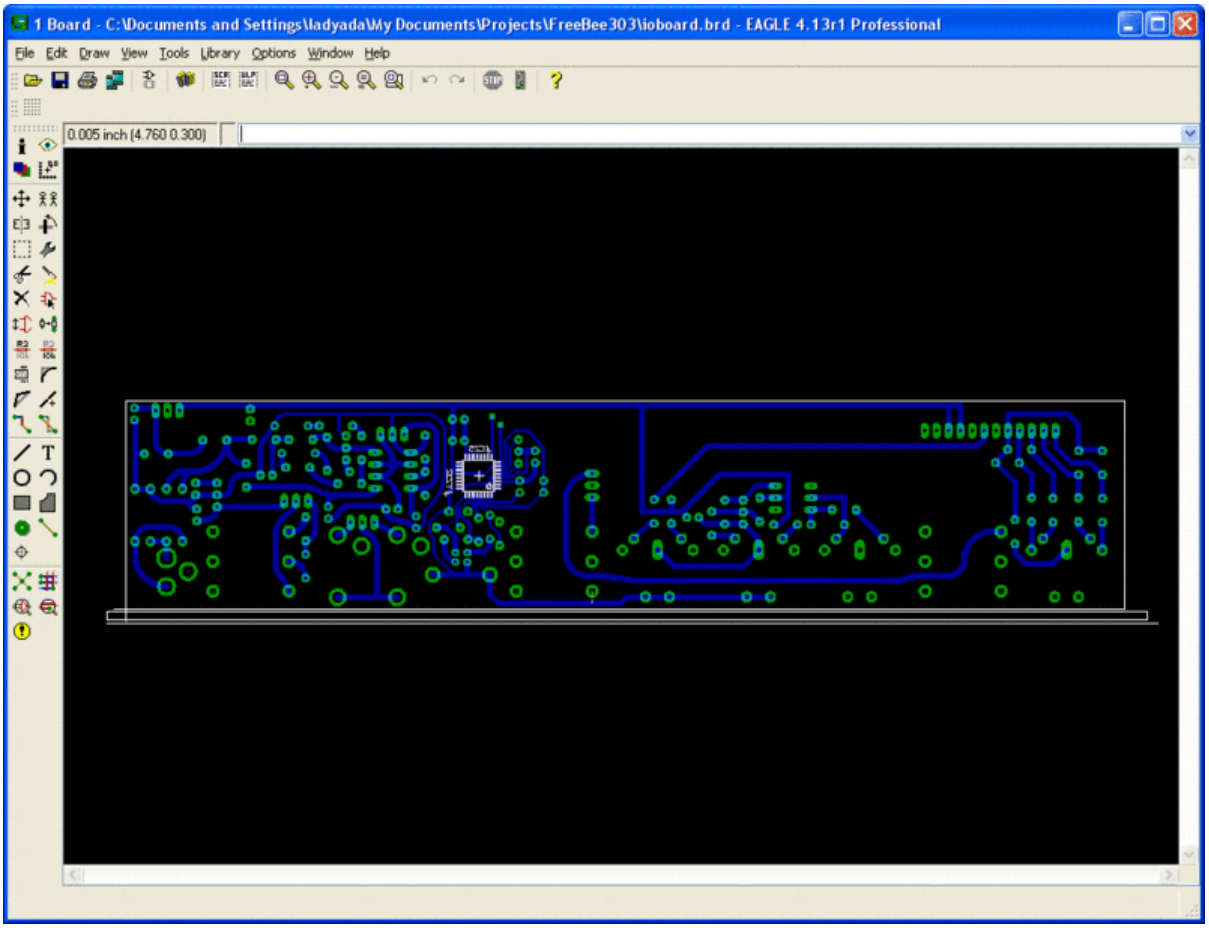
Laser Cut PCB Stencils

Created by lady ada



https://learn.adafruit.com/laser-cut-pcb-stencils

Last updated on 2023-08-29 02:16:13 PM EDT

Table of Contents

Overview 3

Export Cream Layer Gerber 3

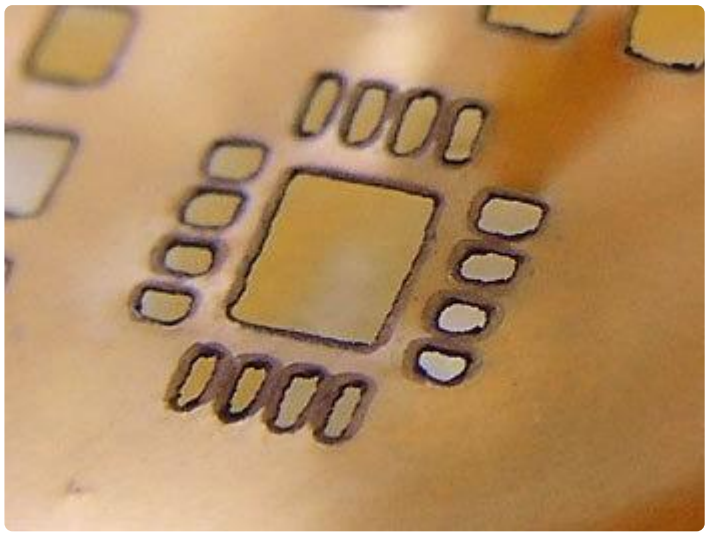
Import Gerbers For Editing 4

Swell Pads 6

Export 8

Cut! 9

Overview



Thanks to Ryan O'Hara at Ohararp.com () for this information, he provides a stencil cutting service and is recommended!

You'll need:

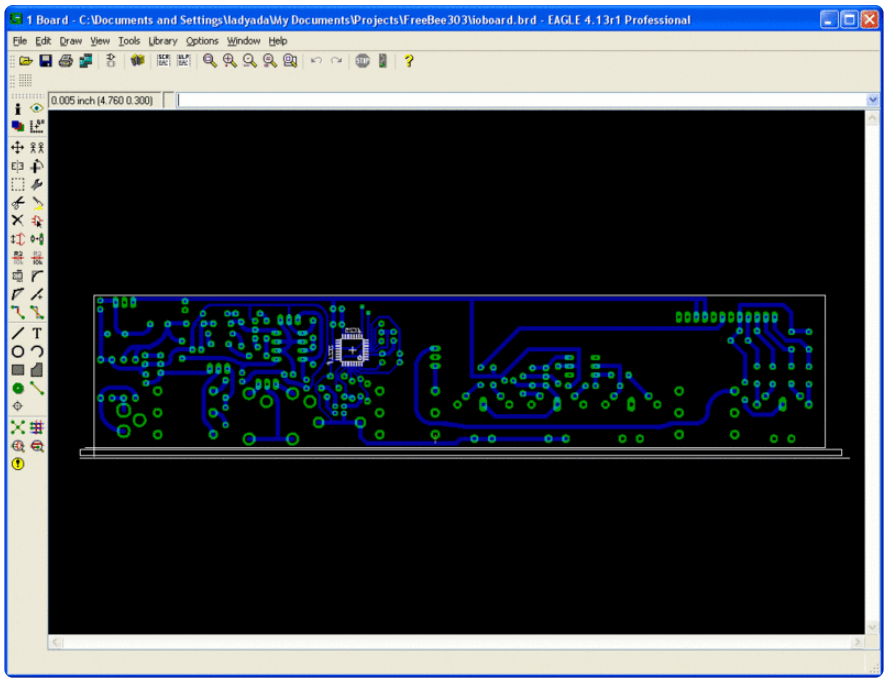
1. A laser cutter
2. Kapton film, I like the 2 mil thick 1 ft square sheets from McMaster-Carr ()
3. Solder paste such as Kester No-Clean ()

Software:

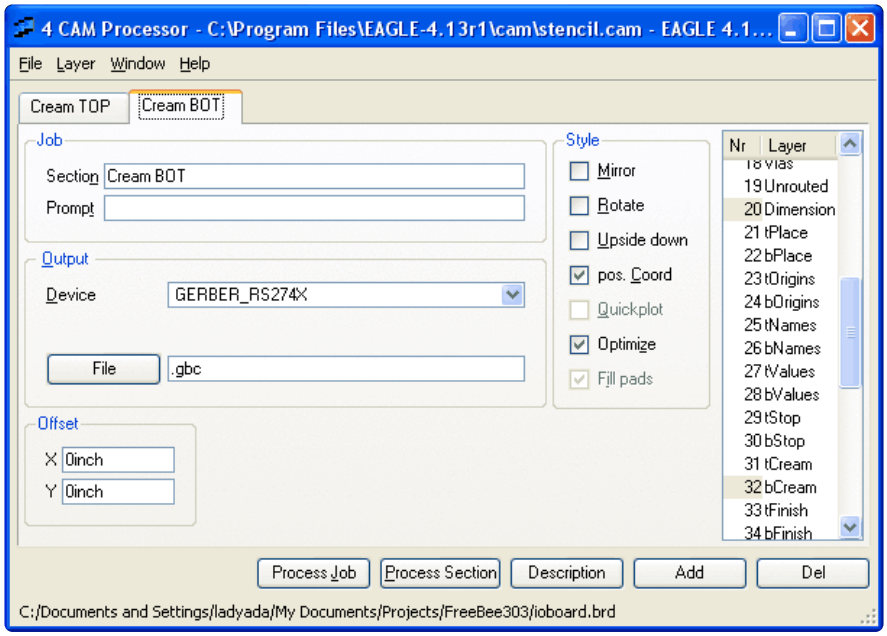
1. PCB layout software (well, thats how I do it) - this example will use EagleCAD
2. Pentalogix ViewMate Gerber viewer software ()
3. PDFCreator () or some other free PDF printer

Export Cream Layer Gerber

This is the PCB we'll be making a stencil for. It only has one chip but of course you can use a more complex layout.



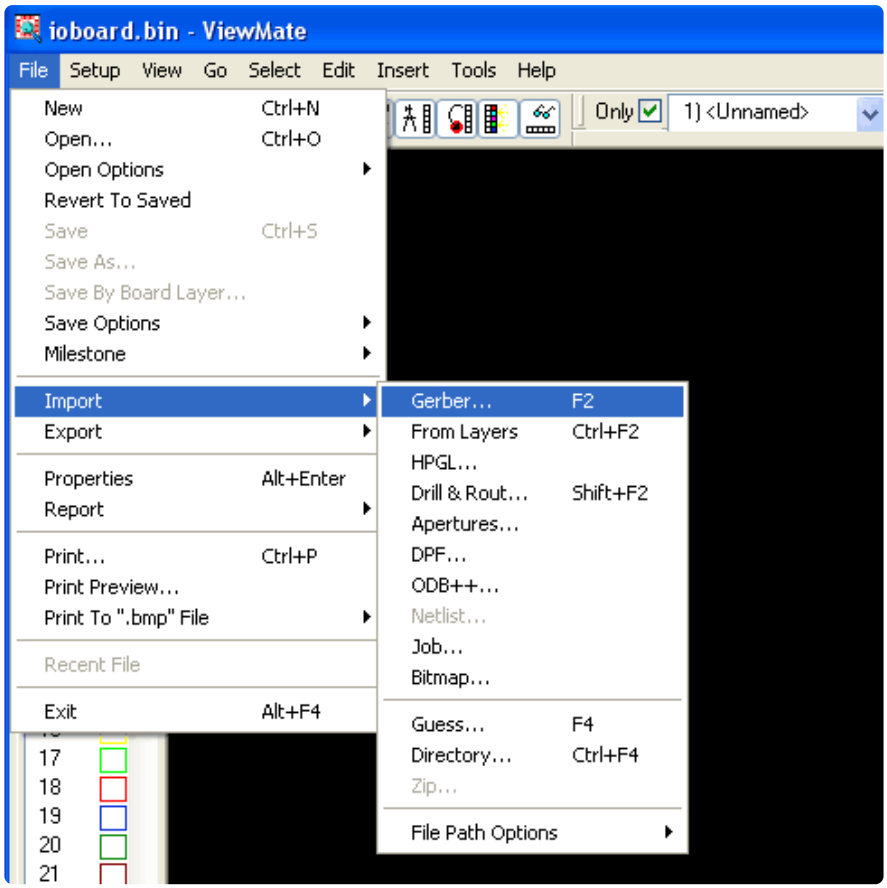
Your PCB software should be able to create/export the Cream Layer (solder paste layer) in Gerber RS274x format. In Eagle you can make your own Job for this quite easily.

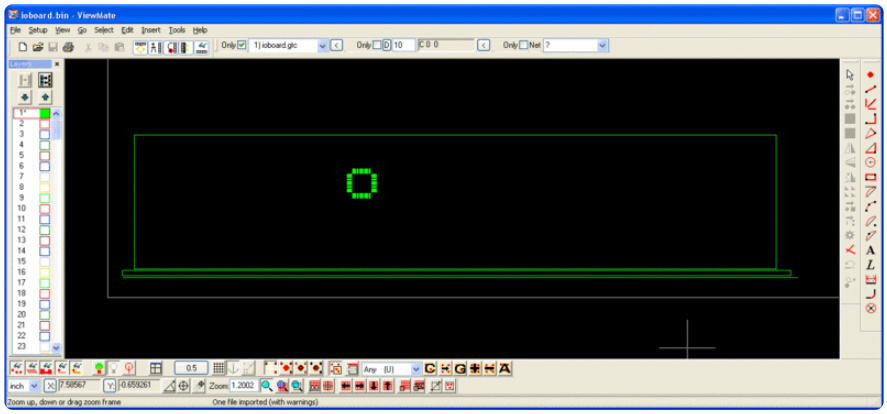


Also export the Dimension layer (PCB outline) since that will help a lot in registration.

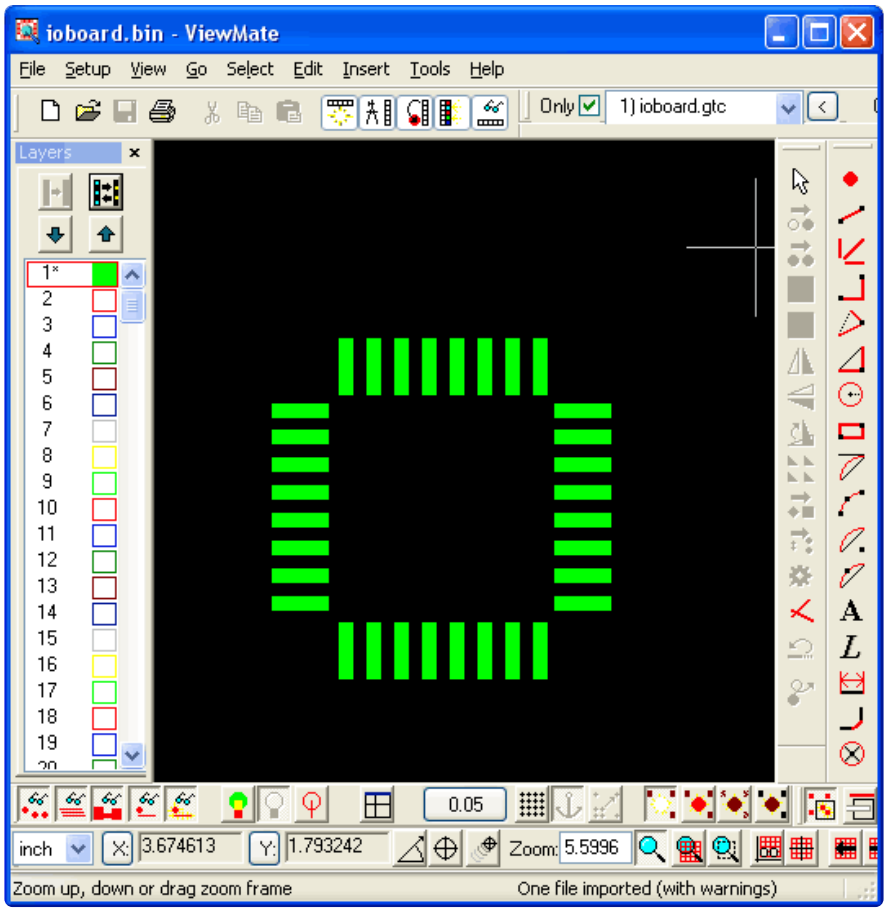
Import Gerbers For Editing

Start up Viewmate and File>Import>Gerber one of the Gerber files generated.





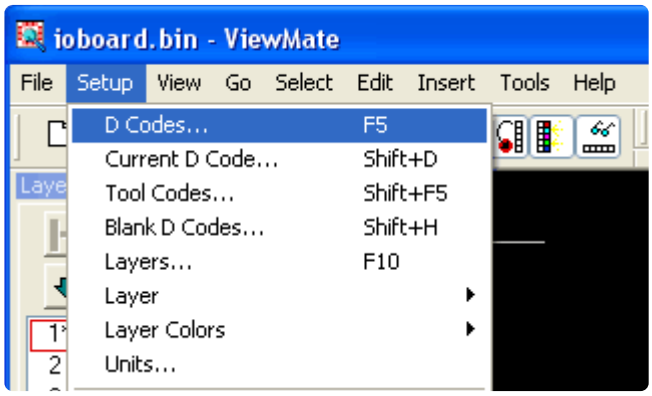
You can zoom in using the Magnifying glass tool.



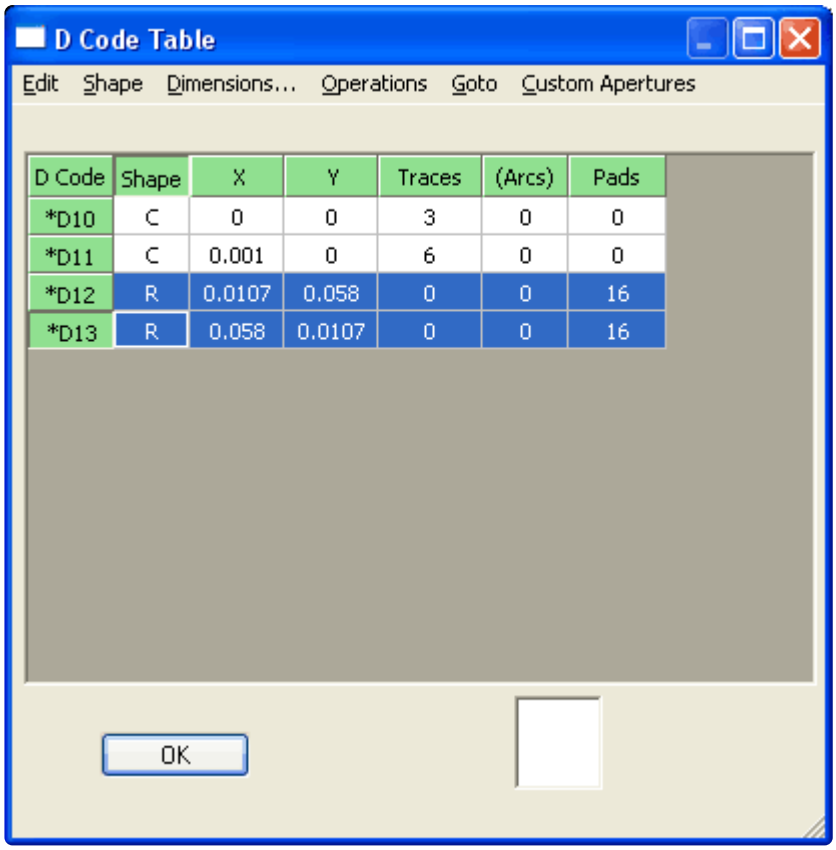
Swell Pads

Next we will make minor adjustnents to shrink the pads a little.

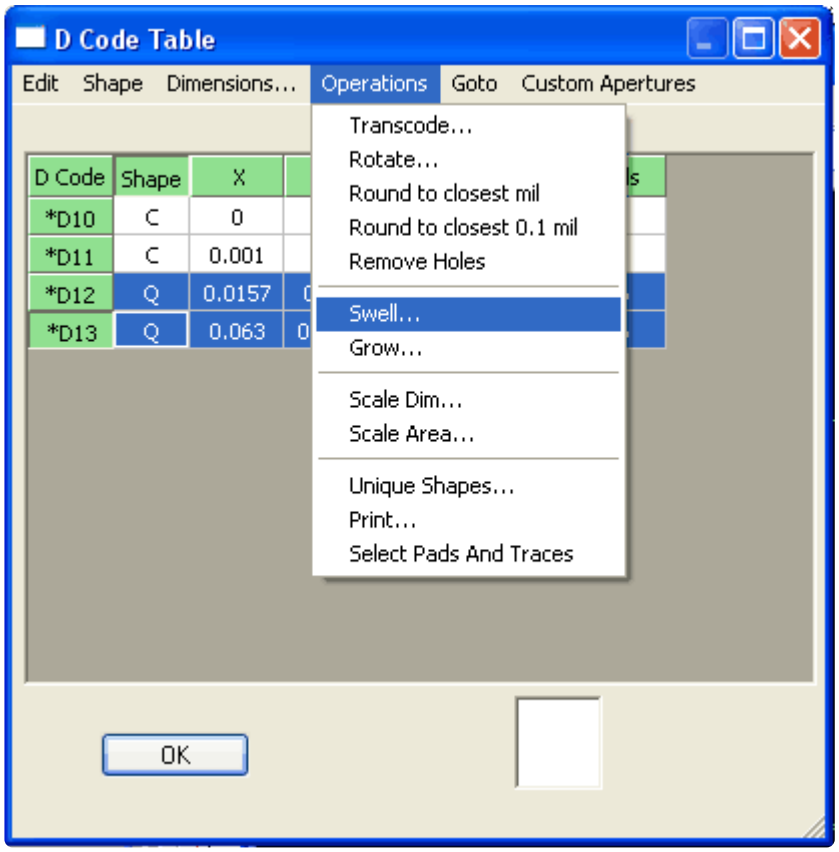
Select Setup>D Codes



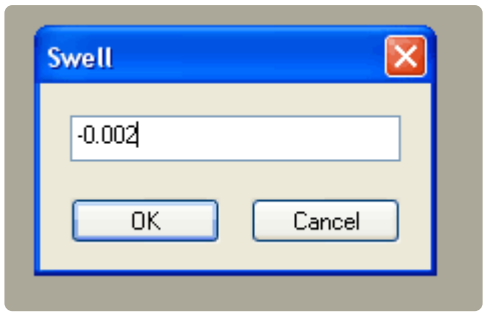
Which will bring up a list of all the pads used. You will probably just want to select all of them.



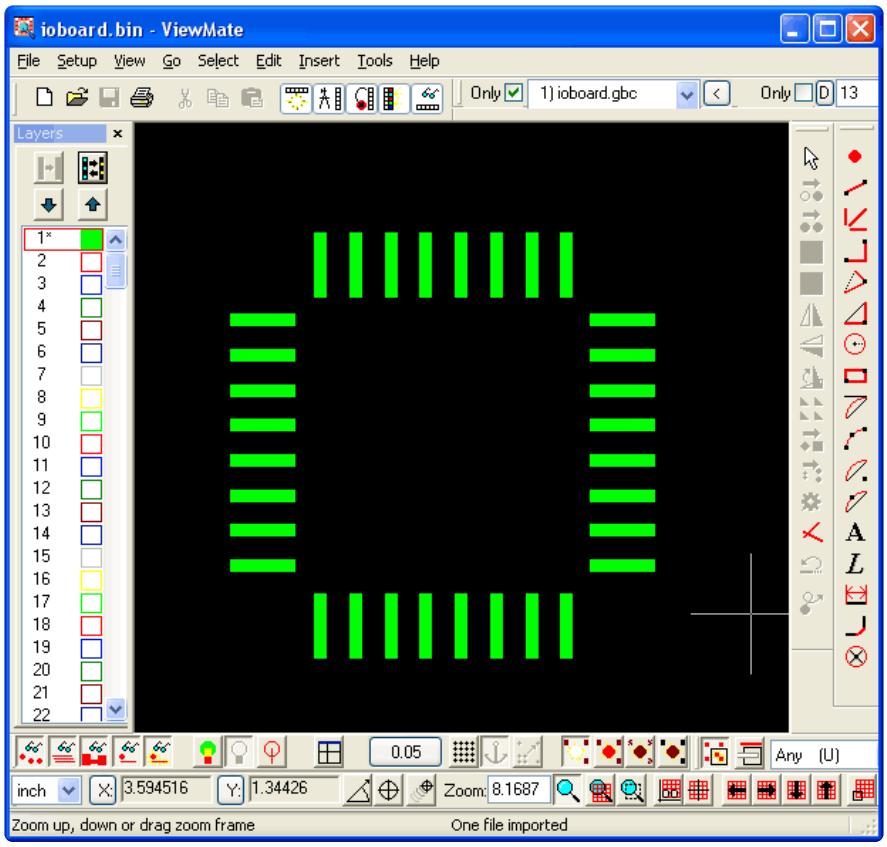
Then select Operations>Swell



and input somewhere around -0.002 (2 mil) to shrink all the pads by 0.002 inches in each direction.

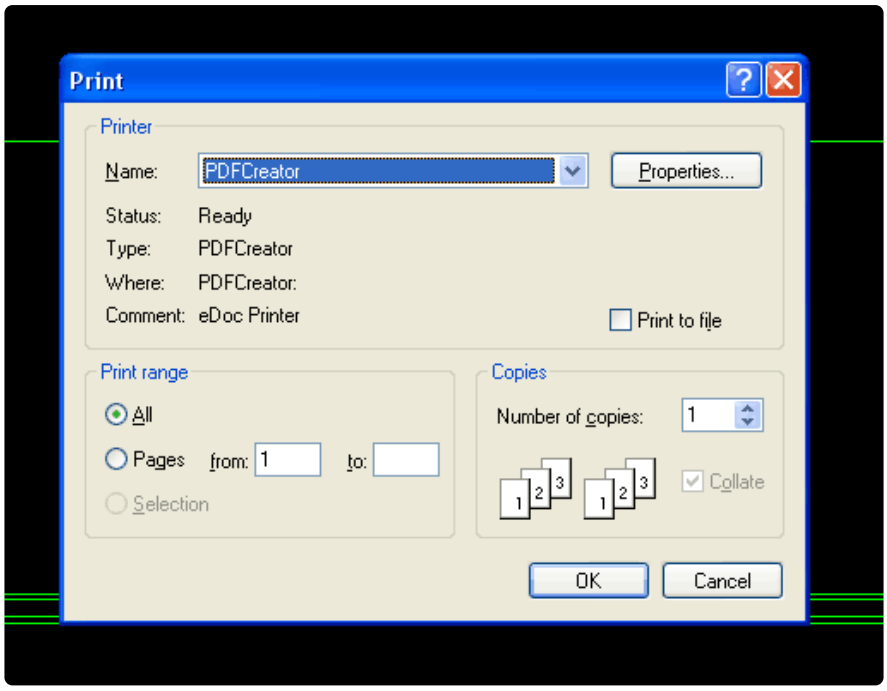


You'll now see that your pads are thinner. This prevents bridging since the laser is not perfectly precise and tends to 'go over' the boundaries by a few mils.



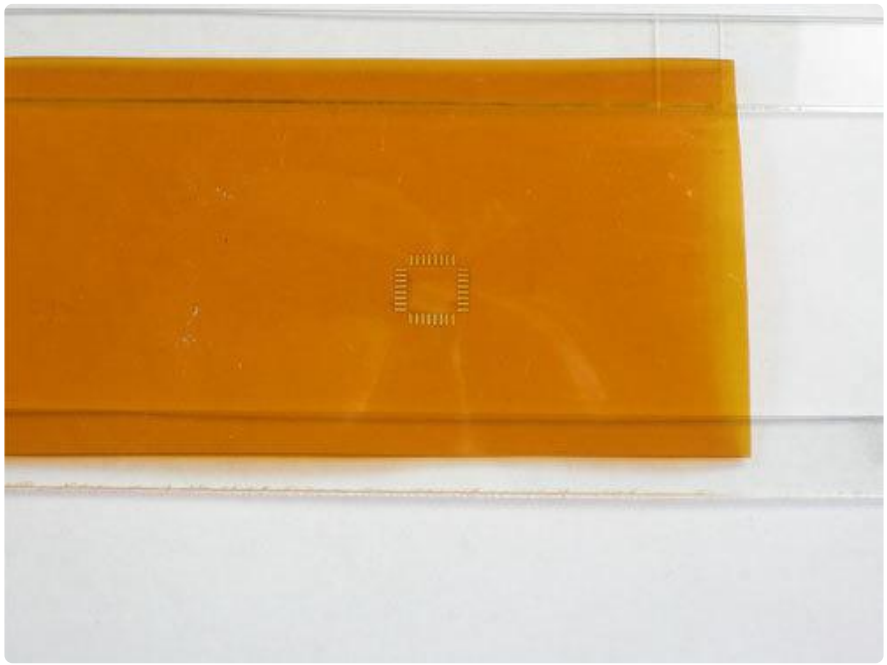
Export

Now we'll export to PDF which will allow for easy importing into Corel Draw. The free version of Viewate doesnt seem to permit exporting, but you can print to PDF which is just as good. You may want to download PDFCreator or a similar print-to-PDF driver if you don't have one installed



Cut!

Import into Corel Draw and use raster not vector, to burn away the kapton film. For a 35W or 45W epilog, 30% speed and 100% power at 600 dpi made for a nice clean edge. Be sure to gently rub the stencil with water and a paper towel to get rid of the burnt kapton.



I usually use the Dimension layer info to make a jig for silkscreening by cutting out the PCB outline in a 0.062" (1/16th) clear acrylic sheet.

Here is a LFCSP 16 (4mm on each side) cut out of 2 mil kapton as above.