More Beagle CAD Paws

Screaming Circuits

Continuing on from my last post [1]...

As I said, I do everything I can to avoid re-using the package footprint when adding the the parts library in Eagle CAD. The schematic symbol can be a different story though. It still takes a lot of caution, but it's less risky (in my opinion) than reusing the package footprint.

Eagle version 6 made some improvements in the way copy and paste works. It's still a little different from your typical word processor, but it's not that difficult.

[2]But before I get to that, I want to mention one item that caused me a fair amount of confusion early on. And that's the way all of this fits together. There are three buttons you will need to worry about. From left to right in the green oval are; the device, the package footprint, and the schematic symbol. In my last post, I pointed out the package footprint and today I'm talking about the schematic symbol.

Really, you only build the footprint and the schematic symbol. Then you connect the two up to create the devices. And, you can build the footprint or schematic symbol in either order, but you have to have them both before the last step (the icon in the green oval with four little AND gates).

If you're using a chip that comes in a couple of different packages (e.g. DIP28, SOIC28, TSSOP28) you most likely only need to make one schematic symbol. You can make the multiple footprints and connect them up in the device section as different variants of the same part.

There are a few exceptions though. Sometimes QFN, QFP or BGA parts will have a few extra pins. In those cases, it's generally better to create a different schematic symbol.

Duane Benson
This solder paste stencil glows blue when goblins are around

http://www.ecnmag.com/blogs/2012/12/more-beagle-cad-paws

Links:
[1] http://blog.screamingcircuits.com/2012/11/beagle-cad-paw-prints.html
[2] http://screamingcircuits.typepad.com/.a/6a00d8341c008a53ef017c3424b815970b-popup