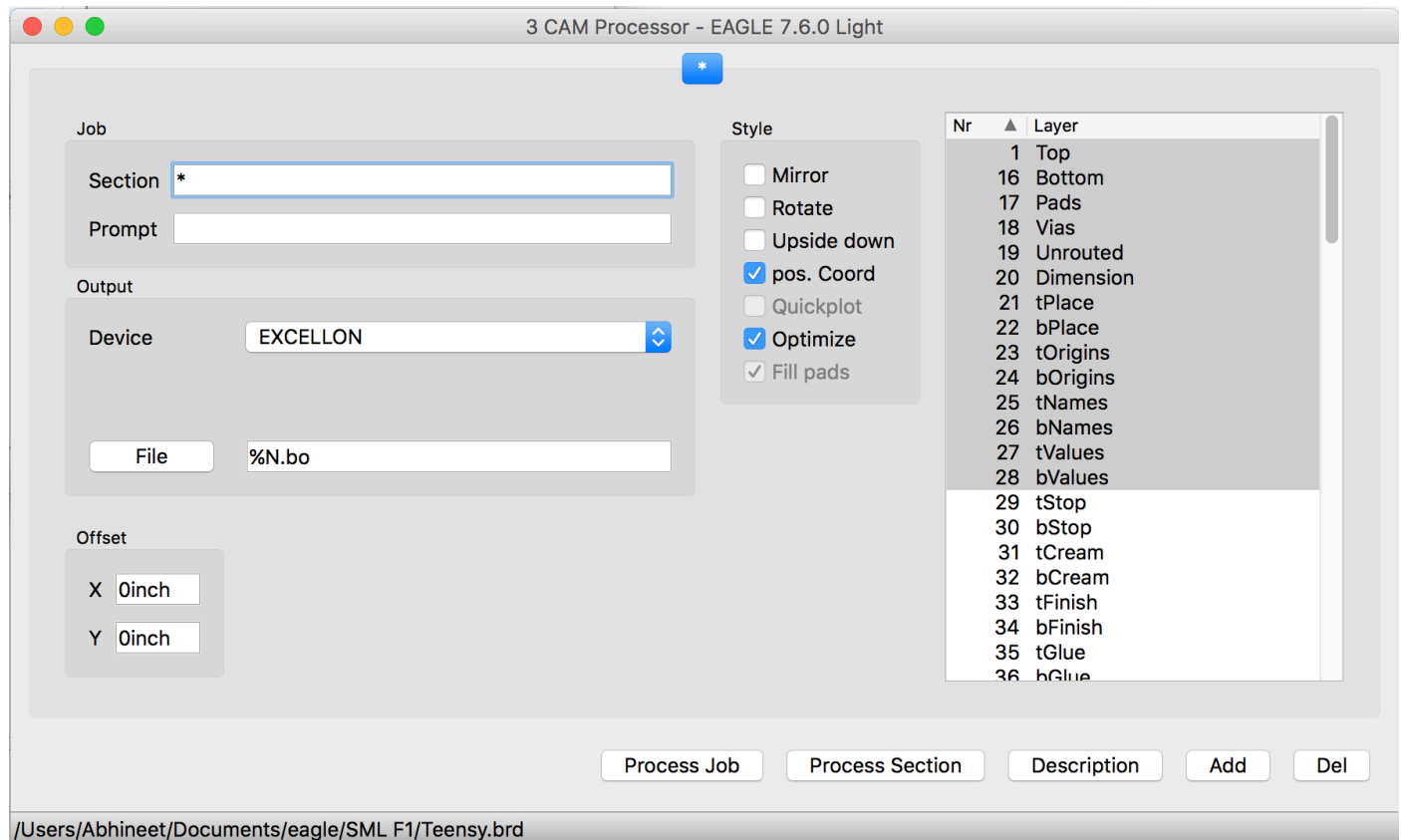


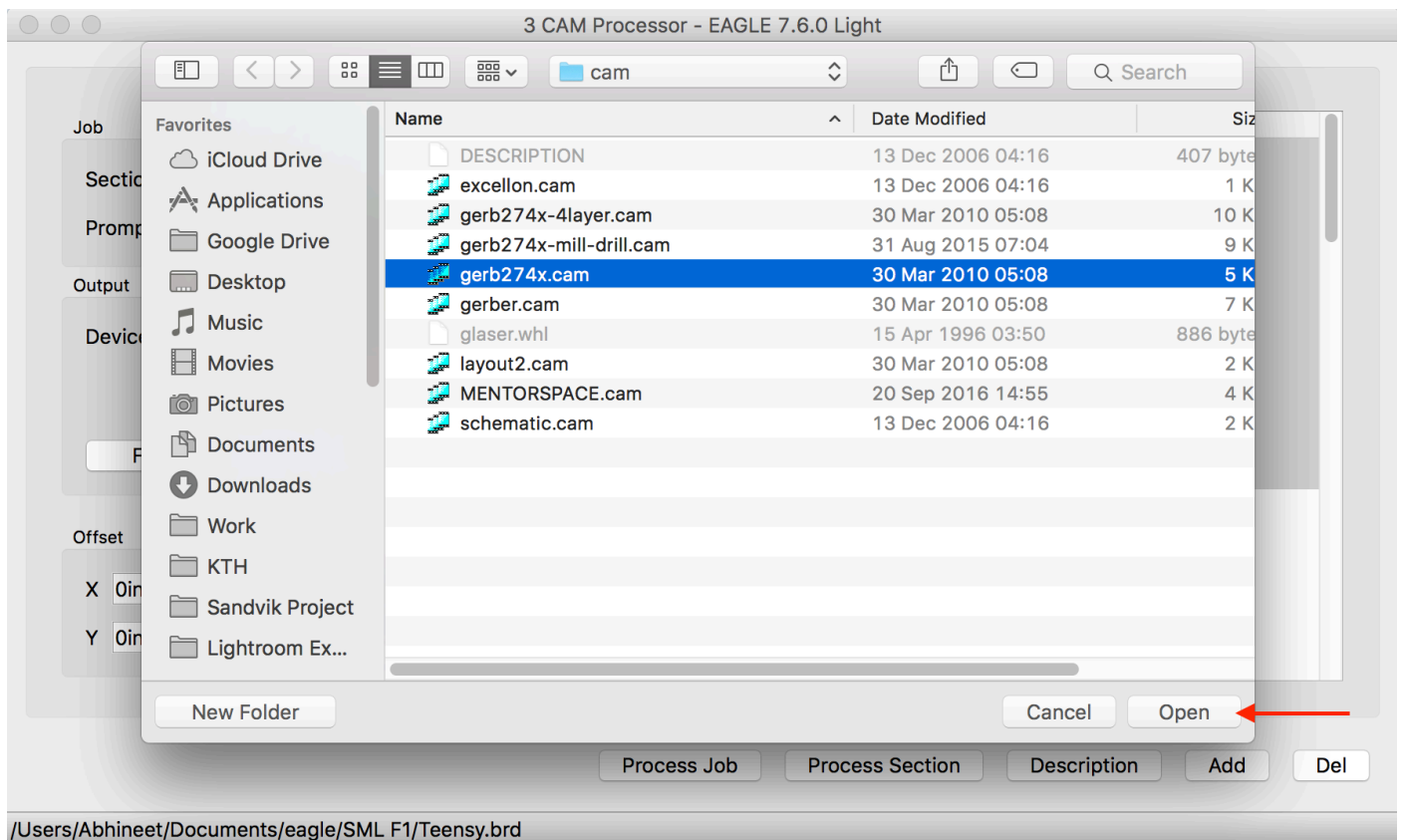
Eagle CAD Tool Tutorial

This is a short tutorial to generate Gerber files in **Eagle CAD** for PCB prototyping on LPKF Protomat E33 machine.

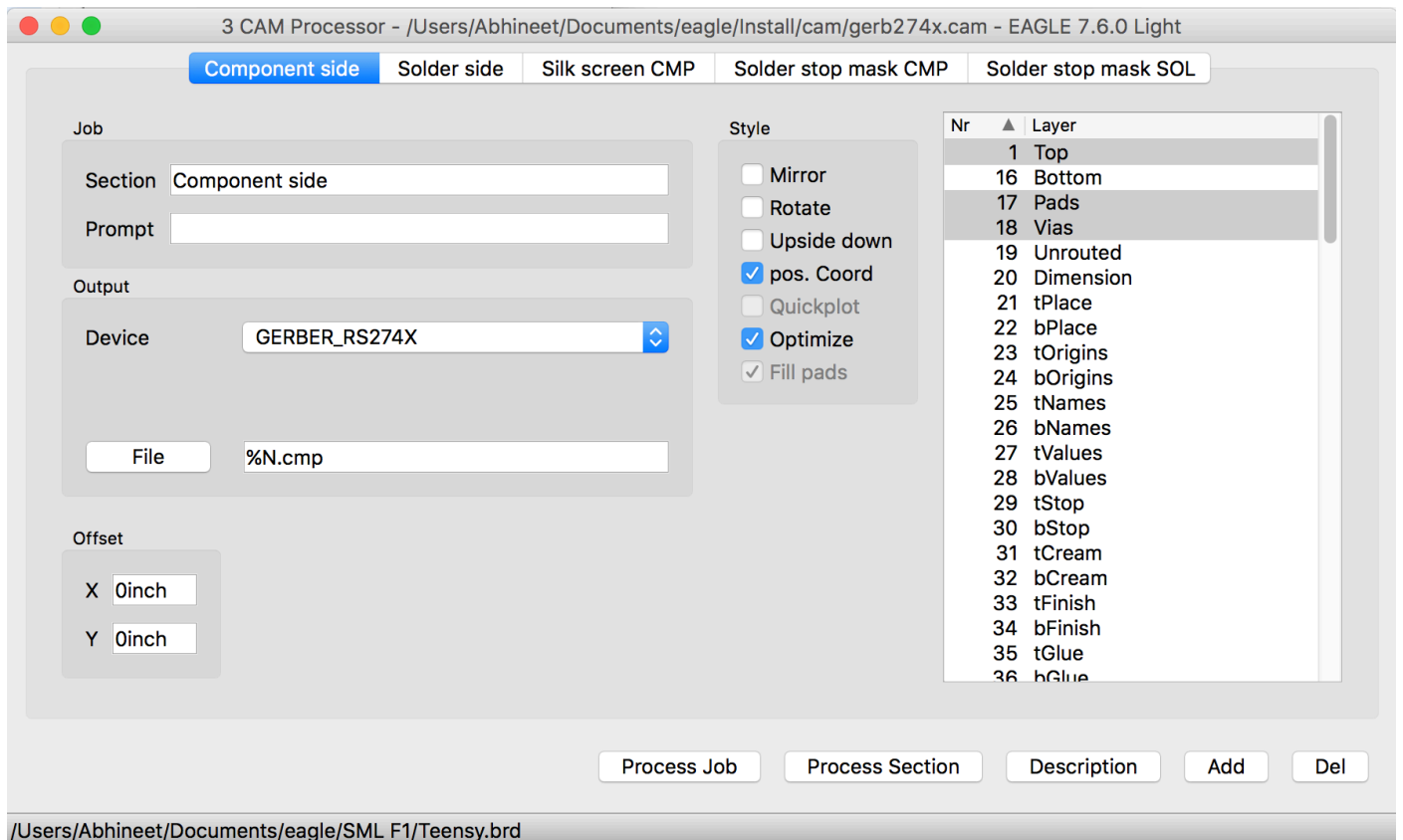
After you have completed working on the PCB Layout, Open the CAM tool.



This is the window that should pop up first, now open one of the given jobs, open **gerb274x.cam**



The CAM processor should open up with the following layers:

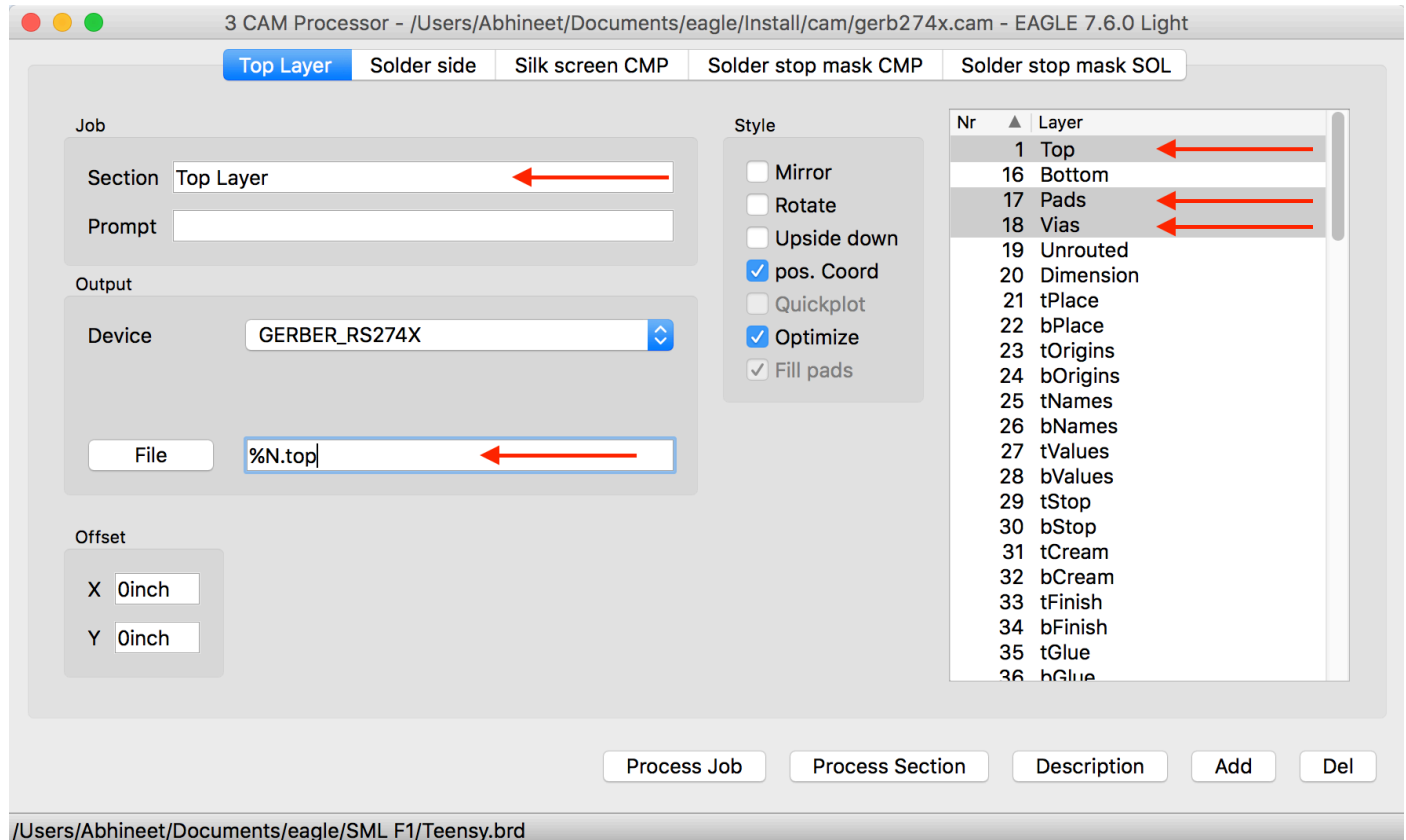


Now we need to modify this cam file to generate just four layers: **Top, Bottom, Outline & Drills**

Step 1. Top Layer

Modify the first layer named *Component Side* with the following settings

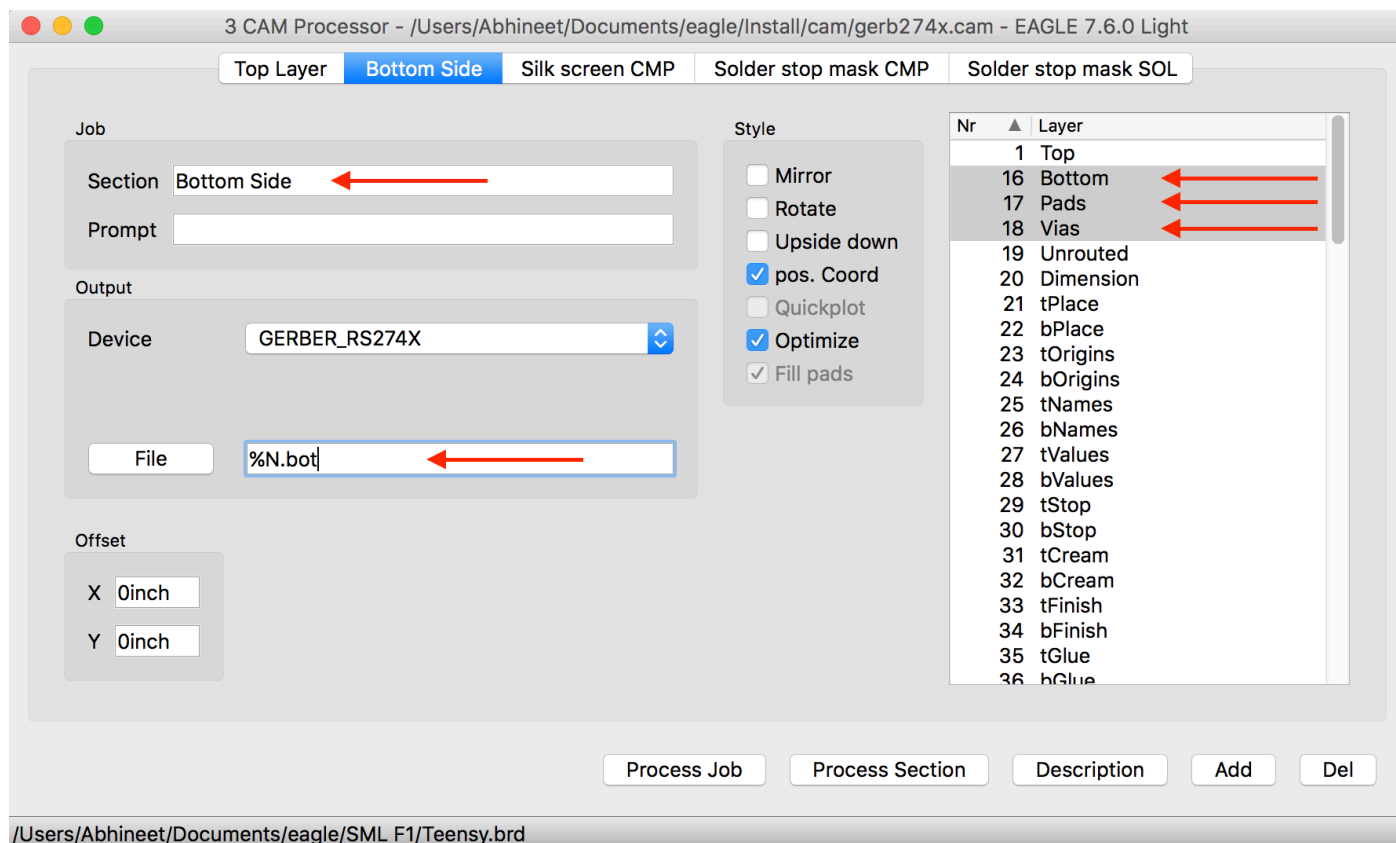
1. Section – Top Layer
2. Device – GERBER_RS274X
3. File – %N.top
4. Layers – Top, Pads, Vias



Step 2. Bottom Layer

Modify the second layer named *Solder Side* with the following settings

1. Section – Bottom Layer
2. Device – GERBER_RS274X
3. File – %N.bot
4. Layers – Bottom, Pads, Vias

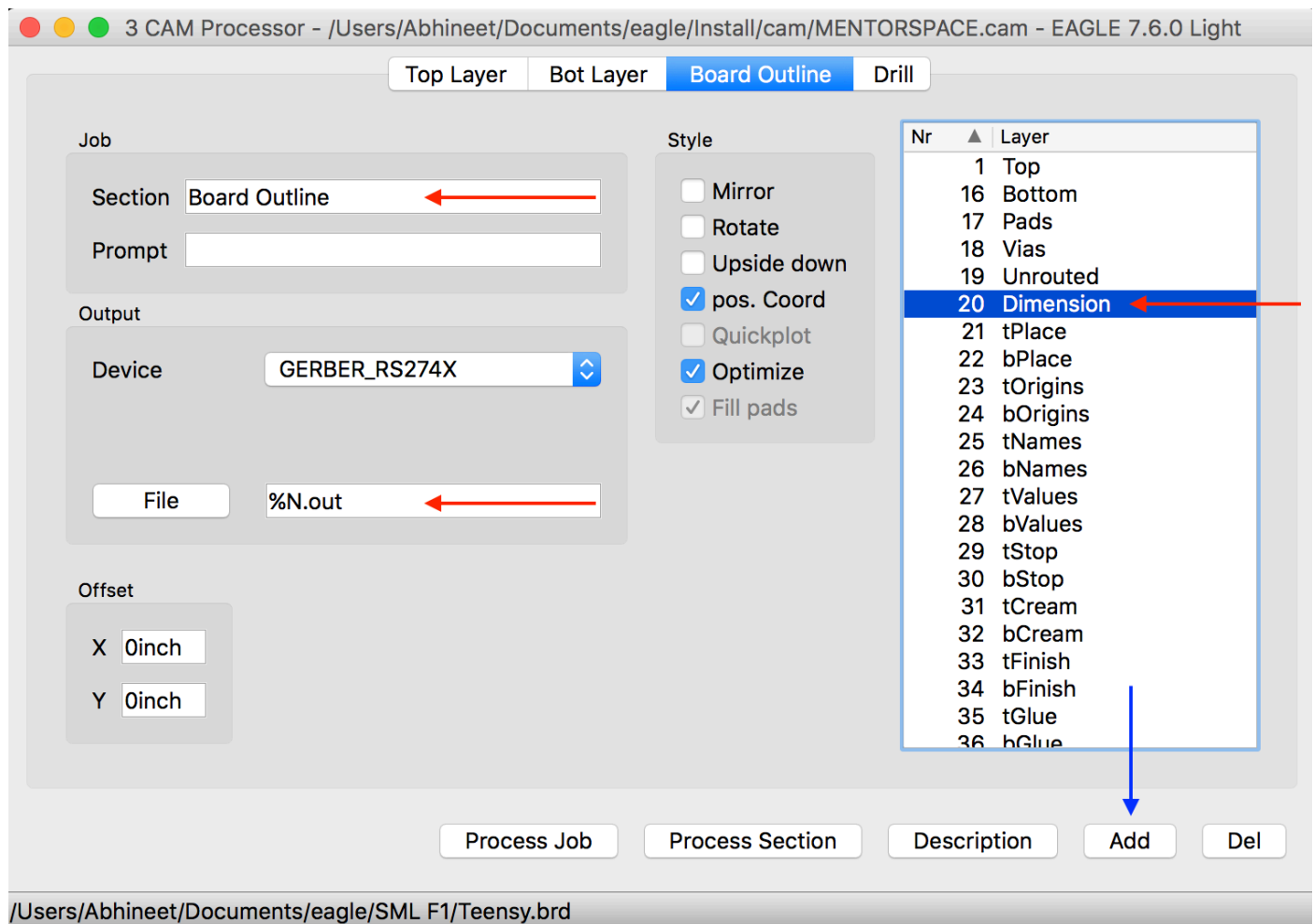


Now delete all the remaining layers, using the **Del** button and add another layer, by using the **Add** button.

Step 3. Outline Layer [The PCB Cutting boundary]

Modify this new layer with the following settings

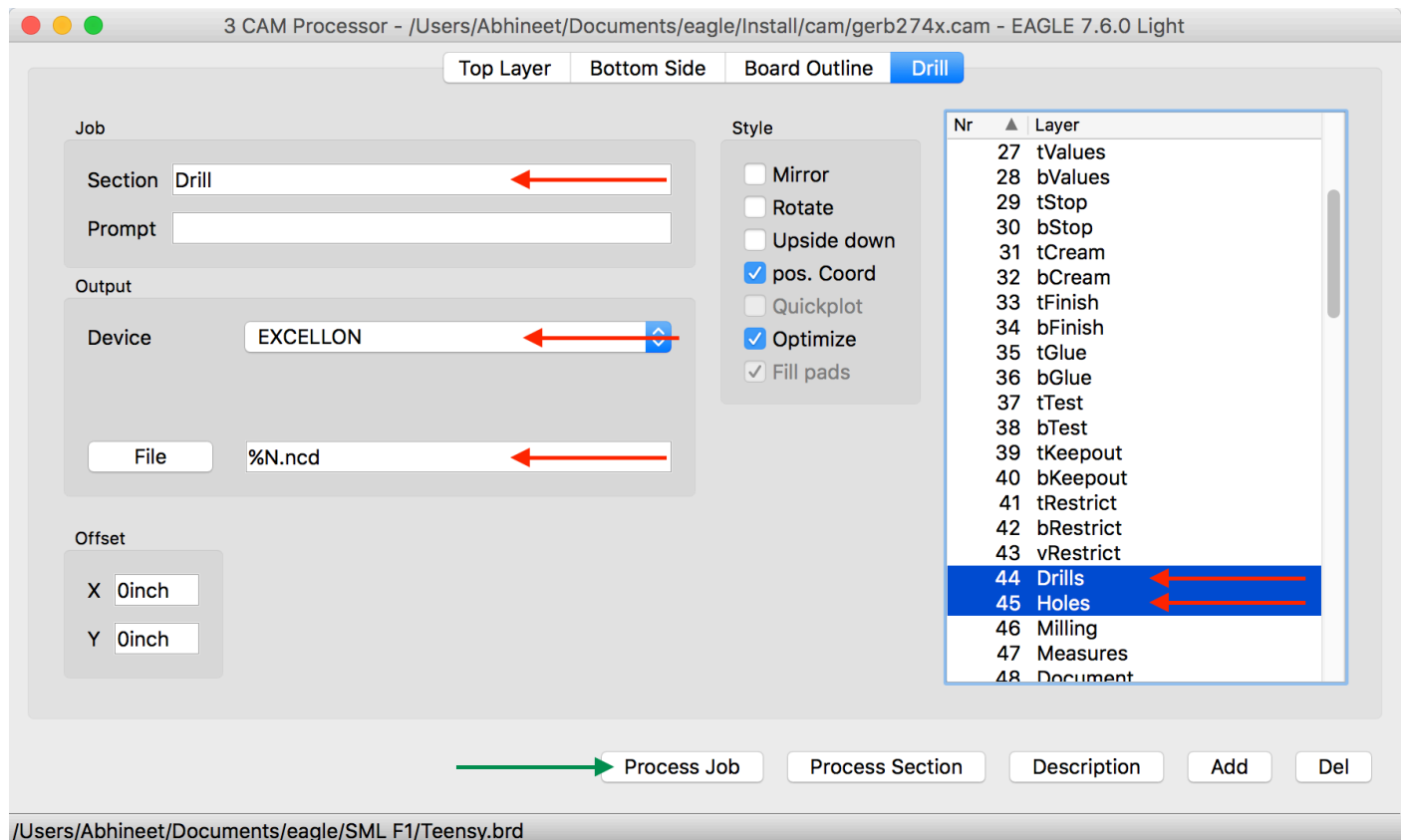
1. Section – Board Outline
2. Device – GERBER_RS274X
3. File – %N.out
4. Layers – Dimension



Step 4. Drill Layer

Modify the new layer with the following settings

1. Section – Drill
2. Device – EXCELLON
3. File – %N.ncd
4. Layers – Drills, Holes



Once you have all these four layers, click on **Process Job**.

Four new files will be created in the same folder as the eagle design files. Now you can transfer this file to the PC connected to the milling machine to proceed with the next steps of PCB prototyping.

You can also save this job so that there is no need to repeat these steps all over again for every PCB you build