

CIRCUIT MAKER

GENERATING GERBER FILES FOR OSH PARK / OSH STENCILS

Patrick Cutno, Miami University

03/18/2016

1 Outline Layer

Select the "Outline" layer

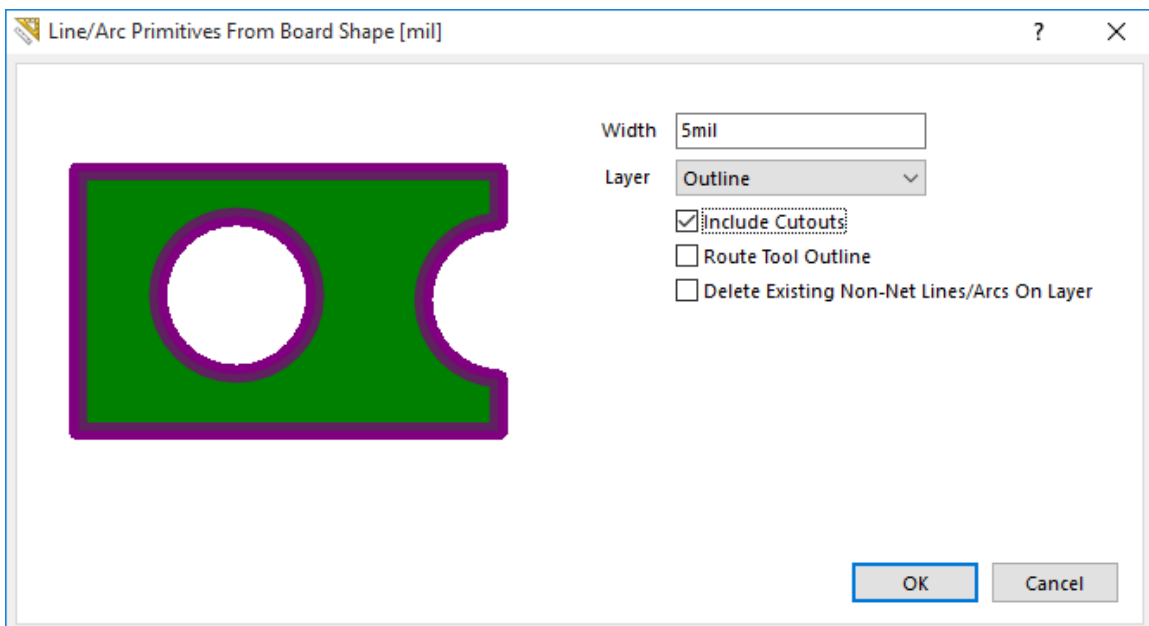


2 Edit Board Size

Run **Home > Board > Board Shape > Edit Board Shape**. Adjust your board dimensions by left clicking and dragging the edges to the desired dimensions. OSH Park requires that the minimum distance between any trace and edge of the board be at least 15 mils. It's recommended to add this to your design rules and run the design rule check after every size adjustment.

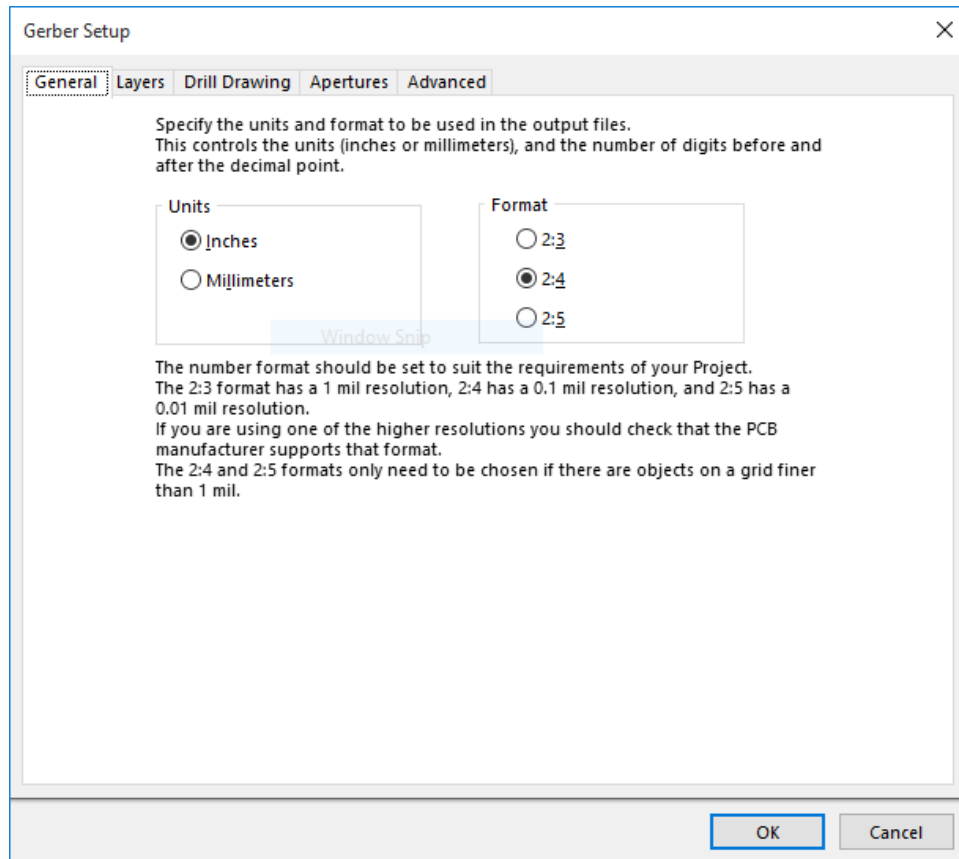
3 Create Board Outline

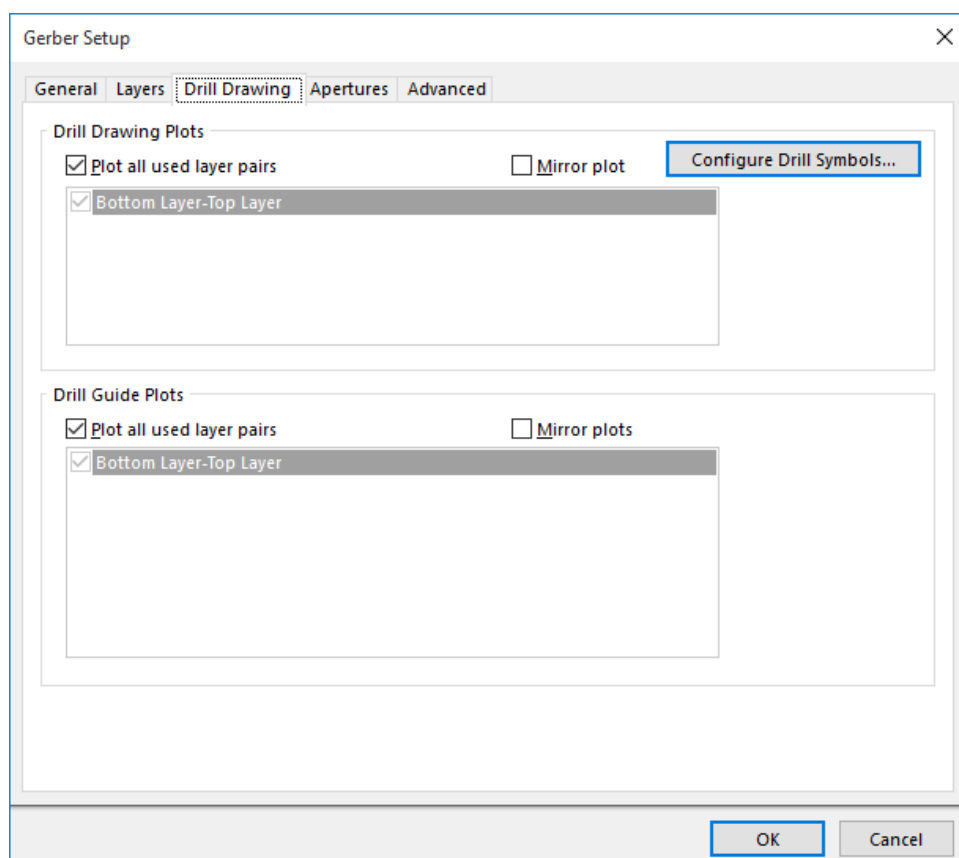
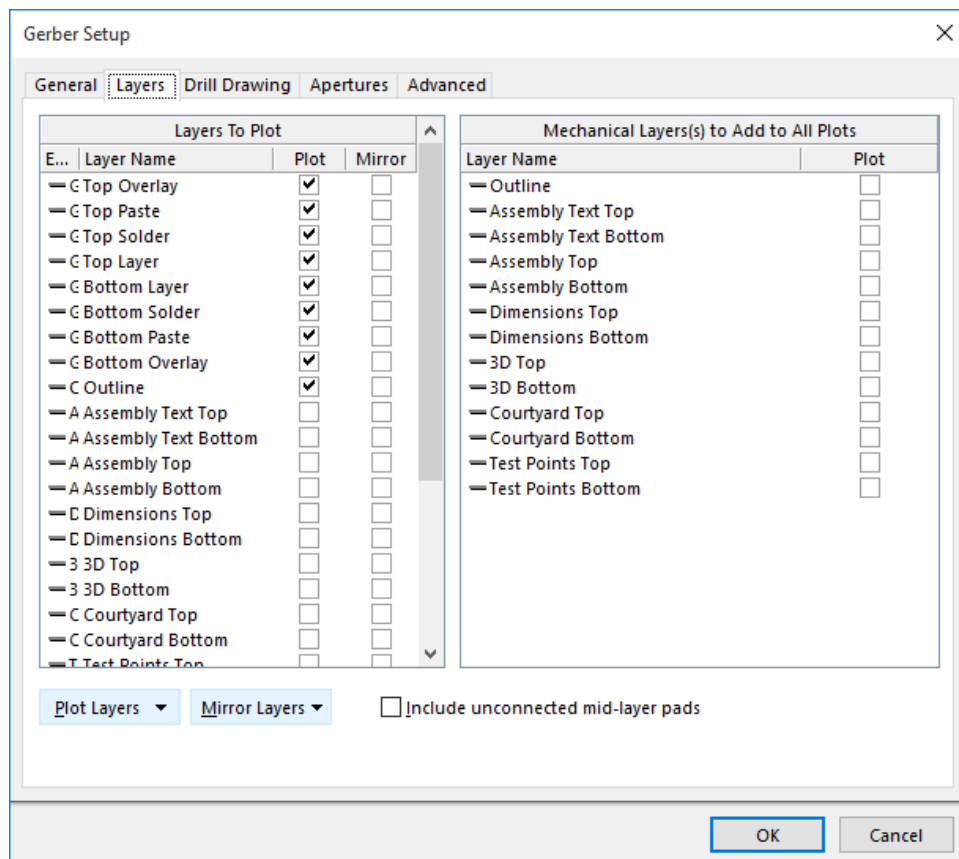
Circuit Maker by default doesn't draw the board outline. Without this step, OSH Park will throw an error as they won't know the dimensions of the board. Run **Home > Board > Board Shape > Create Primitives From Board Shape** and use the following settings:

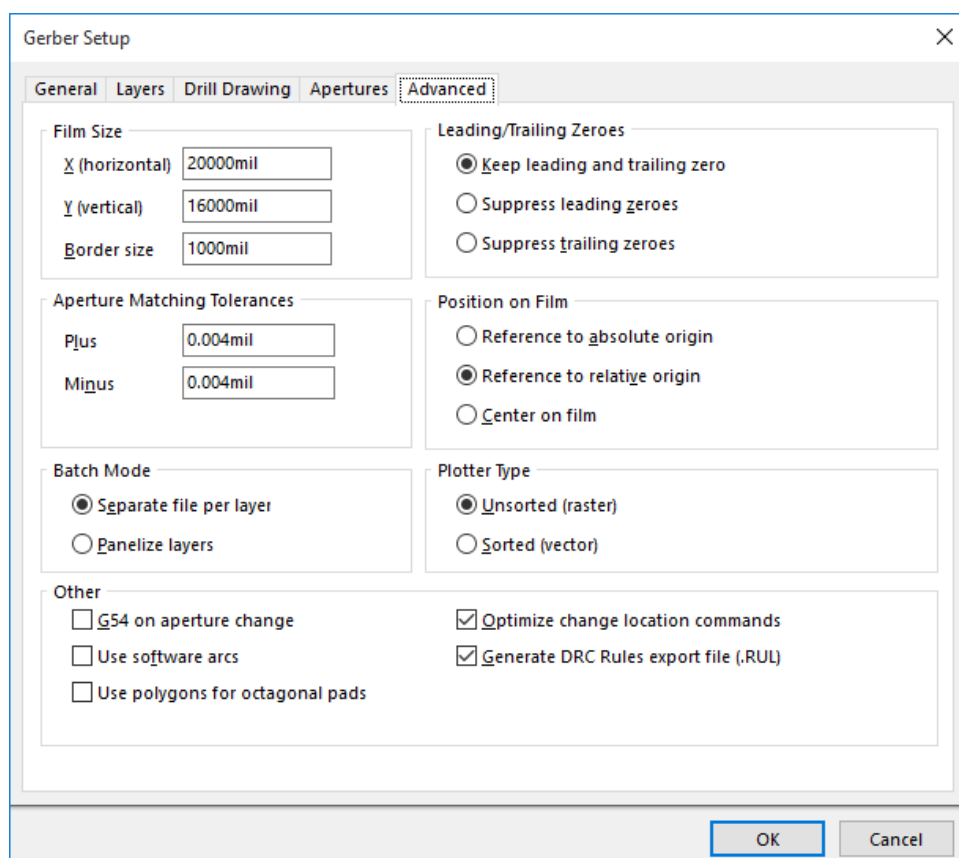
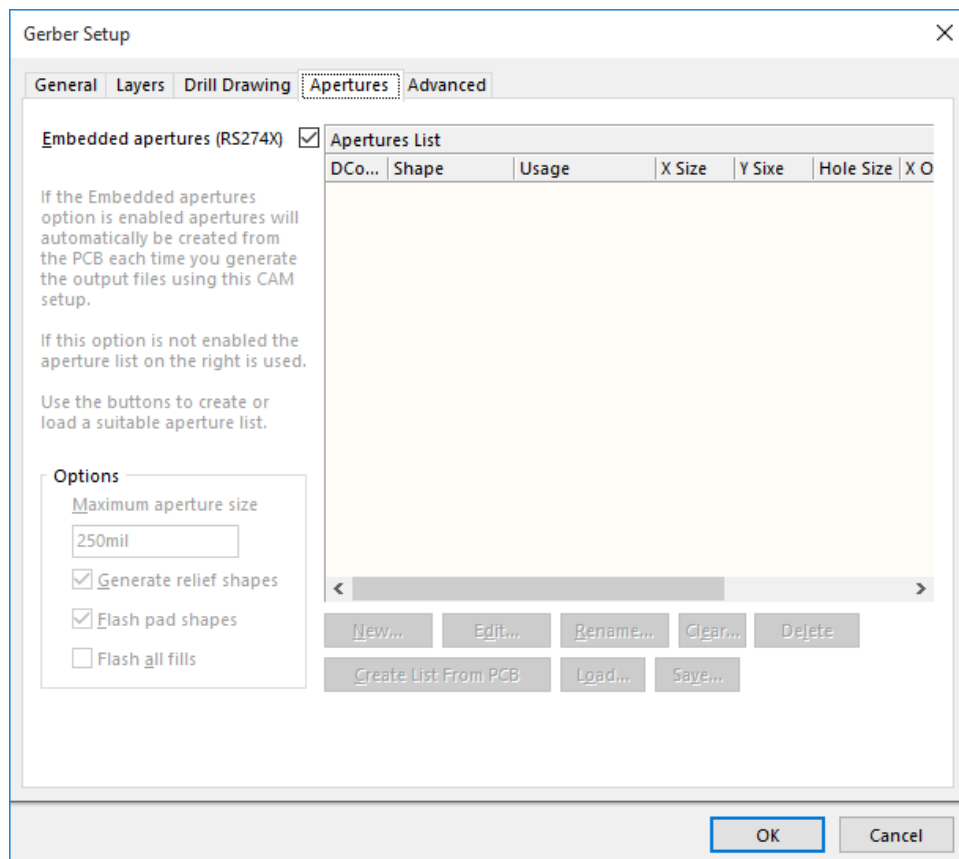


4 Generating Gerber Files

Run **Outputs > Gerber**. You may be asked to save your progress, go ahead and do so. Then use the following settings:







5 Generating Drill Data

Run **Outputs > NC Drill Files**. You may be asked to save again. Use the following settings:

The image shows the 'NC Drill Setup' dialog box with the 'Options' tab selected. The 'NC Drill Format' section contains instructions and two sub-sections: 'Units' and 'Format'. The 'Units' section has radio buttons for 'Inches' (selected) and 'Millimeters'. The 'Format' section has radio buttons for '2:3', '2:4' (selected), and '2:5'. Below these is a paragraph explaining the resolution of each format. The 'Leading/Trailing Zeros' section has radio buttons for 'Keep leading and trailing zeroes' (selected), 'Suppress leading zeroes', and 'Suppress trailing zeroes'. The 'Coordinate Positions' section has radio buttons for 'Reference to absolute origin' and 'Reference to relative origin' (selected). The 'Other' section has checkboxes for 'Optimize change location commands' (checked), 'Generate separate NC Drill files for plated & non-plated holes', 'Use drilled slot command (G85)', 'Generate Board Edge Rout Paths', and 'Generate EIA Binary Drill File (.DRL)'. There is also a text field for 'Rout Tool Dia' set to '200mil'. At the bottom are 'OK' and 'Cancel' buttons.

NC Drill Setup

Options

NC Drill Format

Specify the units and format to be used in the NC Drill output files.

This controls the units (inches or millimeters), and the number of digits before and after the decimal point.

Units

☒ Inches

☐ Millimeters

Format

☐ 2:3

☒ 2:4

☐ 2:5

The number format should be set to suit the requirements of your design. The 2:3 format has a 1 mil resolution, 2:4 has a 0.1 mil resolution, and 2:5 has a 0.01 mil resolution. If you are using one of the higher resolutions you should check that the PCB manufacturer supports that format. The 2:4 and 2:5 formats only need to be chosen if there are holes on a grid finer than 1 mil.

Leading/Trailing Zeros

☒ Keep leading and trailing zeroes

☐ Suppress leading zeroes

☐ Suppress trailing zeroes

Coordinate Positions

☐ Reference to absolute origin

☒ Reference to relative origin

Other

☒ Optimize change location commands

☐ Generate separate NC Drill files for plated & non-plated holes

☐ Use drilled slot command (G85)

☐ Generate Board Edge Rout Paths

Rout Tool Dia

☐ Generate EIA Binary Drill File (.DRL)

OK **Cancel**

6 Extracting The Gerber Files

Using 7-Zip or another program of your choosing, extract the generated Gerber files into a folder called "Gerber".

7 Rename Outline

In the Gerber folder, locate the file PROJECT_NAME.Outline and rename the file to PROJECT_NAME.GKO. Where PROJECT_NAME is the name of your project. OSH Park will look for a .GKO file to contain the outline of the board but Circuit Maker saves this information as a .Outline file, this step is very important.

8 Extracting The Drill Data

Just like the Gerber files, extract the drill files into a folder called "Drill".

9 Adding Drill Data To Gerber

Copy PROJECT_NAME.TXT from the Drill folder into the Gerber folder. Where PROJECT_NAME is the name of your project. This text file contains the locations for all of holes that need to drilled in the PCB, this step is also very important.

10 Creating The Final Zip

Open the Gerber folder, highlight all files in the folder and create a new Zip file. This new zip file is the final end product that you will submit to OSH Park or OSH stencils!

If your project name is PCB, the final contents of your Zip file should be as follows:

