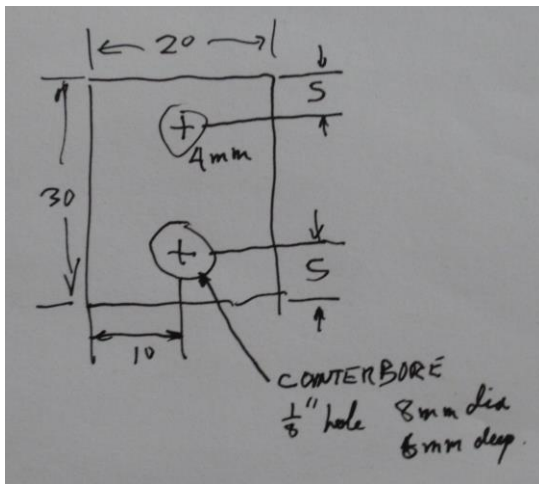


Sketchup Getting Started

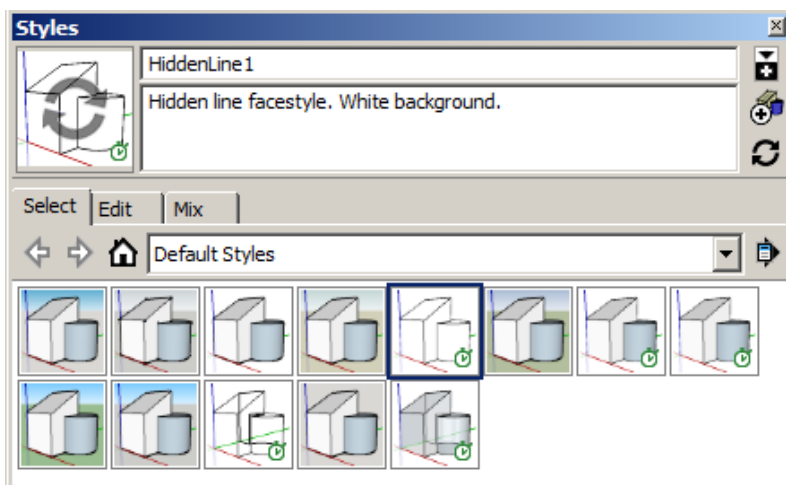
Design:

I always sketch a design on paper as a starting point. I do this because I generally need to measure and gather information about the design which I can easily write down on paper. As an example if I want to create a mounting plate for a power supply I would start with a rough paper sketch of the design without any dimensions. I would then use my calipers or a datasheet to gather the dimensions and add them to the drawing. Finally depending on the mounting hardware I would measure up the diameter of the holes, depth and diameter of the bolt heads in case I want a countersink or counter-bored hole. With all of the details written down I am finally ready to capture the design in Sketchup.

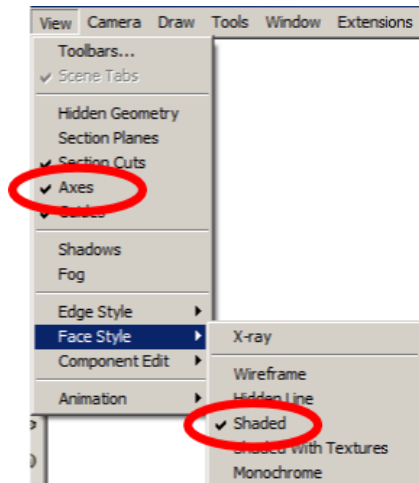


Setup:

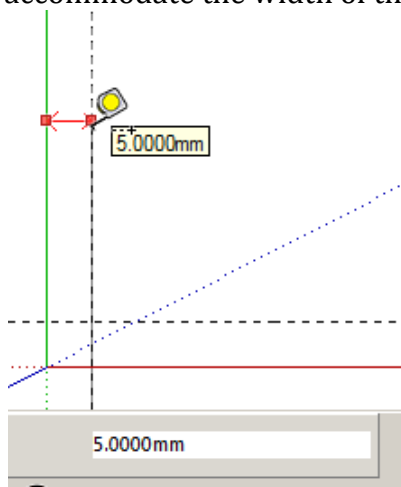
To setup a design that will be cut on a CNC machine I start Sketchup using the HiddenLine style sheet. This is found in the **Windows, styles** menu of Sketchup.



Turn the axis on as well as shading. The axis is important because the SketchUCam software will only generate g-code if the design is located entirely in the upper right quadrant. Turning shading on is also required for SketchUCam to function properly. Unless shading is on you will not be able to use SketchUCam to select objects to cut out.

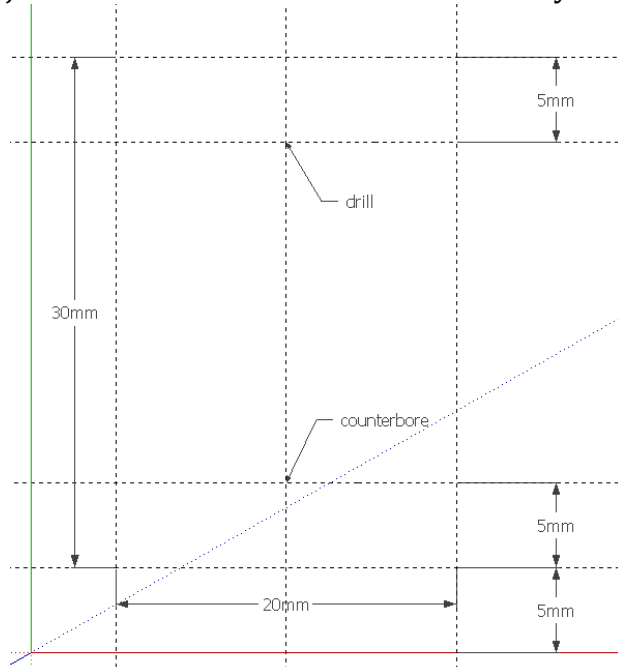


Next I use the guideline tool to draw a vertical and horizontal guide parallel to the X and Y axis. The guideline should be offset by 3mm if you plan on using a 1/16" end mill or by 5mm if you plan on using a 1/8" end mill. This offset is added to ensure that the final design is entirely in the upper right quadrant. The offsets accommodate the width of the end mill, for the final outline cut of your part.

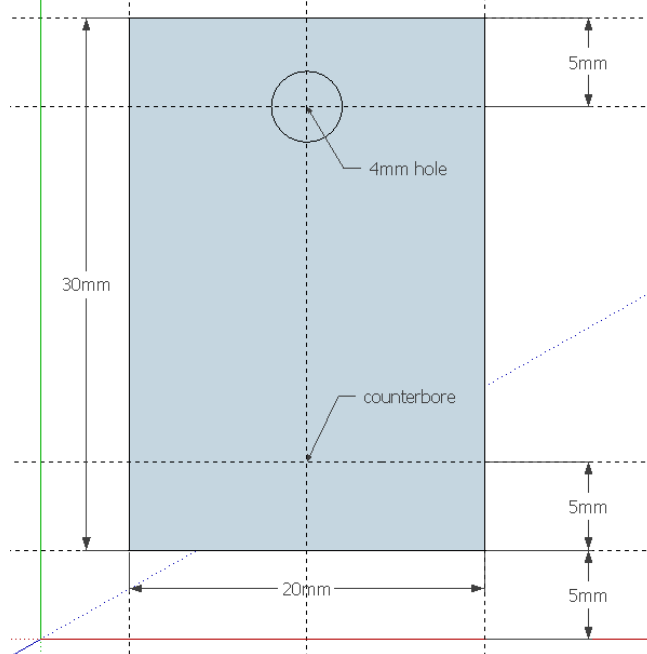


Computer Aided Design using Sketchup

Open a new file and create a blank template as described above. With your paper design in hand I then draw guidelines that match the paper sketch. Later these guides allow me to draw the part and everything just snaps to the guideline grid you just created. I added dimensions so that you can see what I am doing.



Finally I draw the part outline using the pencil and circle tools. I use the circle to cut large holes, smaller holes and countersink holes are added in SketchUCam. I finally add dimensions and comments. The final design should match your paper design.









Computer Aided Manufacturing using SketchUCam

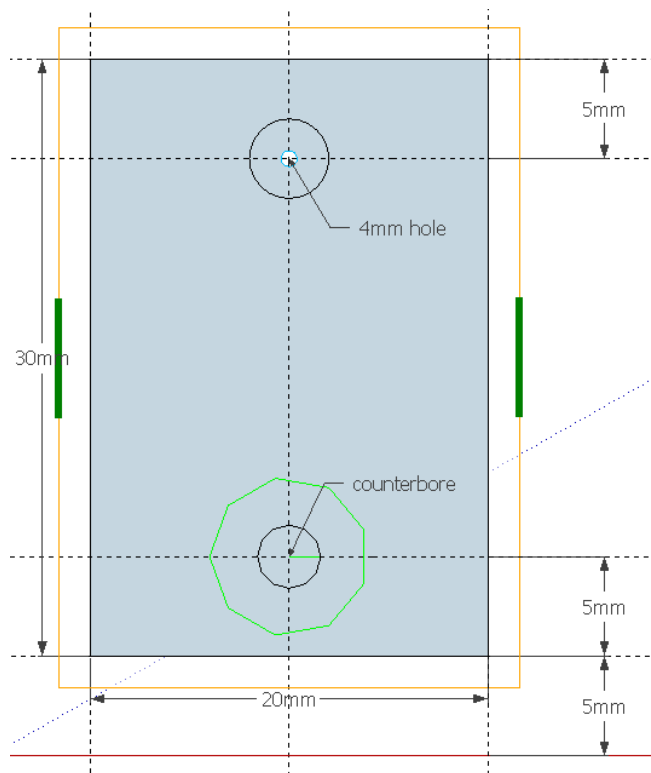
SketchUCam allows you to convert the Sketchup design into a series of g-code commands that the CNC machine will use to cut your part. This is a CAM (computer aided manufacturing) process. Several design properties dictate how you will proceed. First you need to know the diameter of the end mill that you will be using and the exact thickness of the material your design will be cut from.

The first step is to setup Sketchucam for your design. For beginners start with a 1/8" two flute up cut end mill. This mill bit is relatively robust and long enough to cut up to 1/2" material. Hopefully you won't break your bit. The thickness of the material should be measured with calipers because it is often not what you think it is. The final machine setup parameters are the feed rate and mill bit spindle speed. These vary from material to material but here are a few standard recipes to get you started.

	1/8" end mill			1/16" end mill		
	feed mm/s	Plunge mm/s	cut depth mm	feed	plunge	cut depth
MDF	890	380	1.6	800	800	2.54
HDPE	600	350	2.0	500	300	1.4
Baltic ply	800	800	2.54	800	800	2.54

Once Sketchucam has been configured you can use the Sketchucam tool to select features to machine. I like to sequence the CAM operations in the order in which I want to cut the part. For example, if you were to cut the part outline first the part would become loose and subsequent CNC operations wouldn't be accurate. In general I find it best to start by cutting the smallest features first and then work up to the larger features so that the work piece remains as rigid as possible. I start by defining all of the drill holes . Next I add counter bore and countersink holes . Then use the inside cut tool  to cut inside the line of any holes. Finally I use the outside cut tool  to cut the part outline. I add tabs  to the design so that the part doesn't come free during the final cut. The final part will need to be cut free from the material with a knife after the design has been machined. Push the  button to generate the *.cnc file.

The simple example that we have used so far will look as follows when you have completed the CAM operations.



Resources

<https://www.sketchup.com/plans-and-pricing#for-personal>

<https://github.com/swarfer/sketchucam/releases?>

<https://camotics.org/>

Suppliers

End mills

<https://www.ebay.com/str/carbideplus>

Canus Plastic

<http://canusplastics.com/>

Baltic Birch

<https://kjpselecthardwoods.com/>

Plastic

<http://plas-techfab.com/>

Plastic

<https://www.johnstonplastics.com/montreal/>