Creating Gerber PWB Fabrication files with EAGLE V4.X

Prepared by Fred Heep, SFU Engineering Science, 2004 November (updated 2006 June)
*** Based in part on work created by Seiichi Inoue, www.interq.or.jp

In order to fabricate an actual printed wiring board (PWB) based on the ECAD design work you've done in EAGLE, you first need to convert EAGLE's native data format to the industry standard Gerber form.

The Gerber file structure (originally created by Gerber Scientific Instrument Company) consists of a succession of numerical expressions specifically intended to communicate your PWB layout details (such as size and position of holes, thickness of a line/traces, etc.), to the fab shop's photoplotter.

While there are a few companies worldwide that will accept your basic EAGLE (or any other competitive ECAD product) design files (*.brd), the vast majority won't be able to do anything with it. It is the designer's responsibility to prepare and confirm each of the production files required. Additionally, you will need to specify all of the fabrication parameters (tolerances, material types, copper thicknesses, finish styles, etc.) for each project.

EAGLE can convert and generate many of these files for you. To do this, please follow the sequence of steps outlined herein.

🔴 Prepare a folder for your production files

Although the creation of a production folder is not a requirement, it is from experience that I recommend this step. By doing this, you will have an isolated storage area strictly for the final completed work on the design revision level you are about to release. Once the board is fabricated – like it or not, this revision should be considered 'cast in stone', and the files in this folder should archived read-only for future reference.

Once you've created the Production folder, copy only your final *.sch and *.brd files for this revision from your work files folder into the production folder. Change the properties on these 2 files to Read-Only. This will help serve as a reminder later if you try to fix bugs on these files with EAGLE's editors. Always edit on files in your work files folder.

Note: In EAGLE's Control Panel window, you can *right-click* any of the folders and assign a description tag. As your list of projects builds, this extra step usually pays off.
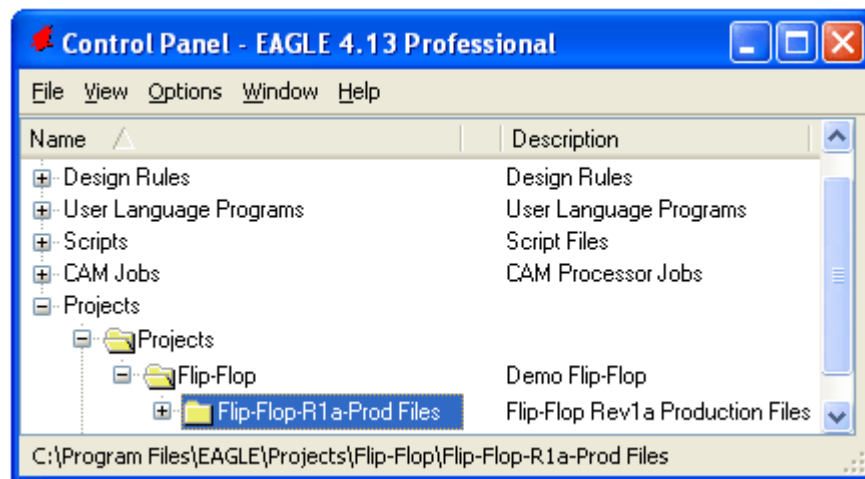
● Open a board file (*. brd)

First, open the target project with the control panel of EAGLE, and display a board file. *Be sure you are opening the *.brd file in the Production Files folder (if you created one).*
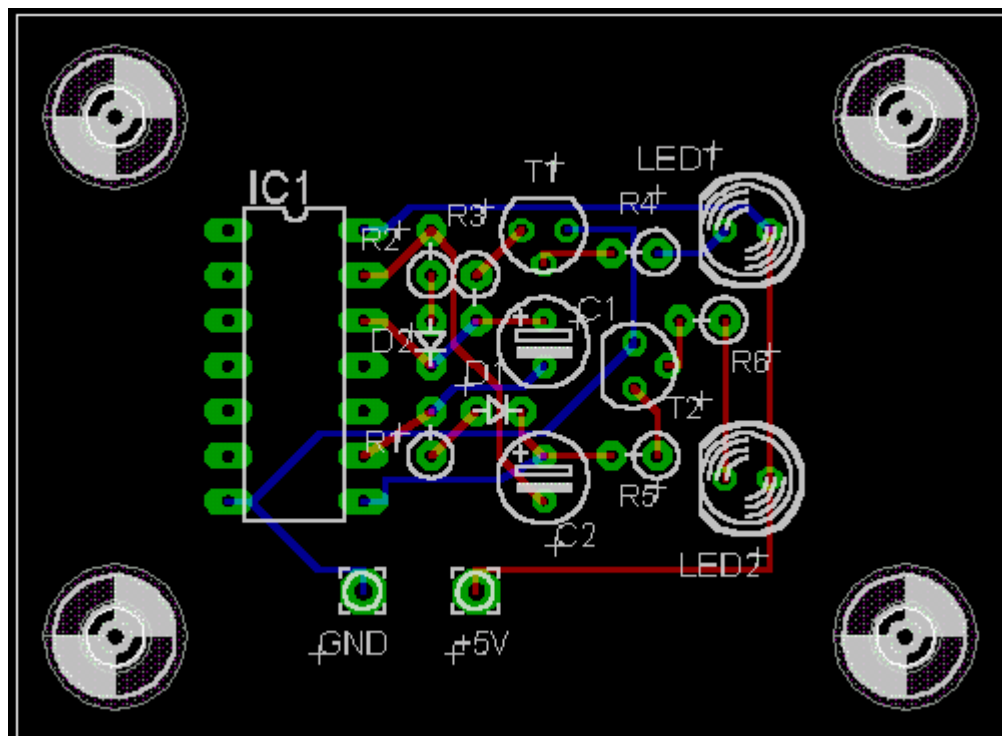
To help ensure that you don't run into any weird problems later (and sometimes these can take an unreasonable time to debug), I strongly recommend you perform the following operations.

Check for extraneous drawing entities

1. Display all layers. In EAGLE's layout editor, select the **Display**/Layer command icon -> **ALL**, *or* on the command line type ***display all*** and press **Enter.**
2. Zoom -> Fit; *or* on the command line type ***window fit*** and press **Enter**.
3. Visually examine the display. If there are any entity insertion points outside the board's dimension line, be sure to either move them into the board area or delete them as applicable.

Ensure PWB design origin point is set at absolute (0,0). **If not** …

4. Use the **Group** command to select all entities (Press F1 for Help if you're unfamiliar with using the *Group* & *Move* commands).
5. Set the **Move** reference point exactly at the board layout dimension line lower-left point, mouse right-click, and drag the grouped entities to the EAGLE layout editor's absolute (0,0) reference point.

🔴 Creation of a Drill Rack file

To avoid problems in upcoming steps, you must begin by creating the Drill Rack file. To Execute this operation, you can either

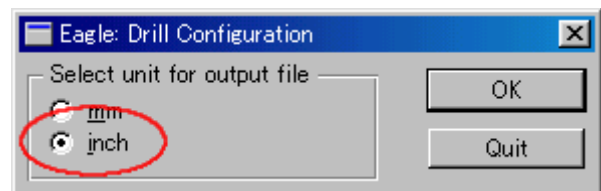- click the icon on the upper toolbar labeled ULP. In the resulting directory listing box select ***drillcfg.ulp***;
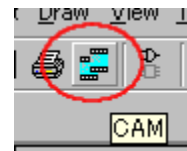
or

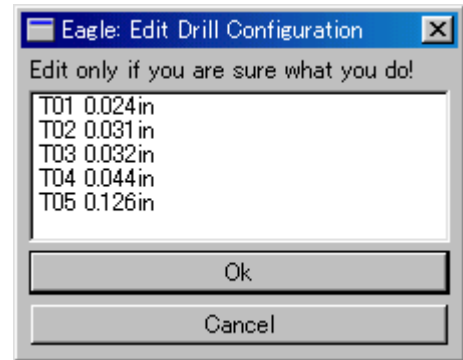- simply type **run drillcfg** on the Command Line at the top of the layout editor work screen, and press **Enter.**
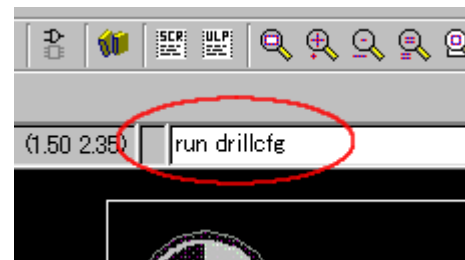
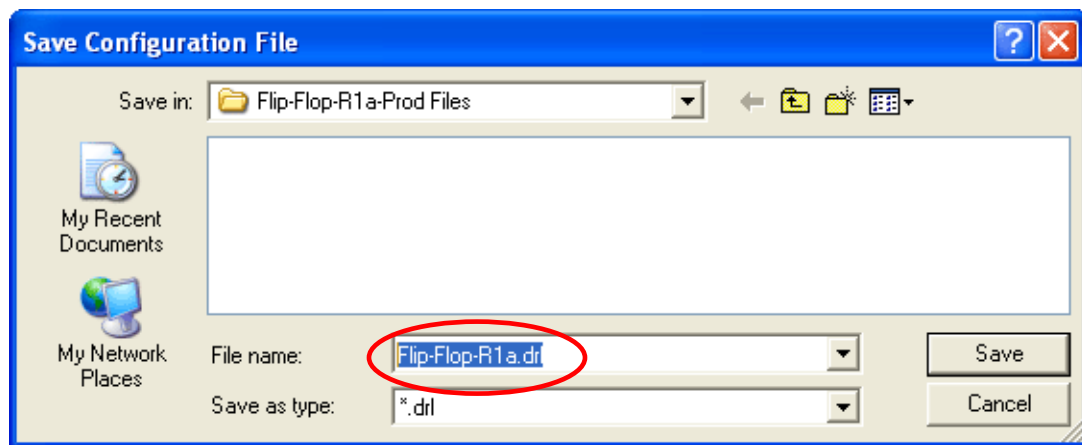Set **Units=inch**, and click **OK**.

The list of the drill sizes used is displayed.

Click the OK button.

Save the Drill Rack file you've just generated in the same folder as the (production) board file (*. brd). The information written to the board file will be used in CAM Processor to be performed next.

If you open the *.drl file in Windows Notepad (or similar editor), you'll see the contents listing similar to that shown below. You'll notice that it is the same as the contents displayed in EAGLE's Edit Drill Configuration dialog box from a couple of steps back.

Each item listed is always prefixed by 'T' = Tool and it's corresponding number incrementally commencing from 01. The associated drill size always starts from the smallest to the largest.
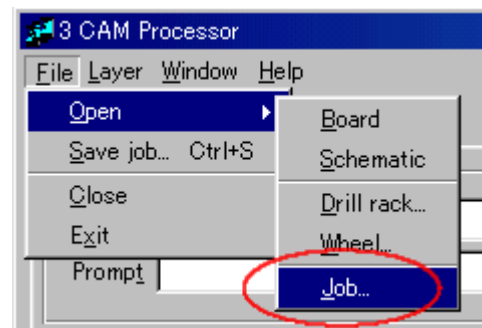
```
T01 0.024in
T02 0.031in
T03 0.032in
T04 0.044in
T05 0.126in
```

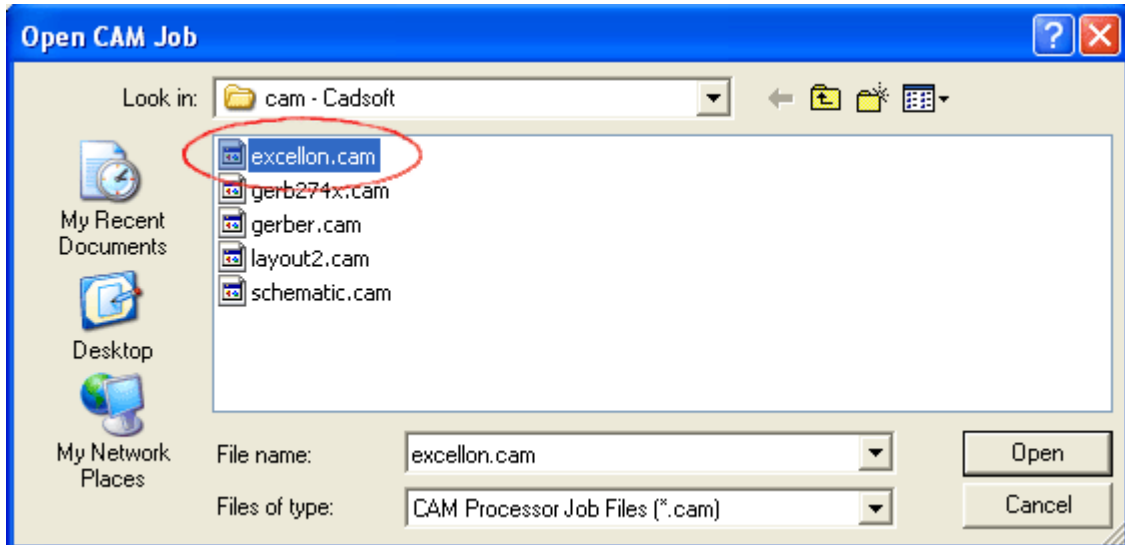## 🔴 Creation of Excellon drill files

In the EAGLE's layout editor window, choose the blue icon "CAM" on the upper tool bar. This opens the CAM Processor Configuration dialog box.
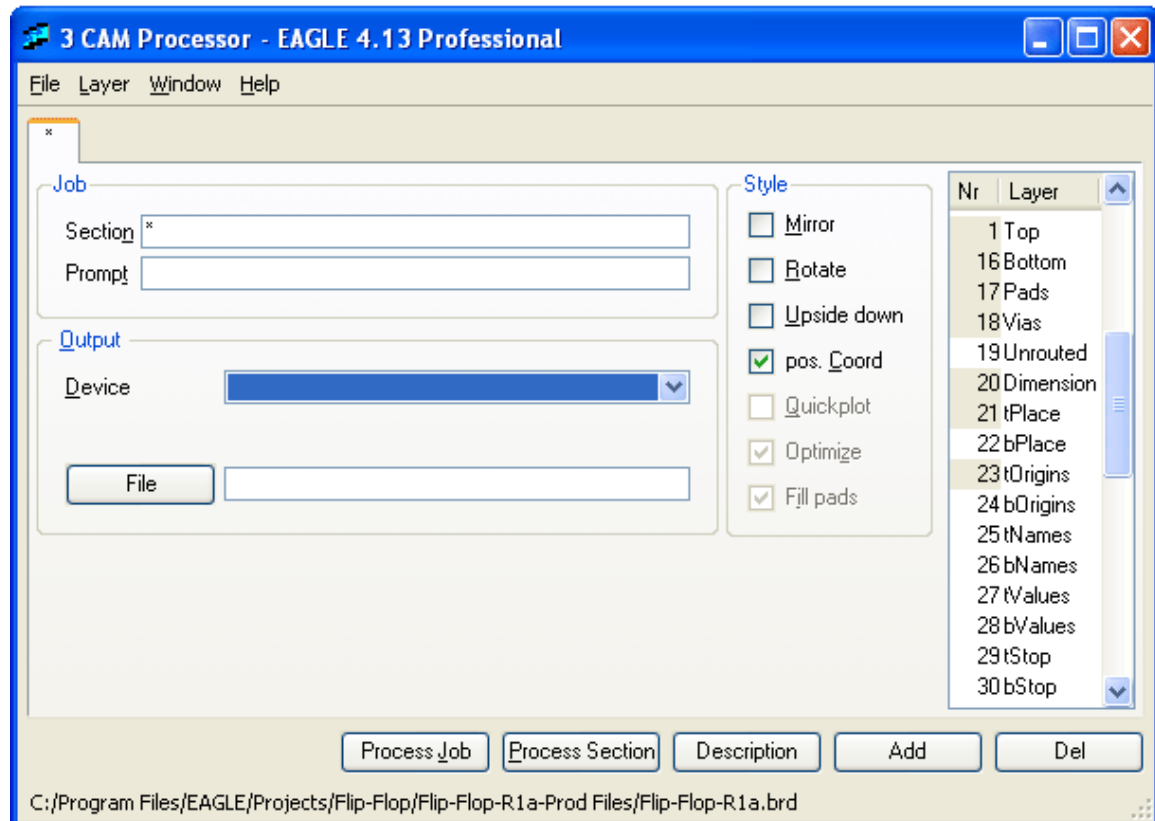
At the top-left of the CAM Processor dialog, choose File -> Open -> Job.

Choose **excellon.cam** from the displayed list. Click **Open**.

Confirm that the **Job** section of the CAM Processor dialog displays **Generate drill data**, and the **Output Device = EXCELLON**. Click the **Process Job** button. All the other settings are predefined in EAGLE and rarely ever need changing – so, don't play with these unless you know exactly what you're doing.

The Excellon drill files (*. drd, *.dri) are generated during by this process.
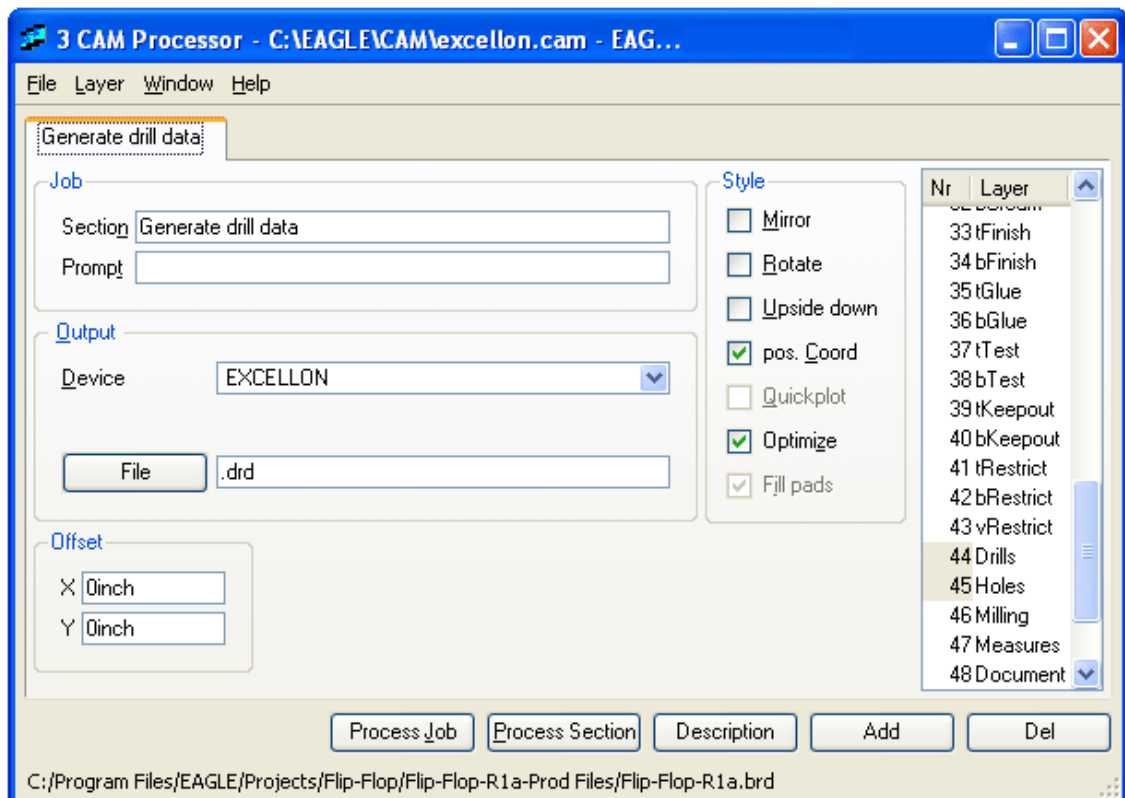
🔴 Creation of Gerber files

Unless otherwise requested by your PWB fabricator, you should prepare your Gerbers in 274X format as described in the following steps. Although there are other (older) Gerber file formats, 274X is the most widely accepted standard in which the tool information is embedded into the Gerber data rather than requiring you to generate and include a separate aperture file. Eagle CAM does support the other formats should you need them.

In the CAM Processor dialog, choose **File** -> **Open** -> **Job** -> **gerb274x.cam** -> **Open**.
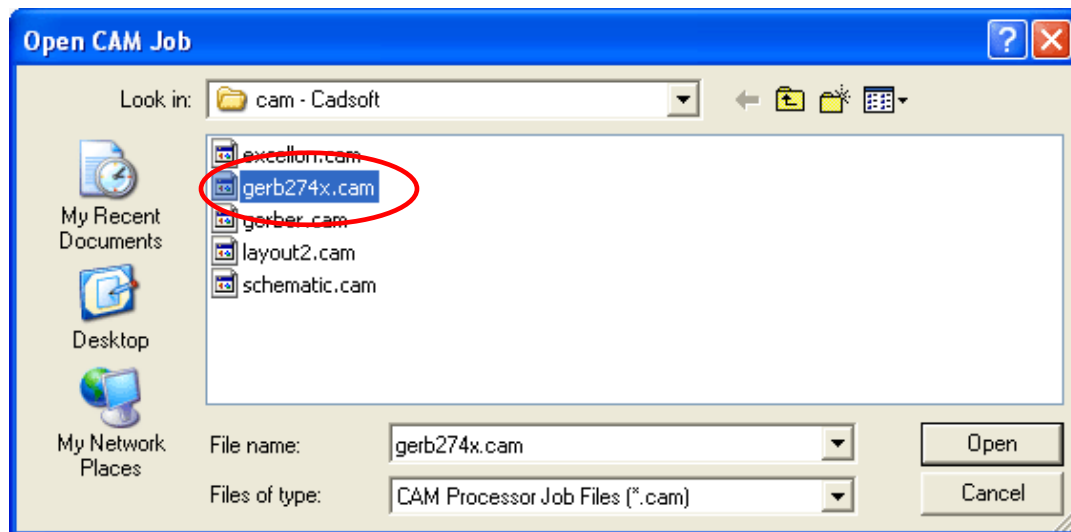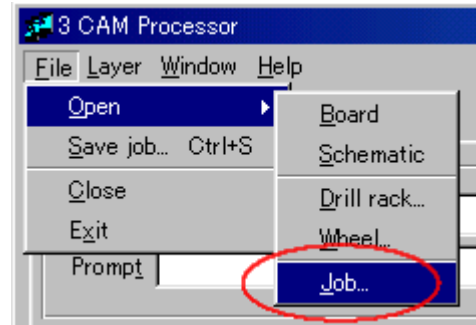
In the main CAM Processor dialog, you should now have the following EAGLE default settings. Near the top of this dialog, you'll see a sequence of tabs – one for each Gerber layer to be generated. Click on each tab in succession. Observe **Style** -> **Mirror**. Ensure that **Mirror** is *deselected* for each. Once confirmed, click **Process Job**.
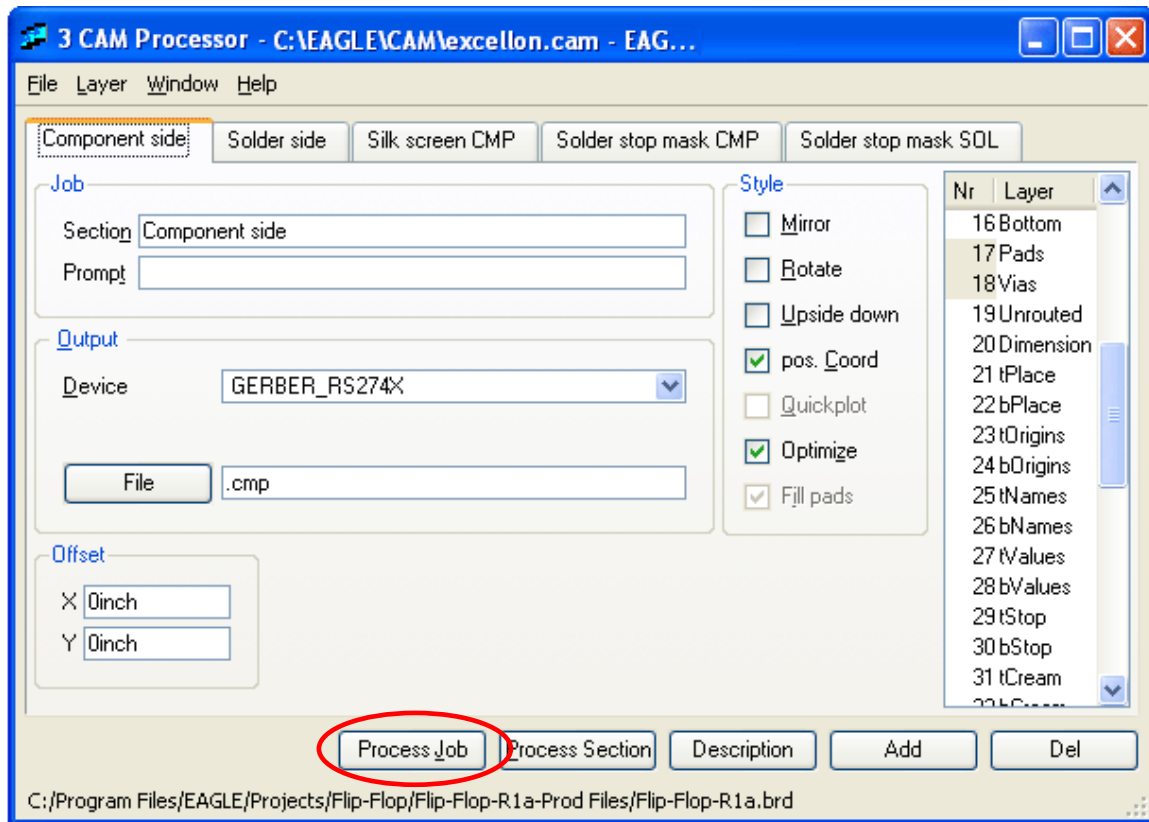
On completion of this process, the Gerber files (*.cmp, *.sol, *.plc, *.stc, *.sts, *.gpi) should be placed in folder where your (production) *.brd file originated.

At this point, you should confirm that your project production files folder contains the following essential files to be supplied to your PWB fabricator.

      *.drl   Tool (drill rack) definition file
      *.drd   Excellon drill file
      *.dri   Drill Station Info File
      *.cmp   Gerber Component (top) solder layer
      *.sol   Gerber Solder (bottom) copper layer
      *.plc   Gerber Component silkscreen identification layer
      *.stc   Gerber Component (top) soldermask layer
      *.sts   Gerber Solder (bottom) soldermask layer
      *.gpi   Gerber Photoplotter information file

Note: These are the production files needed for a 2 layer PWB. If you've designed a 4+ layer board, you should have more Gerber layer files in your folder.

This completes the easy part!

<div style="border: 3px solid black; background: yellow; padding: 20px;">

It's important to remember

The board shop will make your board exactly as the data contained in these files. Most shops won't alert you to any bugs – obvious or not. About the only time you may get any feedback from them is when you have not included all the files they need.

Therefore, it's critical to ensure that the data you send, exactly represents the patterning and physical attributes that you require in assembling and attaining electrical functionality of your circuit.

</div>

To avoid nasty and expensive surprises, you should carefully examine your CAM output data (layer-by-layer) *before* you send the file bundle to the fab shop. More often than not, some flaws are caught in the Gerbers. Most bugs do not show up on EAGLE's Layout Editor screen. Don't be fooled though! ***Just because it 'looks' right doesn't mean that it is right!*** Sometimes bugs are easy to fix, other times they can take hours to track down. They always come down to an error on the designer's part - most often because sufficient time wasn't invested in reading reading EAGLE's tutorials and adhering to the design procedures and rules.

To check your CAM output you'll need a freeware Gerber Viewer like GC-Prevue or Viewmate. Both programs should be available on most Lab computers.

Once you're happy that all checks out in the production files, you should create an order specification file in a simple text editor (like Notepad). Name it README-(your project name).txt

<p style="color: magenta; text-align: center;">For a lot more detail on checking Gerbers and preparing production quote/order specifications, please refer to my document Gerber–Getting Started.pdf.</p>