

Re: Ground and Ground plane on 2 layer board

Source: <http://sci.tech-archive.net/Archive/sci.electronics.design/2007-01/msg00700.html>

- *From:* Rich Grise <rich@xxxxxxxxxxxx>
 - *Date:* Fri, 05 Jan 2007 22:25:20 GMT
-

On Fri, 05 Jan 2007 01:45:04 -0800, Liuc wrote:

My question is: how to use the ground plane? I've read plenty of threads about it, but I didn't fully understand which is the best way to do. My doubts are mainly about:

There aren't really any hard and fast rules – one thing to keep in mind is current paths. Current always flows from the power supply through the load (your circuit) and back. When you're laying out your power supplies and parts, visualize which copper will have the current flowing from and to the supply, and try to minimize the interaction between those current flows for different supplies. I think this is the idea behind the "single point" or "star" ground. If you watch your layout, that shouldn't be necessary. Where the current is flowing is what you need to watch.

Also, I've always said you can't overcapacitate. 10uF or more at the power entry point and a .1 ceramic (or more, depending on power/ground pins) at each chip has always worked for me. :-)

Good Luck!
Rich

.