

Pad Size for TQFP Package??

Source: <http://sci.tech-archive.net/Archive/sci.electronics.design/2006-06/msg00288.html>

- *From:* weg22@xxxxxxxxxxx
 - *Date:* 1 Jun 2006 19:24:29 -0700
-

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

.