Shielding effects of Coplanar Waveguide over Ground

By: Steve Hageman
www.AnalogHome.com

Now that we have gotten through all that design work [1], some might ask, why go to all the bother with CPWG (Coplanar Waveguide over Ground)?

Well, there are several reasons to use CPWG:

1. It allows us to narrow the trace almost arbitrarily for a given layer thickness while maintaining a 50 or 75 ohm impedance. This helps us to more closely match our modern component width with the PCB trace width. This not only makes our RF designs perform better it saves PCB real estate.

2. It allows another degree of freedom in our design equations. With microstrip we only have dielectric thickness and the dielectric constant to use in setting the trace width. CPWG gives us the added variable of top layer ground plane in setting our trace impedance.

3. Less current flows into the air with CPWG compared to a microstrip structure. This is because the topside ground copper shunts the field current locally to the topside ground.

Our RF designs are not only getting smaller but thinner too (have you seen those super low-profile tantalum and ceramic capacitors?). This is great, but our designs still need to meet radiated emissions and conducted susceptibility specifications. This means that our shields are getting closer to the PCB surface also.

Take a look at one of those embedded WiFi, Bluetooth or GPS modules on the market and note how low the shield height is over the PCB. Total module heights under 100 mils are now very common.

The question should be asked: “Just what is the effect of placing that metal shield very close to our carefully designed 50 ohm traces?”

Most RF designers use rules of thumb to space things appropriately. For instance, in microstrip design, many designers use a rule of thumb to keep topside copper away from their microstrip lines by four times the substrate layer thickness so as to not affect their microstrip impedance. With CPWG you don't have to worry about this since the topside ground is purposefully brought up next to the topside RF trace.

But do you have to worry about that close board shield?

To find out, I conducted a series of EM simulations using Sonnet tools [2] and the usual textbook formulas to see the effect of the shield height over the PCB.

For two different substrate thicknesses I compared microstrip with CPWG versus cover height just to get a feel on the sensitivity.

Now, just a word of caution here: It would be easy to interpolate a number off of the following graphs and think that you have a four-digit accurate value. This is simply not the case. These graphs represent the finest “simulated bits” to be found anywhere, but because they are simulated they should be used as a guide for further study and real engineering evaluation. That's how I am using this information – as a trending indicator for future engineering evaluation and proof on real hardware.

Let's take a look at a comparison of trace impedance vs cover height for a microstrip and CPWG on a 62 mil thick substrate (Figure 1).

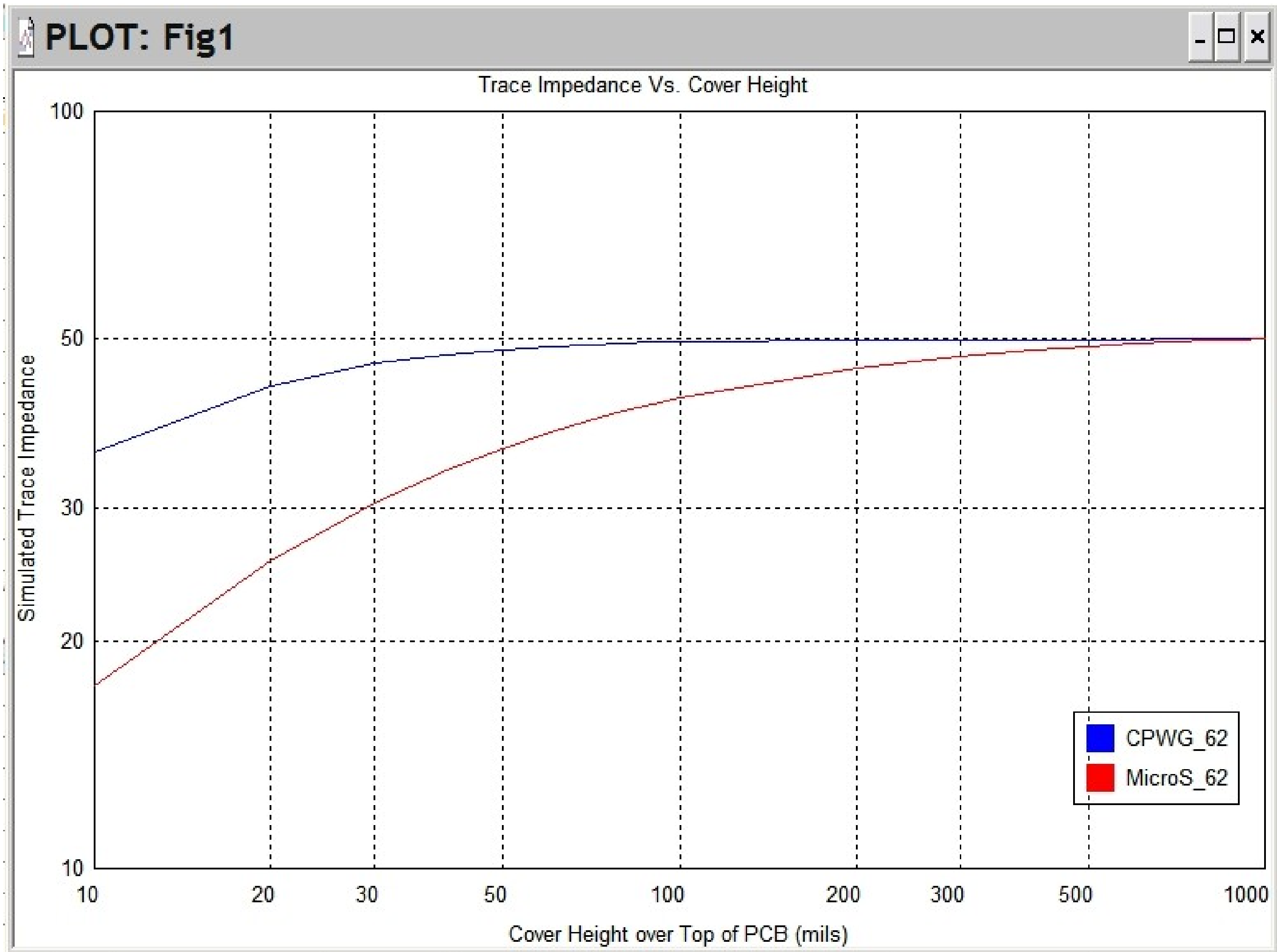

Figure 1. Comparison of trace impedance vs cover height for a microstrip and CPWG on a 62 mil thick substrate.

In Figure 1, it is easy to see that the microstrip trace impedance (red trace) is much more sensitive than the CPWG trace (blue trace) to the cover height.

Why? Well, this is proof that the microstrip trace has much more field in the air above the trace than the CPWG trace, whose field is more concentrated to the topside copper ground return path.

This is great not only because of its lack of sensitivity to close-in metal objects, but also because there are less radiated emissions with CPWG, which makes our shielding job even easier – a true win-win.

What about the sensitivity to substrate thickness? Figure 2 shows a comparison of this. The least sensitive trace is the red one – this is a CPWG trace on a 10 mil thick substrate, while the blue trace is a CPWG on a 62 mil thick substrate.

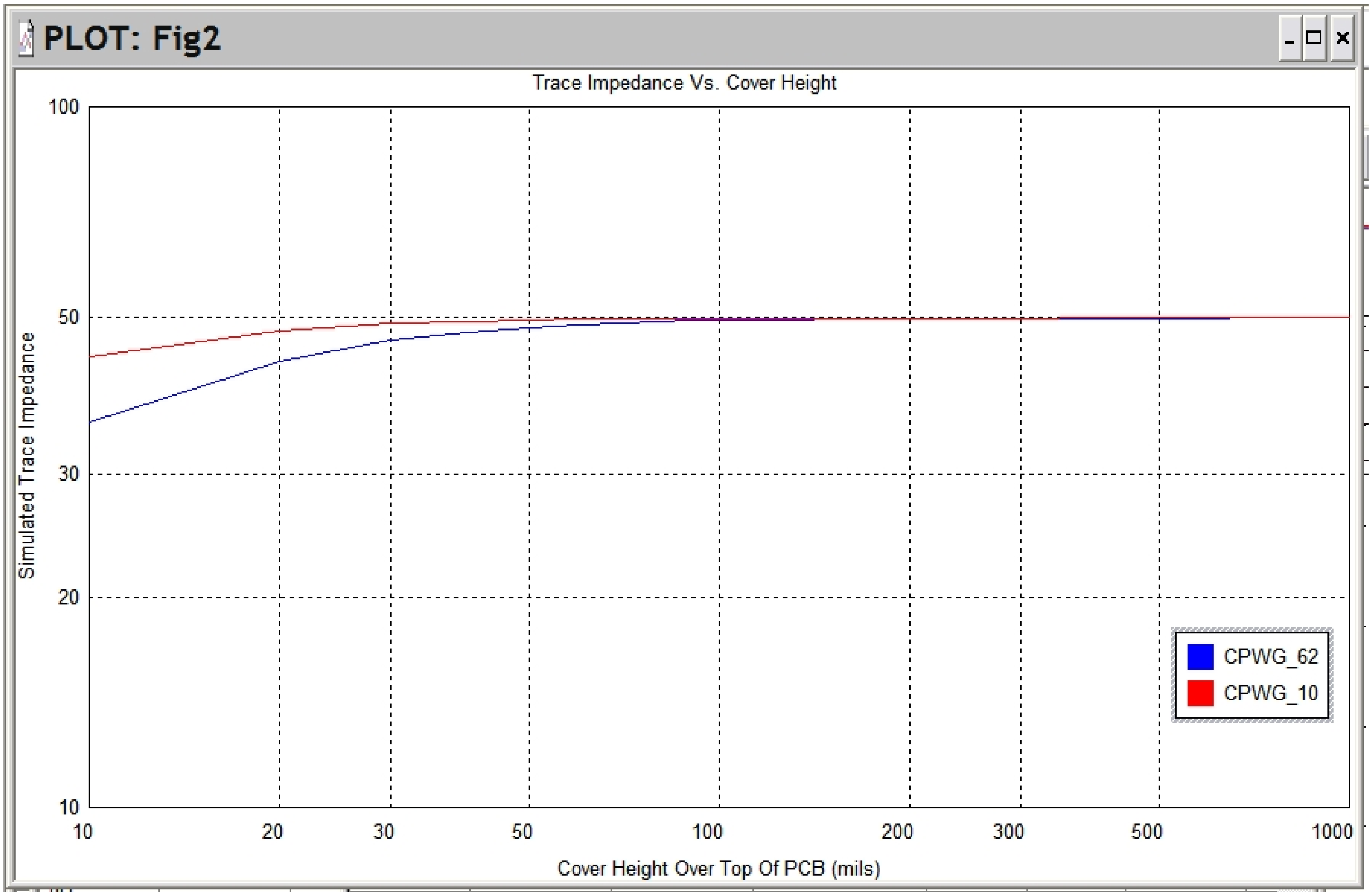

Figure 2. Trace impedance vs. cover height for a CPWG trace on a 10 mil substrate and a 62 mil substrate.

We can say that this shows that the thinner the substrate is, then the less field there is in the air above the CPWG trace. The cover only starts to affect the trace by a significant amount when it gets closer than about a substrate's distance away.

Perhaps a better way to look at data in Figure 1 and 2 is to normalize the distance to the substrate height. Figure 3 shows the data presented this way.

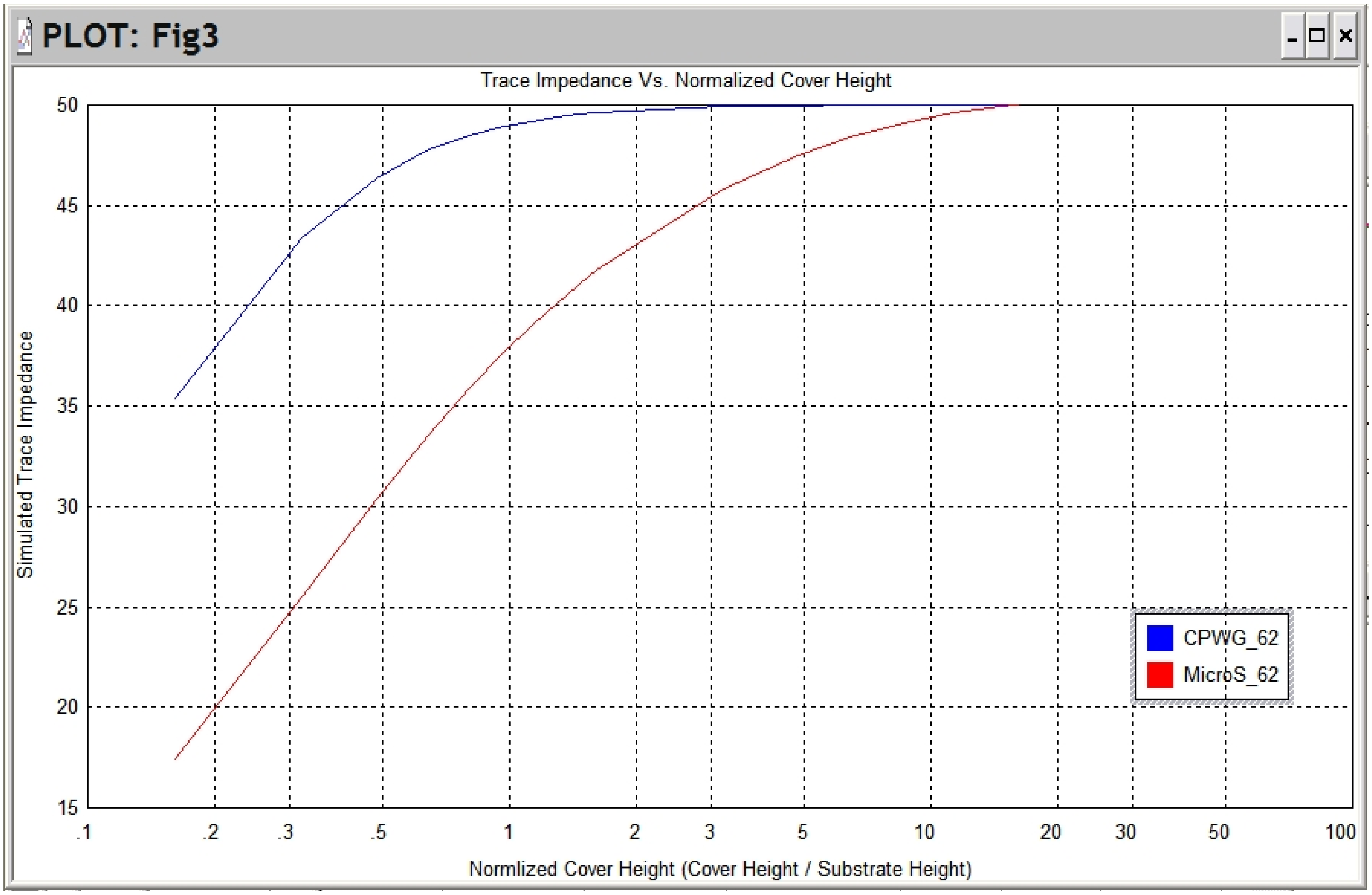

Figure 3. Data in Figures 1 and 2, with normalized distance to substrate height.

By comparing the CPWG and microstrip cases together on a normalized cover height plot, you can see that the CPWG trace (blue) only gets a really significant shift to about 45 ohms at 0.4 cover heights away, whereas the microstrip trace (red) reaches the same very low impedance when the cover height is a whopping 3X the substrate height.

This is something to keep in mind for that new smaller/faster design on the drawing board. While we can't change the underlying physics of wave propagation, with improved dielectric substrates, thinner substrate heights and smaller components, new products can and do continue to get smaller and thinner.

As designers, all we have to do is to be careful when we scale our designs, because that's where we get in trouble – when we scale too much without knowing the limits of the underlying design physics.

Acknowledgment

A version of this article was first published on March 12, 2012 on the website: www.pcbdesign007.com

References

[1] "How to make a Quick Turn PCB that modern RF parts will actually fit on!", AnalogHome App Note 004. http://www.analoghome.com/articles/an004.pdf

[2] Sonnet Software www.sonnetsoftware.com