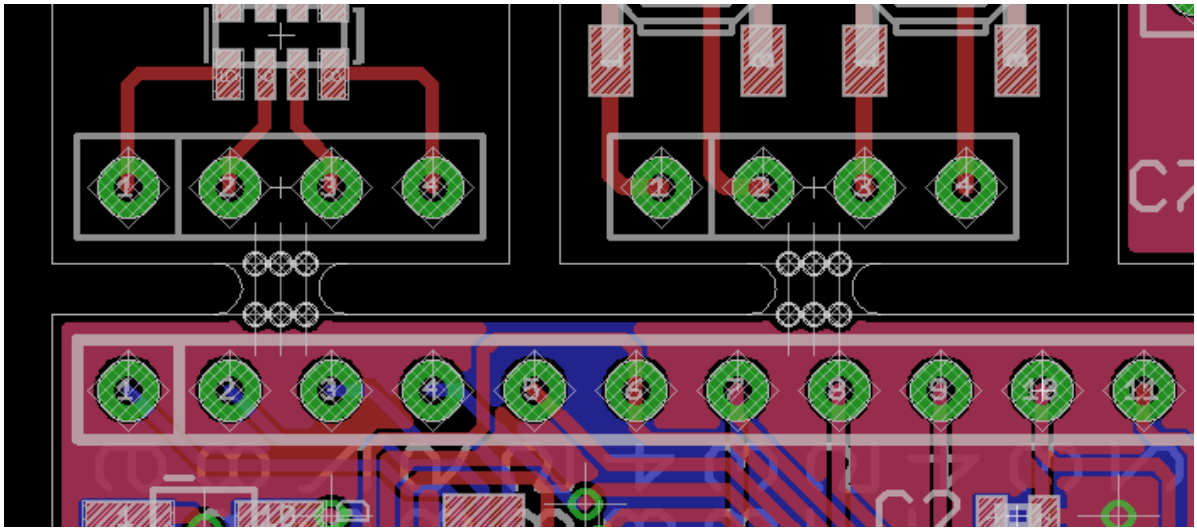
# EAGLE: Panelizing made easy

by Sjaak | Jan 3, 2017 | Blog, EAGLE, PCB, Prototyping, tipsntricks | 0 comments

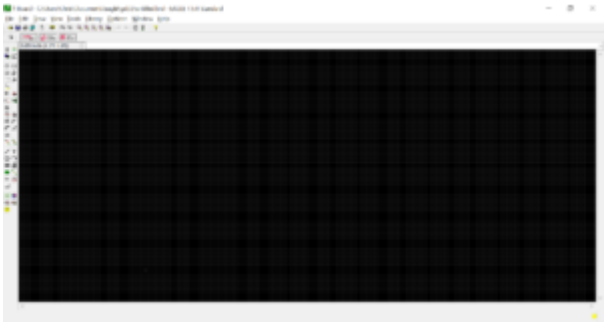


Today I stumble across a neat/new feature in Eagle I haven't seen before. Prolly in there since early versions 7, but at least it is present in version 7.5.0 (current version at writing is 7.7). Some PCB manufacturers will accept panelized PCB and you can save a few bucks by combining a few small designs into a larger design. This will save cost and the environment.

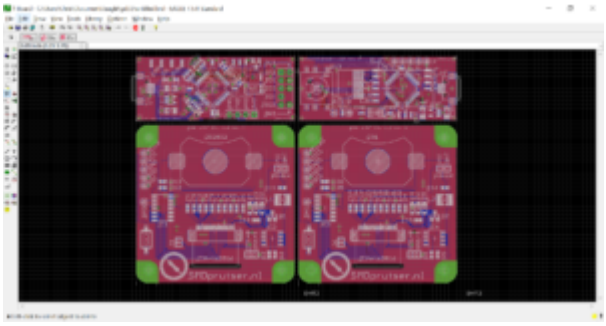
Panelizing is done by machining a slot between two or more boards, but keep them attached by a small amount of PCB material (mousebites). I used to do it by hand: generated all the schematics into multiple sheets and then route the board and finally add the slots with mousebites in the PCB editor. I generally use slotwidth of 50 mil and the smallest drill possible (12 mil) 12.5 mil apart as breakingline. I tend to place the mousebites about 2cm from each other to maintain PCB strength.

The handy feature I found is in the PCB editor (schematics has one too!) is to import an other PCB design. First you design the boards individually and then import them all into an empty board file and place them 50 mil from each other, add mousebites to taste and send them off 🙂

Start with an empty PCB:



File, Import, EAGLE Drawing, select the board(s). Add all the PCBs to the board and maintain 50 mil spacing:



And as a final step add the mousebites:


Generate the gerbers and send them off! But first run the design rule check with the proper one from your boardhouse!

**Downsides/caveats:**

- Componentname needs to be unique, so they will be renumbered in the process. Use another layer as name layer (_tsilk or _bsilk for example) and modify your cam file accordingly.
- Error messages will popup 'Conflicting net class 0' about clearance, accept them and run the DRC as the last step.
- If you use standard PCB sizes from a library you can't delete the dimension layer and this will prolly fail as the resulting PCB won't be joined properly. Use this copy trick to use standard PCB sizes.

Other options are the panelize.ulp (standard EAGLE ulp) or gerber file editors like this one.