

Re: Groundplane in poweramplifier PCB design

Source: <http://sci.tech-archive.net/Archive/sci.electronics.design/2006-04/msg02274.html>

- *From:* kensmith@xxxxxxxxxxxxxxxx (Ken Smith)
 - *Date:* Sun, 16 Apr 2006 16:25:32 +0000 (UTC)
-

In article <e1rcmb\$2o3\$1@xxxxxxxxxxxxxxxxxxxxxxxxxxxx>, Wiebe Cazemier <halfgaar@xxxxxxx> wrote:

Hi,

I've just about finished designing a PCB for a HiFi power amplifier, and now I'm wondering something. Is it a good idea to fill up the unused space on the board with ground copper (without orphans of course)? Or, is it best to keep the ground clear of as much parts as possible, as you do with the supply lines?

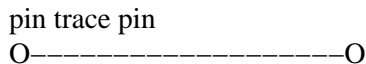
There can also be thermal reasons for putting copper in. You can't ignore these.

You want the extra copper as shielding. If you can arrange to effectively have a thick shorted turn around the whole circuit, this will help to keep AC magnetic fields from going through the PCB.

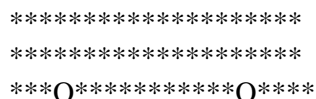
Also remember that every bipolar transistor is an AM radio just waiting to happen. The guy next door will buy a 1KW CB tomorrow.

There are other places in the design where you want the trace to be short and thick. These are the places to look at. It is likely that you will find that you want those traces so wide that they end up really being a plane. ie:

Ascii art:



Becomes:



Re: Groundplane in poweramplifier PCB design

If you have to, you can initially make the plane as a different net than GND. It will flow around the GND traces. Then you can join it up using whatever trick your layout tool allows.

The speaker returns go directly to the main power supply BTW, so the groundplane is not necessary for that.

Thanks in advance,

I assume that this amplifier has feedback. Where are you picking off the two sides of the speaker for feedback? These connections can also be trouble makers.

—
—

kensmith@xxxxxxxxx forging knowledge

.