

Re: Pad Size for TQFP Package??

Source: <http://sci.tech-archive.net/Archive/sci.electronics.design/2006-06/msg00435.html>

- *From:* qrk <SpamTrap@xxxxxxxx>
 - *Date:* Fri, 02 Jun 2006 21:09:21 GMT
-

On 1 Jun 2006 19:24:29 -0700, weg22@xxxxxxxx wrote:

Hi all,

I'm designing a PCB for a PIC18F8722 which has a TQFP 80 package. This is the PCB I've designed for an SMT type device and I was wondering what the standard pad size is for this? From what I've seen on the web, they look rectangular, but I'm not sure of the dimensions.

Also, does anyone know of free software capable of laying out a PCB (could be manually and not through schematic capture) while also being able to generate the gerber files. I know there are a lot of vendors out there which let you download their software if you pay for them to manufacture your PCBs...but I already have someone for this.

Thanks,
weg22

For getting footprint information, <http://www.pclibraries.com/> offers a free footprint viewer called LP Viewer. Lots of footprints for ho-hum parts to the latest icky packaging technology. On the right side of their web page, look for "featured items". You may want to load in a few more libraries from their "library documentation" area.

Usually, the footprint info is in the data sheet or the manufacturer has an area in their web site that has this information.

Mark

.