

Easiest Way To Get Part Library Templates For Pcb?

Source: <http://coding.derkeiler.com/Archive/General/comp.arch.embedded/2008-08/msg00534.html>

- *From:* andersod2 <thechrisanderson@xxxxxxxxxx>
 - *Date:* Thu, 28 Aug 2008 18:18:34 -0700 (PDT)
-

Hey all,

One thing I'd like to know, and like to hear what you have to say about is how to accomplish a uniform / "easy" way of doing the following:

- (1) Obtaining information about whether a particular part is mass-produceable
- (3) Obtaining the datasheet (digikey/mouser are good for this I suppose)
- (11) Most importantly, obtaining some kind of CAD template / drawing library for insertion into your favorite PCB design tool, for quick and dirty board design.

I haven't yet learned an easy way of doing this...I know eagle uses a library file that is common to other PCB programs, but how common? My understanding is that a lot of PCB manufacturers release their own "free" softwares, but for obvious reasons, there are compatibility issues between them. I know the gerber files are pretty common, but I don't even know what that is yet...still learning.

Basically, anything to keep me from having to draw up my own parts after measuring their dimensions myself with a caliper sad.gif ..and obviously, #1 is there because if I have a choice of parts, I'd rather pick the one that is less likely to be obsolete in the near future and gives me the opportunity to make more than one board if it pleases me to do so...

any thoughts? Thanks!

.