

Re: Creating a device in Eagle

Source: <http://sci.tech-archive.net/Archive/sci.electronics.cad/2005-06/msg00101.html>

- *From:* Bob Monsen <rcsurname@xxxxxxxxxxx>
 - *Date:* Sat, 11 Jun 2005 00:10:11 -0700
-

Clifford Heath wrote:

I did my first schematic in Eagle last night.
I wanted to show a piezo device (the CUI Inc CEP-1136,
see http://www.cui.com/pdf/files/buzz_cep-1130.pdf),
so I thought I'd build a device, symbol and package for it.
Not unlike the F/SU in buzzer.lbr.

I have it most of the way there, but I can't work out how
to rename the pins on the symbol or the package. I seem to
have two sets of pin names?! Possibly because I placed the
pins the wrong way around at first and had to pick them up
and rotate. There's also some green text I didn't ask for
on the three pins, saying "I/O 0". Why, and how can I change
it? I'd like to renumber pins also.

Use the 'namer thingy' tool to rename pins. It looks like a little
resistor with R2 and 10k above and below it.

You can change the visibility of the pins by editing the symbol. Use the
little 'wrench thingy', select 'visibility', and then 'off'.

One other thing, this isn't really a footprint, as the
package has three lead wires, it doesn't PC-mount. So I
dropped three "pads" on the package where the wires come
out. I guess that's ok, except the DRC will want to know
why I haven't placed it - won't it? I also want to rename
these pins.

I've dropped the LBR in a.b.s.e, in case it's of use to
anyone.

Clifford Heath.

Re: Creating a device in Eagle

To change the names of pins, edit the symbol, and then you can just click on them with the 'namer thingy' selected, and a dialog pops up. The namer thingy has a button that looks like a resistor with R2 and 10k on it.

Eagle has a bizarre interface. Learning it is like learning to type on a dvorak keyboard.

Regards,
Bob Monsen

.