

## Re: Copper pour or traces?

---

*Source:* <http://sci.tech-archive.net/Archive/sci.electronics.design/2007-01/msg01062.html>

---

- *From:* "John B" <[spamj\\_baraclough@xxxxxxxxxxxxxxxxxxxxxx](mailto:spamj_baraclough@xxxxxxxxxxxxxxxxxxxxxx)>
  - *Date:* 08 Jan 2007 17:14:42 GMT
- 

On 08/01/2007 Matt wrote:

Hello,

I am laying out a little breakout board that spreads synchro data (60/400Hz) throughout a system. This is the first board I have ever layed out that I have contimplated using a 2-layer board instead of a 4-layer board. Because of this, I will not have independent GND/PWR planes. Looking around at other 2-layer PCBs, I see that some folks like to use a copper pour on the TOP and BOT layers. I am assuming one layer is traces+GND and the other is traces+PWR. My questions are these:

Rule 1: Don't assume anything.

It is unlikely that you will ever find a board with an external power plane. Double sided boards will normally have a copper pour on each side both of which are connected to GND. Having said that, go back and invoke rule 1 above.

1. Is this a sound design (no high frequency and the signals are very strong).
2. From reading, I was convinced to try to use hatching instead of a solid copper pour to prevent warping.
3. If I do use cross hatching what should my grid be if my global spacing for all traces, pads, layers, etc. is 8 mils.

Thank you.

Just use solid copper pours. FR4 will not warp if you have copper pours on both sides.

---

Re: Copper pour or traces?

John B

.