How much is impedance affected by an adjacent trace?: Rule of Thumb #25

Eric Bogatin - February 18, 2015

Spoiler summary: If the spacing to an adjacent trace or copper fill is more than the line width, the impact on the impedance of a 50Ω microstrip line is less than 2% per side.

Remember: before you start using rules of thumb, be sure to read the Rule of Thumb #0: Use rules of thumb wisely.

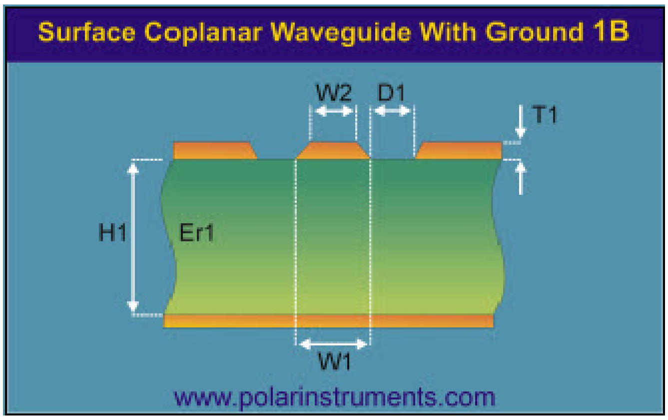
Previous: Rule of Thumb #24: When to worry about corners.

This is a really simple rule of thumb that relates to an important design feature which has a behavior contrary to what most designers believe.

So many of the properties of interconnects are about the behavior of electromagnetic fields as affected by the boundary conditions of the conductors and dielectric geometry.

While many of us studied Maxwell's equations in school, few of us developed a calibrated physical intuition about them. Yet, this is the basis of the electrical properties of interconnects. Our intuition being off results in incorrect expectations.

Consider this example of a microstrip surface trace with adjacent metal on either side. This adjacent trace could be a "ground fill", which you should never use, or just another signal line. The figure below shows an illustration of the geometry from the Polar SI9000 field solver tool.



The conductors on either side of the signal line on the top layer are connected to the return path on the bottom.

The question is, how much is the impedance of the signal line affected by the proximity of the adjacent conductor metal.

A common belief is that the signal line's impedance is going to be decreased when the adjacent grounded metal is brought close to the signal line. And it is reduced significantly.

While the general features of our expectation are correct, the amount of impact is way off. The interactions of the signal line and adjacent conductors, in the presence of the return path below, are dominated by fringe fields, so there are no good approximations to help guide us. We have to use a field solver to explore this design space.

In this system, I engineered the cross section so that when the adjacent coplanar metal is far away, like 20 line widths, the microstrip's characteristic impedance is close to 50Ω.

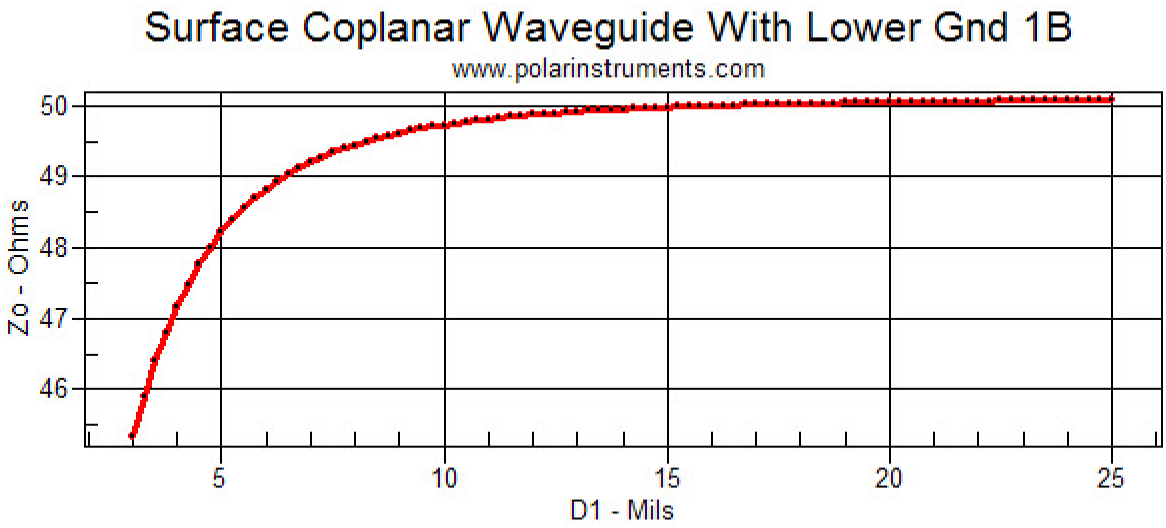
The set up for this example uses the following conditions:

- H1 = 3 mils
- Er1 = 4.3
- W1 = W2 = 5 mils
- T1 = 1.2 mils
- D1 = 100 mils

The resulting impedance is 50.11Ω. For the same conditions, the impedance of just an isolated microstrip line, calculated with the Polar Instruments SI9000 tool, is 50.06Ω. They are the same to within 0.1%.

As we bring the adjacent conductor closer to the signal line, we would expect the presence of the adjacent return path conductor to decrease the impedance of the line. This is indeed exactly what happens, but, by how much?

Here is the calculated characteristic impedance of the microstrip as the spacing between the signal line and adjacent metal decreases:



Note the vertical axis scale is 1Ω per division. When the spacing to the adjacent metal is farther than three line widths – 15 mils – there is no impact.

Even when the spacing is twice the line width away from either side, the microstrip's impedance is reduced by only 0.27Ω, which is 0.5%, and this is from metal on both sides of the surface trace.

As the adjacent metal comes closer, the fringe fields are more sensitive to proximity, and at a spacing equal to the line width, for the 50Ω line, the impact on the line's impedance is less than 2Ω, or 4%. If there was metal on just one side of the signal line, the impedance would be decreased by less than 2%.

This is the origin of the rule of thumb:

> *If the spacing to adjacent metal is more than one line width, the impact on the line impedance from metal on one side will be less than 2%.*

Keep in mind though, the impact on the characteristic impedance from adjacent metal is just one metric of performance. There are much bigger problems from routing adjacent metal to a surface trace, like resonant coupling and far end crosstalk.

Now you try it:

1. Should you worry about the impact on the impedance of a 50Ω microstrip signal line when there is an adjacent trace two line widths away?
2. Suppose the impedance of the microstrip line were to increase by moving the bottom plane farther away. Is the influence of adjacent coplanar metal increased or decreased?

Also see:

- Bogatin's Rules of Thumb
- Guard traces - use 'em, or not?

Additional information on this and other signal integrity topics can be found at the Signal Integrity Academy, www.beTheSignal.com.