

Re: grounded co-planar waveguide PCB design advice please

Source: <http://sci.tech-archive.net/Archive/sci.electronics.design/2006-03/msg01302.html>

- *From:* "tlbs" <tlbs101@xxxxxxxxxx>
 - *Date:* 10 Mar 2006 08:44:10 -0800
-

megoodsen wrote:

Hi,

I'm designing an RF multilayer board that is to work over 10-200MHz.

For the transmission line I'm using grounded co-planar waveguide.

Material is 10 thou Rogers 4350B, with a 50ohm track width of 0.46mm and a track to top ground gap of 0.146mm.

This is working OK.

I now want to shield the board to prevent crosstalk, but I am restricted to very little height.

My plan is to get a flat plate of say 1mm material half mm deep etched with pockets that follow the RF lines and is grounded all elsewhere.

My worry is, will the fairly close proximity (but nearly 4 times the track to gnd distance) of the metal plate above the transmission lines cause problems, and/or will I have resonating cavities too?

I can make the cavities be 3 or 4 times wider than the gnd to gnd gap around the lines if need be.

What do you think?

I designed a multilayer (20 MHz to 1 GHz wideband + digital ECL) RF board a few years ago using Rogers material. I just buried the RF traces on layer 2 under a surface ground plane and a composite power/ground layer 3.

Even then, there were a few surface layer traces necessary, and they got covered by a small shielded can. That probably doesn't help you much. Perhaps there is some portion of the board where the height requirement isn't so strict. You could place some surface mount components in that area and cover them.

Good luck,

Re: grounded co-planar waveguide PCB design advice please

Tom