

There are two versions of SketchUp: SketchUp Pro and SketchUp Make. You want to install **SketchUp Make** because it is free software. You can download from this web site:

<https://www.sketchup.com/products/sketchup-make>

Below is a “getting started” lesson on using SketchUp for making mechanical drawings (also known as technical drawings). A proper mechanical drawing has all the information needed for a machinist to create a part by using the drawing for reference. The drawing must have sufficient dimensions included so that the machinist can create the part properly.

In this lesson you will create a simple, solid block. It does not have any holes drilled in it. In order for the machinist to produce the block, he or she only needs to know the three dimensions (length, width and height). Since this is such a simple object, there really is no need to make a drawing of it, but it is a good starting point for you to learn how to create a mechanical drawing with SketchUp. After you have the basics of creating a block, you can learn how to create holes in the block, remove material from the block to create various shapes, etc.

This lesson is based on the YouTube video with link provided below. The video was created using an Apple computer. Since most of our members will be using computers running Windows, I have added the instructions for the lesson for a Windows machine. Nevertheless, watching the video should also help you with the lesson. Just keep in mind that the Windows version of SketchUp has a different user interface (the menu structure is somewhat different).

<https://youtu.be/zcXGmVixDAI>

First, install SketchUp on your computer using the link near the top of this page.

After installation, start SketchUp

On the start screen click **Start using SketchUp** (button at bottom right corner)

Open the **File** menu and select **New**. Then use **Save** in the **File** menu to save the file.

If a template selection dialog appears, select "Woodworking – Millimeters, otherwise, open the **Window** menu at top of screen and select **Preferences/Template - Woodworking - Millimeters**


Open the **View** menu and select **toolbars**. In the dialog, make sure the box for **Large Tool Set** is checked (all other boxes should not have a check in them). This will allow you to see all of the tools you need to



use on the toolbar. Especially important is the **Dimension tool**, which looks like this:


Open the **Window** menu and select **model info**. On left side of dialog, select **units**. In **Format** slot select **Decimal** and in slot to the right of that select **cm**. In the **Precision** slot select **0.00cm**. Your drawing is now configured to use the metric system of measurement using the unit of centimeters (cm). The precision of the drawing will be 0.01 cm (*i.e.*, one hundredth of a centimeter).



Click on the Rectangle tool , move mouse and click at origin (where blue, green and red lines intersect), then release the mouse button. Start to move mouse to make a rectangle, then type on the keyboard **4cm, 2cm** and press **Enter** key. This will set the size of the rectangle to 4 centimeters by 2 centimeters (you can see the size listed at the bottom right of the screen).

The rectangle will probably be very small on the screen so you will need to zoom in. First, select the **Pan** tool (looks like a hand) and then use the mouse to drag the rectangle to the center of the screen. Then rotate the mouse scroll wheel to zoom in the view, using the **Pan** tool as needed to keep the rectangle centered on the screen.



Select the **Push/Pull** tool  and click on the surface of the rectangle, release the mouse button, move mouse up slightly, then type **3cm** and press the **Enter** key. You should now have a block 4 cm long, 2 cm wide and 3 cm high.

Mechanical drawings are commonly done by **orthographic projection** with a **front view, top view** and **side view**. These three views combined usually are sufficient to properly depict an object. An orthographic projection uses parallel lines for edges of the object in each plane. This is different than drawing an object in perspective, where the edges are not parallel, but converge on a vanishing point. This is done when an artist wants to make a realistic-looking drawing of an object, where parts that are more distant from the viewer appear smaller. In engineering drawings we usually don't want to use perspective. We want the lines to be parallel so that we can use a scale (ruler) to measure distances on the object. Now you will configure your drawing to use orthographic projection.

Open the **Camera** menu and select **parallel projection**.

Now you will create the standard front, top and side views of your object. You will also create an **isometric** view, which makes the object appear like a 3-D object.

Open the **Camera** menu and select **standard views/front**

Open the **View** menu and select **animation/add scene**

Locate the **Scenes** panel (probably at right-bottom of screen). Using the scroll button at the far right side of the screen, scroll down until you can see all of the **Scenes** panel. Near the bottom of the panel you will find a **Name** slot for the scene. Right now that name will probably be **Scene one**. Change the name to **Front View**.

Now you will repeat the procedure to create the other views.

Open **Camera** menu and select **standard views/top**

Open **View** menu and select **animation/add scene**

Rename scene to "top view"

Open **Camera** menu and select **standard views/right**

Open **View** menu and select **animation/add scene**


Rename scene to "right view"

Open **Camera** menu and select **standard views/iso**

Open **View** menu and select **animation/add scene**

Rename scene to "isometric view"

Select the front view scene (tab at upper left of window)

Select **Dimension** tool , click bottom-left corner of rectangle then click bottom-right corner, then move mouse down a little and click again. This will apply a dimension label to the length of the block at the bottom

Click on top view tab

Use **Dimension** tool to add dimension to left side of block

Click on right view tab

Use **Dimension** tool to add dimension to right side of block

Click on isometric view tab

Now you will see all three dimensions of the block. Your drawing is now complete and is sufficient for a machinist to create the block for you. However, there is no easy way to print off all three views (front, top and side) on one sheet of paper, as is usually done in a mechanical drawing. It is possible to create a drawing that has all three views and you can learn how to do this in another lesson.

