

CamBam Tutorial for Laser Cutter V2

This tutorial will walk you through a couple of the basic tools provided in CamBam and how to configure the different elements so that they can be cut on the Laser Cutter V2.

Note:


The rear left corner of the cutting bed on Laser Cutter V2 is the home position, or x:0, y:0 coordinate. In CamBam, the Y axis runs upwards on your screen, so imagine that your CamBam design will be cut “face down” on the cutting bed.

This doesn't really matter for most cuts but if you are engraving a pattern, it will be etched on the **back** of the material. If the material isn't transparent, you won't see the pattern so you'll need to flip your drawing on the Y axis in CamBam so the cutter engraves it “face up”. See the engraving section below for more info on how to do this. If you have a material that is different on one side, you will want the front side to be facing down on the cutting bed or, again, flip the drawing and do it face up.

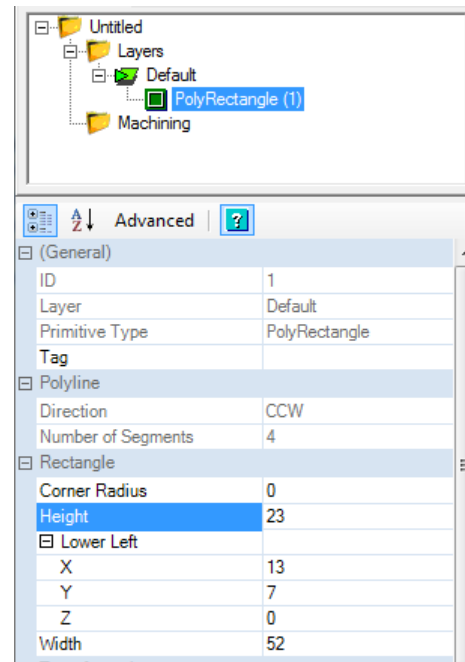
Basic Tips



Hold SHIFT while clicking-and-dragging to move stuff around

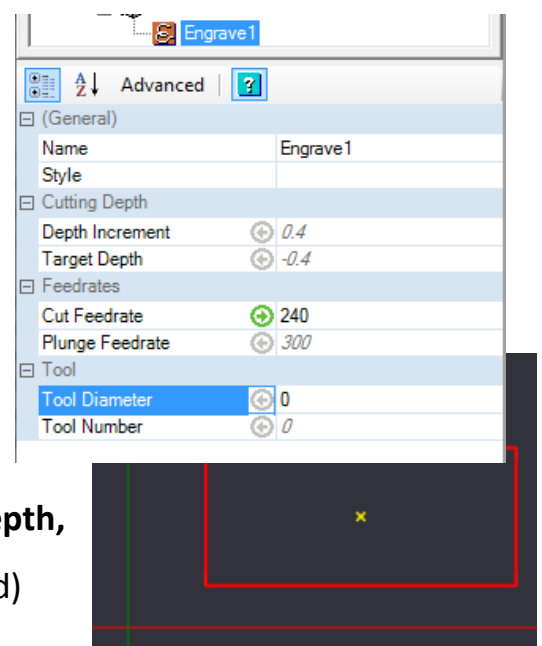
Cut a Rectangle

- To start simple... we'll cut a rectangle. Click the  button to create one.
- Click once on the preview area to specify the first corner, then again to specify the opposite corner.

- You'll notice in the example to the right that we've drawn **above and to the right** of the 'home' or 0,0 coordinate. This means our drawing is in the positive area of the y and x axis. You want all of your drawing elements to be in the positive coordinates.




- You can set the properties of your rectangle in the options on the left-hand side. The "Lower Left" category of values determines the 'position' of the rectangle on the cutting bed (it is calculated from the lower left corner). Width and Height change the size.
- Now that we have our rectangle, we need to tell CamBam what to do with it when we go to perform the cut.
- With the rectangle selected, click the  icon to create an "Engrave" (note that we are doing a **cut** not an **engrave**, even though we use this tool. The difference is in the **feed rate**)
- An engrave cuts exactly on the line you've drawn. This is fine for most cuts but if you need precise dimensions you'll want to account for the width of the laser beam, which you can do with the "profile" tool (use the  button)
- There are four options of significance for the Engrave; **Depth Increment**, **Target Depth**, **Cut Feedrate** and **Spindle Speed** (pictured)





- You'll need to set Depth Increment to a positive number and Target Depth to the negative equal of Depth Increment. It doesn't matter what the numbers are as they pertain to the Z axis, which the laser cutter doesn't use, but we want these number to be equal so that CamBam thinks we only need to do one 'machining' pass, ie. not multiple layers.
- The Cut Feedrate is the speed at which we cut. 240mm/s works well for acrylic up to a few mm thick. If you only want to engrave and not cut, a federate above 4000 is good.
- The spindle speed is used to specify the laser power (%). Anything number over 100 uses full power.

Engrave Letters

- To create a text element, click the  button.
- Enter the text you want, choose a font, size and style, then click OK.
- Click anywhere to place the text.

At this stage, you have a number of options for how you might like to cut the text. You can use an **engrave** or a **profile** at a cutting federate to cut the letters out of the material 'stencil style', or you can do a **pocket** at a faster federate which etches the entire letter shape onto the material, which is what we'll do in this example.

- Click the  button to create the **pocket** object.
- The following options highlighted with a  are the important ones you need to pay attention to:

[-] Cutting Depth	
Clearance Plane	3
Depth Increment	0.4
Final Depth Increment	0
Stock Surface	0
Target Depth	-0.4
[-] Experimental	
Transform	
[-] Feedrates	
Cut Feedrate	5000
Plunge Feedrate	300
[-] G-Code Options	
Custom MOP Footer	
Custom MOP Header	
Start Point	
Velocity Mode	Undefined
Work Plane	XY
[-] Lead In/Out	
Lead In Move	
Lead Out Move	
[-] Options	
Collision Detection	True
Cut Ordering	Depth First
Milling Direction	Conventional
Optimisation Mode	New (0.9.8)
Region Fill Style	Horizontal Hatch
Roughing / Finishing	Finishing
[-] Spindle Control	
Spindle Direction	CW
Spindle Range	0
Spindle Speed	100
[-] Step Over	
Finish Stepover	0
Finish Stepover At Target I	False
Max Crossover Distance	0.7
Roughing Clearance	0
StepOver	0.4
Stepover Feedrate	Plunge Feedrate
[-] Tool	
Tool Diameter	0.4
Tool Number	0
Tool Profile	Unspecified

Lets go through what these options do:

- Once again, you need the **target depth** to be the negative equal of the **depth increment** so CamBam calculates a single pass.

- The feedrate for an etch is significantly faster than a cut – here we've used 5000.
- The **region fill style** determines how the tool moves across the pattern. Horizontal hatch does it in horizontal lines, but feel free to try the other options.
- **Roughing/finishing** is of little significance in laser cutting but selecting **Finishing** may produce better quality toolpath calculation.
- The **spindle speed** is used to determine the laser power by the firmware. This is a percentage, so a number of 100 or greater will use maximum power. You can try lower power and/or a higher feedrate if you like.
- Lastly, we set the **tool diameter** so CamBam knows how to space the paths the tool follows. A value of 0.4 seems pretty accurate for the LCV2 but you can measure the width of a cut if you need to know the exact value.

Flipping Your Drawing on the Y Axis

- To flip an object on the Y axis, click Edit > Transform > Mirror.
- Click once on the preview window to start the mirror, then click again immediately to the right (and at the same Y coordinate) to flip the selection on the Y axis – it's best that you try this out to see how it works.

Exporting the G-Code

- Firstly, you'll want to order the machining tasks (engraves, pockets, etc) by clicking and dragging them in the tree-view so that the final cut-out of the

shape occurs last. This stops the piece moving as it's cut free from the material while it is still etching / doing other stuff. I.e. Work from the inside out.

- Right-click the “part” (Part 1 by default) and select “Produce Gcode”
- Save your file somewhere ready for importing into Pronterface!