Re: Pad Size for TQFP Package??

Source: http://sci.tech−archive.net/Archive/sci.electronics.design/2006−06/msg00307.html

- *From*: pbdelete@xxxxxxxxxxxxxxxxxxxxxxxxxxxxxxxxxxxx
- *Date*: 02 Jun 2006 03:33:54 GMT

weg22@xxxxxxxxxx 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.

Try:
gschem http://www.geda.seul.org/
pcb http://pcb.sourceforge.net/

gsch2pcb will convert your gschem schematics into pcb format.
QFP packages etc.. are included. Just specify footprint=QFP80 (or whatever)
in your schematic attributes. And verify correct output in pcb.
'pcb' will make your gerber files. And let you autoroute or manually adjust
layout.

.