

Eagle Board Fabrication Tutorial

Introduction

In the two previous Eagle tutorials, we covered the basic steps of creating a schematic and board layout for a simple circuit in the Eagle Schematic Capture and Board Layout program. In this tutorial, we use Eagle's CAM processor to create board description files, called Gerber files, describing each layer of the board for manufacturing.

Discussion Overview

Gerber files contain location and size information for all the features on a layer. In order for a board fabrication house to manufacture a PCB, they need detailed information for each of the features of the design. For example, for the top layer, the fabrication house will need to have the exact location and size information for each component pad, and in the case of through hole parts, the drill size for each component pad holes. They also need the information about the size and location of the copper traces on this layer as well as the silk screen layer information about the size, orientation and location of components' outlines, names, etc.

Eagle uses what's called a CAM processor to create the Gerber files for all the necessary layers for board fabrication. The next section will walk you through this process step by step.

Procedure

Start off by loading the board view of your design. Figure 1 shows the layout of the design from our previous tutorial.

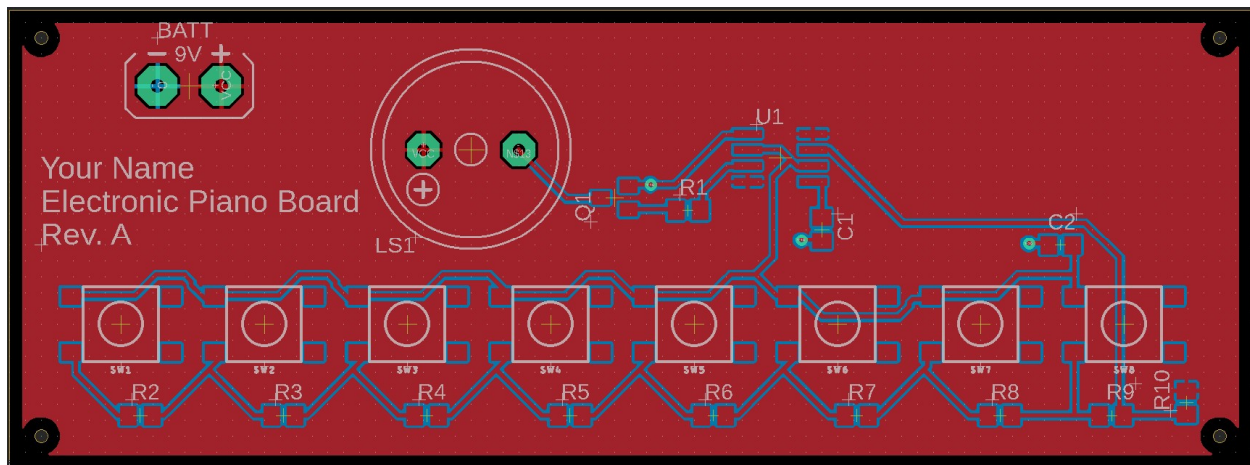




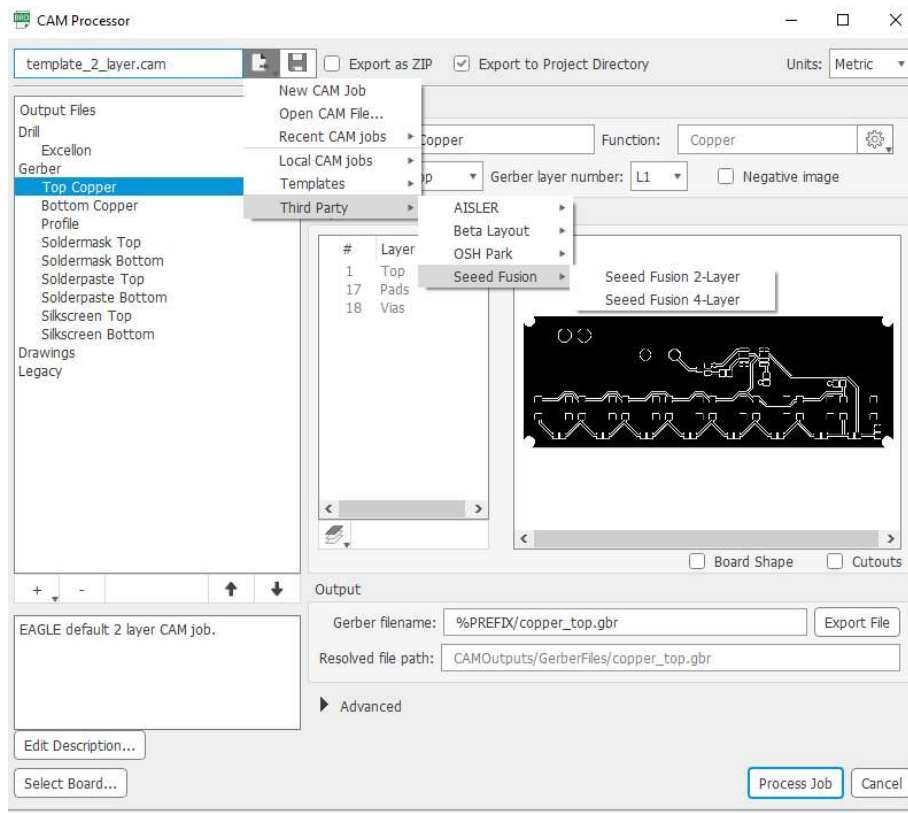
Figure 1 – DIY Piano Board Layout

In order to load your own design, start by cloning and updating your EE-Workshop repo. **Note:** *If you're design is already open, you can skip this step.*

1. Clone your EE-Workshop repo to your desktop.
2. If necessary, update your repo by executing the following commands in a command window from the EE-Workshop folder created on your desktop
 - a. git fetch upstream
 - b. git pull upstream master

Note: Above layout is available as a .png image file for your reference in the folder EE-Workshop/Lesson 4/Eagle/DIYpiano folder.

3. Once you have opened the board view of your design, click on the “Ratsnest” icon  to update the metal pours in the design. Your design should look similar to the one shown in Figure 1.
4. Click on the “CAM Processor” icon  to open the “CAM Processor” dialogue window.
5. Click the document icon next to the CAM file name and go to **Third Party -> Seeed Fusion -> Seeed Fusion 2-Layer**



This will load the Gerber template file used by “Seeed Studios Fusion PCB” website.

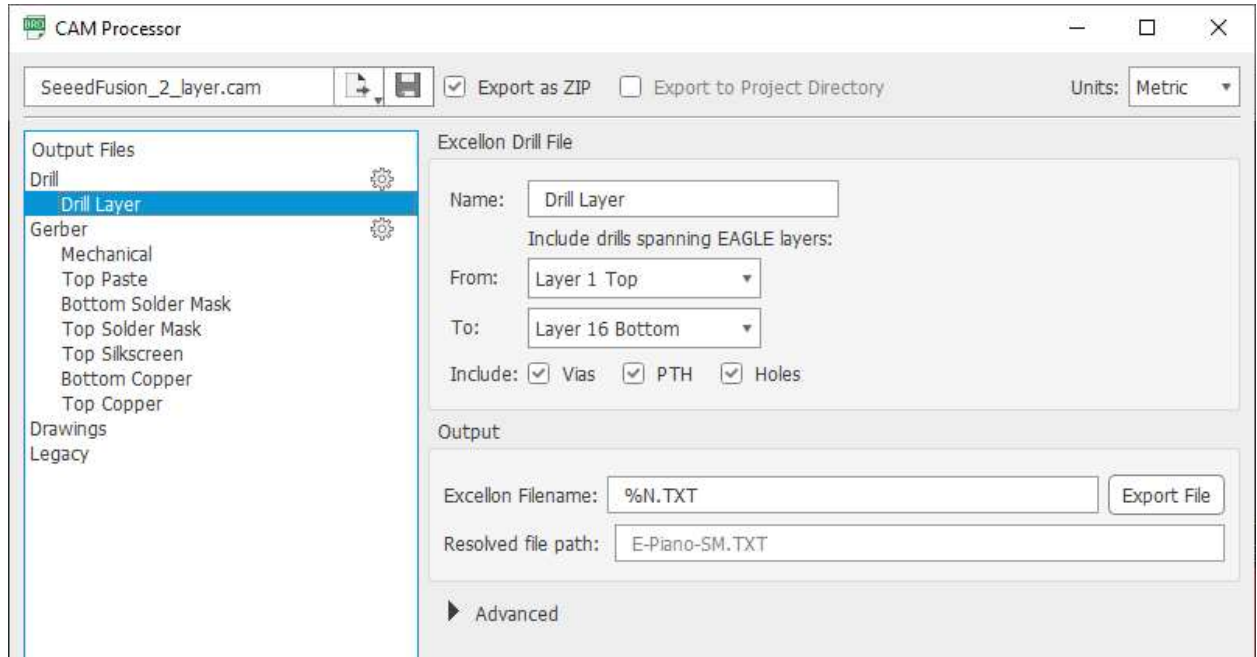
The Gerber template defines the drill and Gerber files needed by Seeed Studios Fusion

PCB manufacturing process to build our board.

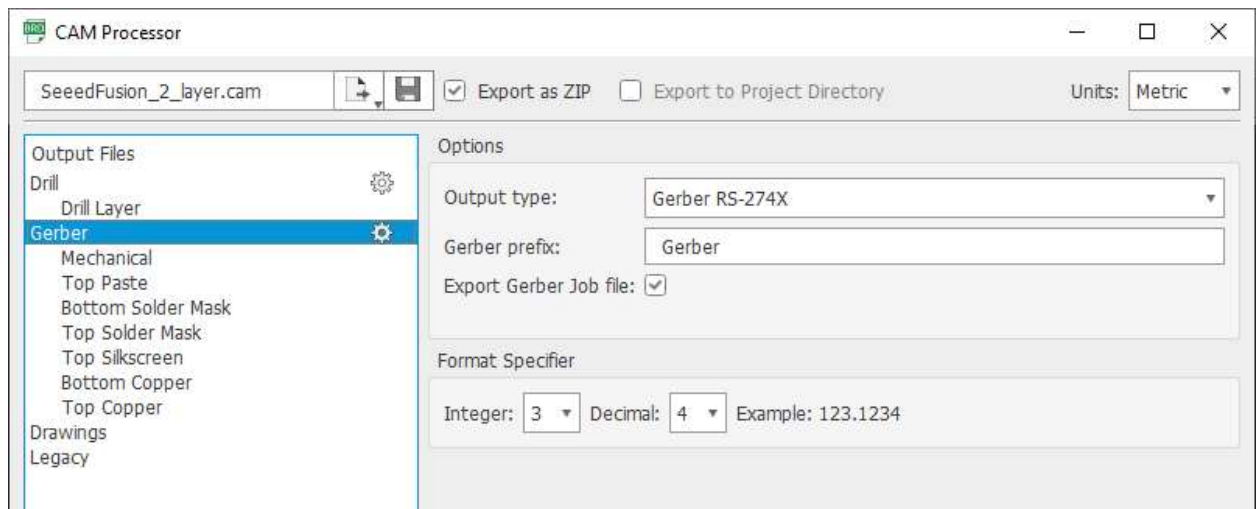
Each Gerber file in the “Output Files” window on the left defines a number of layers that will be included in that file. In the “Layers” sub-window on the right, the layer information included in the selected file are given. For example, the “Top Copper” Gerber file includes the Top (1), Pads (17) and Vias (18) layers.

6. Click on each layer and examine the information for that layer in the “Layers” sub-window on the right. You will notice that there are Gerber files whose layers do not contain any information. The “Bottom Paste” Gerber file, for example, would normally include the outlines for surface mount pads placed on the bottom layer. However, since we do not have any parts placed on the bottom layer, this Gerber file looks empty. This is also the case for the “Bottom Silkscreen” Gerber file. We will remove these files from our list.
7. Right click on the “Bottom Paste” Gerber and select “Delete”. Do the same for the “Bottom Silkscreen”.
8. When done, you should have the following Gerber files:
 - a. Mechanical: Contains the board shape, dimensions and cutouts. The layers included in this Gerber file are “Dimension”, “Milling”, “Board Shape” and “Cutouts”.
 - b. Top Paste: Contains the information about the surface-mount pads on the top layer of your design. The layer included in this Gerber file is “tCream”.
 - c. Bottom Solder Mask: Contains the openings in the solder mask for the bottom of the board where metal pads are exposed for soldering. Solder mask is the green material that covers the entire board that protects the metal layer underneath. The layer included in this Gerber file is “bStop”.
 - d. Top Solder Mask: Contains the openings in the solder mask for the top of the board where metal pads are exposed for soldering. The layer included in this Gerber file is “tStop”.
 - e. Top Silkscreen: Contains component outline and names to be printed on the top of the board. The layers included in this Gerber file are “tPlace” and “tNames”.
 - f. Bottom Copper: Contains the information about the copper traces, pads and vias on the bottom layer of your design. As you can see, this Gerber, for example, contains the copper information for the ground metal fill. The layers included in this Gerber file are “Bottom”, “Pads” and “Vias”.
 - g. Top Copper: Contains the information about the copper traces, pads and vias on the top layer of your design. The layers included in this Gerber file are “Top”, “Pads” and “Vias”.

9. In addition to the above Gerber files, you can also find a “Drill” file. This file contains the information for the size and location of all the holes that need to be drilled in the board. Click on “Drill Layer” under the “Drill” file. The settings for this drill file should look as shown below.



10. Click on the “Gerber” setting icon itself, and you can see a number of settings for the collection of the Gerber files as shown below.



11. Make sure “Export as ZIP” is checked, and then click on “Process Job”. Select your “E-Piano” folder for the destination of the Gerber ZIP file and click “Save”. This will create a .zip file where all the Gerber files will be placed in.

12. Ordering boards

Next step in our manufacturing process is to upload our Gerber files to a PCB fabrication online service to check the files and order the boards. We will use seeed studio as our board fabrication house.

a. Load the following URL in your favorite browser:

https://www.seeedstudio.io/fusion_pcb.html

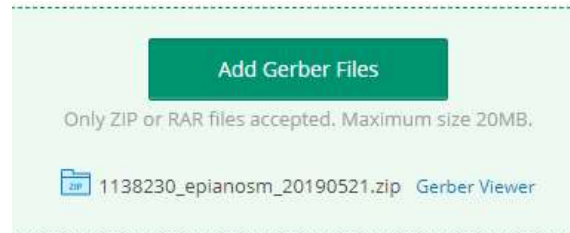
b. On this site, you can see a number of settings for the board you’d like to fabricate.

The screenshot shows the Seeed Studio PCB configuration interface. The main area contains various settings for a PCB, including Base Material (FR-4 TG130), No. of Layers (2 layers), PCB Dimensions (100 x 100 mm), PCB Quantity (10), No. of Different Designs (1), PCB Thickness (1.60 mm), PCB Color (Green), Surface Finish (HASL), Minimum Solder Mask Dam (0.4mm), Copper Weight (1oz), Minimum Drill Hole Size (0.3mm), Trace Width / Spacing (6/6 mil), Blind or Buried Vias (No), Plated Half-holes / Castellated Holes (No), and Impedance Control (No). A sidebar on the right displays the PCB Cost (USD\$4.90) and a Sub-Total (USD\$4.90). An 'Add to Cart' button is visible at the bottom right.

On the right, you will see a cost estimate for the boards with the

configuration/settings selected on the left.

- c. On Sseed's PCB site, click on "Add Gerber Files" and upload your .zip file.
- d. Once the file has been uploaded, click on the "Gerber Viewer" to examine the board layers.



- e. In the Gerber viewer, check to see if there are any "Error Messages". If there are any errors in the upload process, they will show up here.

You can also check each layer uploaded by turning on the layer from the list of available layers on the left. For example, in the image below, all the layers for the top of the board have been turned on.

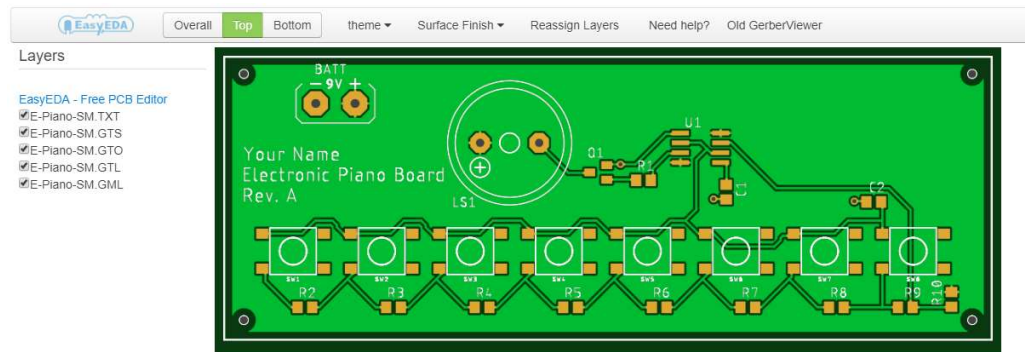


Figure 2 - Uploaded Gerber File Viewer

- f. Return back to Sseed's PCB page where you had uploaded the files. You can see that the PCB dimensions and number of layers have already been filled in based on the Gerber files you just uploaded. Set the remaining options as follows.
 - i. Base Material: FR-4 TG130
 - ii. No. of Layers: 2 Layers
 - iii. PCB Quantity: 5
 - iv. No. of Different Designs: 1
 - v. PCB Thickness: 1.60mm

- vi. PCB Color: Select your desired color
- vii. Surface Finish: HASL
- viii. Minimum Solder Mask Dam: 0.4mm
- ix. Copper Weight: 1oz.
- x. Minimum Drill Hole Size: 0.3mm
- xi. Trace Width/Spacing: 6/6mil
- xii. Blind or Buried Vias: No
- xiii. Plated Half-holes/Castellated Holes: No
- xiv. Impedance Control: No

g. Next click on “SMT Stencil” to order a stencil for your board.

A stencil is a thin film with openings for the surface-mount parts’ pads. A stencil is used to place solder paste on the pads on the board.

- i. Click on “Add Gerber Files” and upload the same .zip file that you used for ordering your PCB.
- ii. Set the remaining options as follows.
 - 1. PCB Dimensions:..... “10.0cm * 15.0cm Frameless...”
 - 2. Stencil Quantity..... 1
 - 3. Fiducial Mark “No Fiducial”
 - 4. Thickness 0.12
 - 5. Polishing Technique..... “Polished”

h. Click on “Add to Cart”.

- i. Click on the cart icon  to proceed to checkout.

Appendix – Useful Links

Schematic capture in Eagle:

<https://www.youtube.com/watch?v=1AXwjZoyNno>

Laying out a board in Eagle:

<https://www.youtube.com/watch?v=CCTs0mNXY24>