Eagle PCB -> LPKF Milling Machine Mini-How-To

EAGLE 4.09rl (Linux) CircuitCAM 3.0 (99) and BoardMaster 3.0 (45)

Steve D. Sharples

Last update: 1st October 2002... new stuff in yellow

Random link: Linux on a Dell Latitude CP M233XT
Random link: Agfa Snapscan 310 with Linux
Random link: strip_nl - strips newlines/CR (\n) from files (using Perl)

1. PREPARATION - OCTAGONS

There's some inconsistancy between the way Eagle generates the Gerber plotting data for octagons, and the way that CircuitCam interprets this data. This can lead to `disappearing pads' apparently, if any of the pads in your circuit are octagons. The fix to this problem was provided by **Andrew Sterian** in the news.cadsoft.de support forum (6 Nov 2001).

Additionally, it is necessary to make the octagons appear as circles to get a board that can be reliably-soldered: the pads get milled as octagons ok, but the track, rather than going to the edge of the pad, gets milled right to the hole (and is thus insulated from the rest of the pad) which means that the majority of solder you put between the pad and the leg of the component will not be electrically-connected to the track(s), leading to dangers of non-connections. However, quite usefully, within Eagle it's still an octagon, and any "copper pour" you put around the pins will be octagonal. The area between the pad and the pour area is milled out by the milling machine, with a circular pad and an octagonal "surround," so it actually looks pretty octagonal anyway.

- close down Eagle
- locate the file "eagle.def" (on Linux machines this is in /opt/eagle/bin)
- make a backup of this file (eagle_orig.def or something)
- search for the string "AMOC8" ... the line reads:
  `"%%AMOC8*\n5,1,8,0,0,1.08239X$1,22.5*\n"\ Octagons are emulated with a circle (using 8 vertices)`
- change the "22.5" to "0.0" ie:
  `"%%AMOC8*\n5,1,8,0,0,1.08239X$1,0.0*\n"\ Octagons are emulated with a circle (using 8 vertices)`
- comment out the line (ie, stick a ";" at the start of it):
  `;Octagon = "%%AD%sOC8,%6.4f*%%\n" ; (code, diameter)`
- uncomment the line:
  `Octagon = "%%AD%sC,%6.4f*%%\n" ; (code, diameter)`
- save the file (as "eagle.def")

2. EXPORTING GERBER FILES

2a. Top and Bottom Layers

- restart Eagle
- from the Control Panel, click on "CAM Jobs"
- double-click on "gerb274x.cam" ... the CAM processor window pops up
- click File -> Open -> Board ... select your board
- the "Component side" tab should be top-most.
- click "Process section" - will create "your_board.cmp"
- click on the "Solder side" tab
- in the "Style" section, DESELECT "Mirror" (you don't want this!)

- click "Process section" - will create "your_board.sol"

2b. Board Outline

- With the "Solder side" tab still at the top...
- click on "Add" ... will create another "Solder side" tab
- rename "Job Section" to "Outline"
- DESELECT all the selected layer, then SELECT "20 Dimension"
- change "Output File" to ".otl" (or something)
- click on "Process section" - will create "your_board.otl"
- close the CAM Processor window

3. EXPORTING THE EXCELLON DRILL FILE

3a. Drill Configuration File ("Aperture file")

- load up your board into the board editor
- from the command line, type: "run drillcfg.ulp"
- a window pops up, asking for units... default is mm, select INCH.
- another window will show you the file it's created... something like:
  ```
  T01 0.024in
  T02 0.031in
  T03 0.032in
  T04 0.040in
  ```
- click "Ok" it'll suggest "your_board.drl" as the output filename.
- click "Ok"

3b. Excellon Drill File

- from the Control Panel, click on "CAM Jobs"
- double-click on "excellon.cam" ... the CAM processor window pops up
- click File -> Open -> Board ... select your board
- click on "Process Job" - will create your_board.drd (and some other things)

4. THE FILES YOU NEED

- locate the files you've just created... on Linux, somewhere like ~/eagle/your_project/
- save the following files to a floppy/ftp them/move them somewhere else:
  - your_board.cmp - Gerber file, top layer
  - your_board.sol - Gerber file, bottom layer
  - your_board.otl - Gerber file, board outline layer
  - your_board.drl - drill configuration/"Aperture" file
  - your_board.drd - Excellon drill file

5. IMPORTING THE FILES INTO CIRCUITCAM

5a. The Gerber Files (Top, Bottom and BoardOutline)

- start CircuitCam
- click File -> Import
- find your files, select "your_board.sol" ... click "Ok"
- CircuitCam should recognise it as "Gerber-X"
- in the "Aperture/Tool list" box, type "bottom" (or something)
- Select "Bottom Layer" ... click "Ok"
- Bottom Layer tracks and pads should appear (in green)

- click File -> Import
- select "your_board.cmp" ... click "Ok"
- CircuitCam should recognise it as "Gerber-X"
- in the "Aperture/Tool list" box, type "top" (or something)
- Select "Top Layer" ... click "Ok"
- Top Layer tracks and pads should appear (in red)
- import the BoardOutline layer in the same way as the above 2 layers
- the BoardOutline should appear (in yellow)

5b. The Excellon Drill Data

- click File -> Import
- select "your_board.drl" ... click "Ok"
- click the "Apertures/Tools select" button
- select "Eagle_Excellon.txt" from the "Aperture/Tool Template" list
- in the "Aperture/Tool list" box, type "drills" (or something)
- click "Ok" - a window should pop up saying "4 apertures were recognised" (or something) ... click "Ok"
- click File -> Import
- select "your_board.drd" ... click "Ok"
- CircuitCam should recognise it as "Excellon"
- in the "Aperture/Tool list" box, SELECT "drills" FROM THE LIST
- Select "DrillPlated" (or DrillUnplated for single-sided boards) layer... click "Ok"
- The drill holes should appear (in blue) - note that if you have selected the destination layer as "DrillUnplated" you won't be able to see them... change the display order using View -> Layers...

6. INSULATING THE COPPER

- click Edit -> Insulate...
- select "Bottom layer"
- click "Run"
- click Edit -> Insulate...
- select "Top layer"
- click "Run"

7. BOARD CUT-OUT

- click Edit -> Contour Routing...
- select "Source layer" ... "BoardOutline"
- select "Destination layer" ... "CuttingOutside"
- tool should be "Contour Router 2.0mm long"
- click "Run"
- click on the INSIDE of the BoardOutline layer (yellow) to select it
- if you selected the correct object (the outline should go pale yellow) then click the "select layer" button to select ALL objects in the BoardOutline layer
- once selected, ERASE the layer (using the "X" button) - all the yellow should go
- click near the grey CuttingOutside rectangle to select it
- use the "+" and "-" keys on the keypad to move the "*" around the rectangle
- press "Ctrl-g" to make a gap in the middle of one side
- use the "+" and "-" keys to move the "*" around to the other side
- press "Ctrl-g" to make a gap in the middle of the other side
- de-select everything (right-click -> Cancel)

8. SAVING AND EXPORTING

- click File -> Save As...
- save as "your_brd.cam" ... MAXIMUM 8 LETTERS (BoardMaster is very old)
- click File -> Export -> LPKF CircuitBoardPlotter... saves "your_brd.LMD"

9. IMPORTING BOARD INTO BOARDMASTER

- start BoardMaster
- click File -> Import -> LMD/LPR...
- select your file "your_brd.LMD" ... click "Ok"
- check everything's there and ok (view both sides, select "Real Tools" etc)

Any comments, or even if you read this Mini-How-To and actually use it, please mail me. Also mail me if I've failed to reference a source of knowledge for anything, I'll be happy to correct any omissions.

Hope it's useful!
