

Re: Pad Size for TQFP Package??

Source: <http://sci.tech-archive.net/Archive/sci.electronics.design/2006-06/msg00297.html>

- *From:* Tim Wescott <tim@xxxxxxxxxxxxxxxxxxxx>
 - *Date:* Thu, 01 Jun 2006 19:56:46 -0700
-

weg22@xxxxxxxxxxx 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 the PCB layout check the data sheet as Genome says. If Microchip doesn't put it on their datasheets there'll be a separate sheet for the package.

Eagle is a good package. Their free version will let you do small (3"x4") two-layer boards. It includes schematic capture, layout and plot to Gerber.

Tim Wescott
Wescott Design Services
<http://www.wescottdesign.com>

Posting from Google? See <http://cfaj.freeshell.org/google/>

"Applied Control Theory for Embedded Systems" came out in April.
See details at <http://www.wescottdesign.com/actfes/actfes.html>

.