Olimex panel preparation from Eagle CAD files

ver.1.1 24/04/12

Setup (to do only once)

1. Download `Olimex_Gerb274x.cam`
2. Download `DrawDSSpanel.scr`
3. Download `DrawDSQpanel.scr`
4. Modify `EAGLE.DEF` file following these instructions:

   - Go in your EAGLE BIN folder
   - Make copy of `EAGLE.DEF` file
   - Open `EAGLE.DEF` and replace the [EXCELLON] section with:

```
 [EXCELLON]

Type = DrillStation
Long = "Excellon drill station"
Init = "%%\n"
Reset = "M30\n"
ResX = 100000
ResY = 100000
;Rack = ""
Select = "%s\n" ; (Drill code)
Drill = "X%6.0fY%6.0f\n" ; (x, y)
Info = "Drill File Info:\n"\
"\n"\
" Data Mode : Absolute\n"\
" Units : 1/1000 Inch\n"\
" End Of Block : CR/LF\n"\
"\n"
```

Panel creation

What follows assumes that you want to create a single panel with 3 PCBs: 2 of type A and 1 of type B. The brd files are `A.brd` and `B.brd`.

The boards A and B need to be checked (e.g. for track width and drilling diameters) *before* proceeding with the following steps.

Panel

1. Create a new Eagle BRD file without schematic. (File/New/Board).
2. Click the SCR icon. Open `DrawDSSpanel.scr` or `DrawDSQpanel.scr` depending if you are creating a DSS/SSS or DSQ/SSQ panel. This draws in the Document layer a rectangle the size of the panel. The Document layer is not used in the PCB production.
3. Save the brd file you just created as `panel.brd`. This will be your panel file.
4. Close Eagle.

Copy board A to the panel

5. Open `A.brd`.
6. Click the UPL icon. Open `panelize.ulp`
7. Click Execute.
8. Eagle shows a window titled “Eagle: descriptions”. Click [Execute]
9. Eagle has altered the `A.brd` file. If you save it, the changes will be permanent.
10. Click the Group icon. Using the mouse, select the *entire* board.
11. Type the command `cut;`
12. The modified board is now in the Eagle clipboard
13. *Without closing Eagle*, open `panel.brd`. It should be in the File/Open Recent… menu.
14. Eagle will ask “Save?” *Click [No]*.
15. Eagle opens `panel.brd`
16. Click the Paste icon
17. Eagle may show a message regarding conflicting classes. Click [Yes].
18. Position PCB A in the panel and click to put it in the desiderate position. Don’t worry at this stage how part names and values look like.
19. Click the paste icon again for the second copy of the PCB A
20. Position PCB A in the panel and click to put it in the desiderate position.
21. Click the Save icon.

Copy board B to the panel *(repeat what you did for PCB A)*

22. Open `B.brd`.
23. Click the UPL icon. Open `panelize.ulp`
24. Click Execute.
25. Eagle will show a window titled “Eagle: descriptions”. Click [Execute]
26. Eagle has altered the `B.brd` file. If you save it the changes will be permanent.
27. Click the Group icon. Select the entire board.
28. Type the command `cut;`
29. The modified board is now in the Eagle clipboard
30. Without closing Eagle, open `panel.brd`. It should be in the File/Open Recent… menu.
31. Eagle will ask “Save?” Click [No].
32. Eagle opens `panel.brd`
33. Click the paste icon
34. Eagle may show a message about a library update. Click [Ok].
35. Eagle may show a message regarding conflicting classes. Click [Yes].
36. Position the PCB in the panel and click to put it in the desiderate position. Don’t worry at this stage how part names and values look like.
37. Click the Save icon.
38. The panel is now ready in Eagle format.

CAM files generation

39. If it is not open, open `panel.brd`
40. Click on the UPL icon. Open `drillcfg.ulp`
41. Select mm and click [Ok]
42. Eagle shows a list of diameters. Click [Ok]
43. Save the `panel.drl` file in same directory of `panel.brd`

44. Open the CAM processor (File/CAM Processor)
45. In the CAM processor, open the `excellon.cam` job (File/Open/Job…)
46. Click [Rack] and select the `panel.drl` you just created.
47. Click [Process Job]
48. If Eagle shows a warning about extremely large plot of data, click [Yes to All].
49. Eagle should have generate two new file: `panel.drd` and `panel.dri` in the same directory of `panel.brd`

50. In the CAM processor, open the `Olimex_Gerb274x.cam` job (File/Open/Job…)
51. Eagle will warn you that `excellon.cam` had been modified. Click [No].
52. The `Olimex_Gerb274x.cam` is set to produce Silkscreens for *top and bottom layers* and for *Names (e.g. R1, C10) and values (e.g. 10k, 100n)*. Moreover, it produces files with double side PCBs. To set the CAM files generation according to your preference you have to unselect what you don't need.
53. This is the list of the Layers on the right side of the CAM processor to enable/disable:

Information	**File**	**Layers**
Top tracks (remove for single side PCBs)	Component (top) side	1: Top
Top silkscreen, names	Silk screen TOP	125: _tNames[1]
Top silkscreen, values	Silk screen TOP	27: tValues
Bottom silkscreen, names	Silk screen BOTTOM	126: _bNames[2]
Bottom silkscreen, values	Silk screen BOTTOM	28: bValues

54. Click [Process Job]
55. If Eagle shows a warning about extremely large plot of data, click [Yes to All].
56. Eagle will take a while to complete the job.
57. Eagle may show the warning "No layers active!". This happens if you remove all layers from a file generation (e.g. Silkscreen from the bottom layer). Click [Ok]
58. The files are ready.

59. Create a zip file with following files and send it to fastpcb@olimex.com

`panel.cmp`	Top Copper layer (include only for double side PCBs)
`panel.sol`	Bottom Copper layer
`panel.stc`	Top stopmask (include only for double side PCBs)
`panel.sts`	Bottom stopmask
`panel.plc`	Top silkscreen
`panel.pls`	Bottom silkscreen (include only if you wish to order the bottom silk screen)
`panel.drl`	Drill rack file
`panel.drd`	Holes coordinates file
`readme.txt`	Text file with instructions for billing and delivery.

[1] Please note that this is not a standard Eagle layer.
[2] Please note that this is not a standard Eagle layer.

Summary

a) Create `panel.brd` and run the `DrawXXXpanel.scr` file.

b) For each PCB:
   1. open it
   2. run `panelize.ulp`
   3. select the entire PCB and copy it with `cut;`
   4. paste it in `panel.brd`

c) Open `panel.brd`
d) Run `drillcfg.ulp` and save `panel.drl`

e) Open CAM processor
f) Run `excellon.cam using` [Rack] = `panel.drl`

g) Open `Olimex_Gerb274x.cam`
h) Select/deselect the appropriate layers[3]
i) Click [Process Job]

j) Put the following files in a single ZIP file:

`panel.cmp`	(only for double side PCBs)
`panel.sol`	
`panel.stc`	(only for double side PCBs)
`panel.sts`	
`panel.plc`	
`panel.pls`	(only for bottom silkscreen)
`panel.drl`	
`panel.drd`	
`readme.txt`	

k) Send the ZIP file to fastpcb@olimex.com

---

[3] You may want to save a version of `Olimex_Gerb274x.cam` with your favourite layers selection for future use. E.g. as `myOlimex_Gerb274x.cam`