

Re: PCB pad/hole size advice

Source: <http://sci.tech--archive.net/Archive/sci.electronics.design/2005-04/msg00573.html>

- *From:* Peter Bennett <peterbb@xxxxxxxxxxxxxxxxxxxxx>
 - *Date:* Sun, 03 Apr 2005 11:49:06 -0700
-

On Sun, 3 Apr 2005 12:29:01 +0100, "Andrew" <ajhuk@xxxxxxxxxxxxx> wrote:

>I am a rather novice amateur PCB designer. I have designed a PCB using
>Protel 99 SE, using their standard library components and I am looking at
>getting prototype PCBs manufactured. I now find that the libraries have
>used numerous different hole and pad sizes, and that more different hole
>sizes = increase cost! Further I find that some manufacturers use imperial
>drill sizes and others metric.

>

>So, I have two questions:

>1. the libraries mainly use the following pad/hole sizes (in mil=1/1000th
>inch)

>pad=62, hole=28mil/0.7112mm = decoupling capacitors

>pad=62, hole=32mil/0.8128mm = resistors

>pad=50, hole=32mil/0.8128mm = dip packages

>pad=62, hole=32mil/0.8128mm = vias

I've found that many of Protel's hole sizes are a bit small for my liking – in particular, the holes in their post header footprints are definitely too small.

I use .035" for most through-hole parts, and .025" for vias. Post headers need .040, and 1 amp diodes and larger electrolytic capacitors will need larger holes (and pads)

>

>It would seem to me that I could reasonably use a single hole size for all
>of the above. One supplier (Olimex) has standard drill sizes of 0.7mm
>(27.6mil) and 0.9mm (35.4mil). Would using 0.9mm for all of the above
>components be a sensible choice (finished hole size would be a little
>smaller due to plating)?

If the board maker quotes unplated hole sizes, you have to allow .003 or so for plating. When I order boards from the local shops, I specify finished hole size, and let the board shop figure out what drill to use (I don't know, or care, what drill sizes they have).

>2. I would prefer a design that I could send to whatever manufacturer can
>offer the best price/delivery terms. This may not consistently be the same

Re: PCB pad/hole size advice

>company depending on circumstances during the life of the product. How can
>I have a single design that can use either imperial or metric drill sizes
>depending on the supplier? I don't want to have to have two different
>design files, one imperial and one metric.
>

I would expect the board shop to translate between English and Metric dimensions as needed. There should be no need to create artwork and drill files in both measurement systems.

—
Peter Bennett, VE7CEI
peterbb4 (at) interchange.ubc.ca
new newsgroup users info : <http://vancouver-webpages.com/nnq>
GPS and NMEA info: <http://vancouver-webpages.com/peter>
Vancouver Power Squadron: <http://vancouver.powersquadron.ca>
.

• **References:**

◆ **PCB pad/hole size advice**

◇ From: Andrew

- Prev by Date: **Re: OT: Civil War Narrowly Averted in Schiavo Case**
- Next by Date: **Re: OT: Air America makes a year!**
- Previous by thread: **Re: PCB pad/hole size advice**
- Next by thread: **I loathe RS Components !**
- Index(es):
 - ◆ **Date**
 - ◆ **Thread**