavity inside the square.](#) We have now successfully created the space for the volume from the problem specification that will be the focus of this simulation.

The last step in the **Geometry** section is to create the fluid volume which will hold the material being simulated. [In the Prepare tab of the Model Editor, select the Volume Extract tool in the Analysis section.](#) This will allow us to create a volume by selecting the capping surface and a seed face for our volume. [The capping surface will be the topmost face of the box while the seed face can be any face inside of the box.](#) The image below can be used as reference when creating the volume.

[Press the green checkmark](#) once the selections are made and the volume will be generated. AIM will automatically switch to a translucent model view so that the inside volume can be viewed with ease.

Suppress

Now that the geometry of the flow volume has been created, we can suppress the outer box from the physics calculation. [Right click the Solid in the geometry tree and select Suppress for Physics.](#)

[Go to Step 3: Mesh](#)

[Go to all ANSYS AIM Learning Modules](#)