Using PCB Traces as Components – Introduction

Printed circuit boards (PCBs) have a thin layer of electrically conductive copper on top of an insulating substrate – base which provides mechanical stability for conductive layer. Traces are etched on copper to isolate different parts of circuit from each other. Discrete components are then soldered on top of these traces to form an electrical circuit with desired functionality. Ideally these traces should have zero resistance, capacitance and inductance. However, no material is a perfect conductor, so traces have always some resistance, capacitance and inductance.

For the last few years, I've had a printed excerpt out of Texas Instruments design reference book “Op Amps for Everyone” taped on my kitchen cabinet. Excerpt is pages 17-28 and 17-29, summary of circuit board layout techniques. The very first bullet point is “Think of the PCB as a component of the design.”.

I've considered that as a warning to consider traces as passive components and analyze their effects on the circuit. In the past I've always tried to minimize their effect, since any unideal properties of PCB tends to lead to degraded performance: Power losses in trace resistance, noise coupling into sensitive analog circuitry and so on.

Lately when designing shields I've been somewhat unhappy with the final layouts. The shields define the mechanical dimensions of the boards: connectors have their places which cannot be modified. Especially on simple designs there is excessive board area which stays unused. I find this unused board area wasteful, and try to make the best out of it: Maybe I'll add some instructions on the silkscreen, or maybe the excess area can be used as a graphical element with copper, solder mask and silkscreen used as colors.

One morning when I was mourning for all the empty space in our next board revision I read that sentence again: “Think of the PCB as a component of the design.”. Since we are having discrete capacitors, inductors and resistors on the board and unused board space which will form those same components, maybe PCB itself could replace some of the parts?

This blog series investigates the theory of parasitic PCB properties and practicality of intentionally using these parasitic properties as components in design. Concept itself is not new, integrated circuits and high frequency designs have utilized these properties for long. We'll focus instead in macro scale applications at sub-megahertz frequencies.
Using PCB Traces as Components - Capacitors

A capacitor is formed by two electrically conductive surfaces parallel to each other. Formula for capacitance is

\[ C = \varepsilon_r \varepsilon_0 \frac{A}{d}, \]

where

- \( C \) is the capacitance, in Farads;
- \( A \) is the area of overlap of the two plates, in square meters;
- \( \varepsilon_r \) is the relative static permittivity (sometimes called the dielectric constant) of the material between the plates (for a vacuum, \( \varepsilon_r = 1 \));
- \( \varepsilon_0 \) is the electric constant (\( \varepsilon_0 \approx 8.854 \times 10^{-12} \text{ F m}^{-1} \)); and
- \( d \) is the separation between the plates, in meters;


Capacitor could be formed in two separate ways: On same layer, thin and long traces would be “combed” to create a capacitor. Exact analysis of capacitance is quite hard, since between the traces is some amount of solder mask and some amount of air. Let us assume for the sake of simplicity:

1) Copper has uniform 35 um height.
2) Half of height is air, which has relative permittivity of vacuum (close enough)
3) Half of height is solder mask, which has relative permittivity of 3.3 [https://www.sigcon.com/Pubs/edn/PassivationandSolderMask.htm]
4) Combined effect can be modeled as material with relative permittivity of \((1+3.3) / 2 = 2.1\)

If the traces are 0.15 mm apart and parallel each other for 9.5 mm, capacitance between two traces would be

\[ 2.2 \times 8.854 \frac{pF}{m} \times \frac{350 \mu m \times 9500 \mu m}{150 \mu m} \approx 0.432 \text{ pF}. \]

It would be very hard to get useful capacitances created this way.

Another way to create capacitor would be to use different layers. On a 4-layer board isolation between two layers could be as small as 200 um.

We assume relative permeability of 4.8 [http://en.wikipedia.org/wiki/FR-4].

On a 10 mm * 10 mm space this would create

\[ 4.8 \times 8.854 \frac{pF}{m} \times \frac{10 mm \times 10 mm}{0.2 mm} \approx 21 \text{ pF}. \]

Buried capacitance is used in high speed designs at several hundreds of megahertz, but replacing
discrete capacitors with PCB structures does not seem practical.
Using PCB Traces as Components – Inductors

Inductance is property of conductors which “resists” change of current by inducing a voltage opposing change of current both in conductor itself and nearby other conductors. PCB traces can be used to create a planar inductor.

Single layer PCB trace inductors have been extensively studied, (http://www-smirc.stanford.edu/papers/JSSC99OCT-mohan.pdf). There also exists an online calculator (http://www.circuits.dk/calculator_planar_coil_inductor.htm) which inserts values into formulas and gives out the results.

Multilayer spiral inductors use same design as single layer inductors and the inductors in each layer are connected in series. In addition to summing of inductance because of series connection, these layers exhibit mutual inductance because of property of inducing voltages into nearby conductors. EDN has published an article about designing multilayer spiral inductors: (http://www.edn.com/design/components-and-packaging/4363548/A-new-calculation-for-designing-multilayer-planar-spiral-inductors). Article reports that 10 uH is reachable on 4-layer PCB in 9 mm * 9 mm area.

Recent design of ours uses a discrete 3.3 uH inductor as SMPS filter inductor. Let’s see if we could replace it with PCB trace!

Inductor is required to carry at least 350 mA DC current, so let’s design for 500 mA capacity for some safety marginal. Inner layers of PCB have 17 um copper thickness. Quick testing with Saturn PCB calculator (http://www.saturnpcb.com/pcb_toolkit.htm) reveals that a 0.5 mm thick trace in inner layer can carry 0.6 A of current with up to 14 parallel traces when temperature rise of 45 degrees celsius is allowed. Of course this might be a misleading figure since there will be conductors both above and below, so there isn’t really anywhere where to heat could conduct from inner layers in the middle of inductor.

First we select 12 mm * 12 mm PCB area and 150 um isolation between traces and trace width of 500 um. Then we select 7 as number of turns. This leads to inner diameter of 3200 um and fill ratio of ~0.58. I wrote a quick python script to calculate inductance of single, two and four layer PCBs. According to script, this inductor would have inductance of ~3.1 uH, which is enough for our purposes!

But how about other parasitic properties? We can estimate trace length to equal 4 sides * average diameter * number of turns, or \(4 \times \frac{12 \text{ mm} + 3.2 \text{ mm}}{2} \times 7 \approx 213 \text{ mm}\).

Outer layers have copper thickness of 35 um, according to Saturn PCB calculator trace resistance would be ~0.3 ohms. Inner layers with half of the copper thickness have double resistance. In total the resistance would be ~0.3+0.6+0.6+0.3 ohms or ~2 ohms in total. Power and voltage loss over such a resistance is a lot higher than over a discrete inductor.

Planar cores which allow to have high inductance with smaller number of turns do exist. A magnetic core would allow wider traces for same inductance value, which in turn leads to decreased resistance and increased current handling capacity. However, using such a core would defeat the
purpose of decreasing component count on design.

However, for the sake of scientific curiosity we might actually produce a few boards to test the values given by script. Even though coreless planar inductors are not useful for SMPS filtering, they might have applications in signal conditioning, especially if inductance values have precise values inside same batch of PCBs.
Using PCB Traces as Components – Resistors

In this part of PCB trace component series we investigate possibility of using PCB trace as a resistor. Since copper is a good conductor, maybe PCB trace could practically replace current sense resistors in design? PCB trace resistors have also been used as a heating element in 3D-printer heated beds. As the resistance changes with temperature, temperature of the heating element could be estimated with the resistance of element.

Let's continue with our previous SMPS example: we have a nominal 350 mA of current, and we want to have a ~50% margin. Sense resistor should be able to carry 500 mA of current and it should produce a reasonable voltage for ADC over that current range. Using a current sense amplifier is an option.

STM32 has VDD of 3.3 volts which is used as reference to ADC. 500 mA should produce a voltage signal of 3 V. Using a 6 ohm resistor would wasteful, so let's select a current amplifier. MAX4376 [http://datasheets.maximintegrated.com/en/ds/MAX4376-MAX4378.pdf] would seem to be good starting point. It's reasonably priced and has fixed internal gain of 20/50/100 which offers flexibility in selecting the resistance value.

If we start with the gain of 50, full range sense voltage now becomes 3 V / 50 = 60 mV. To have 60 mV voltage over sense resistor at 500 mA of current, resistance must be 0.030 ohms. In inner layers with 17 um copper thickness a trace with 0.2 mm width and 5 mm length would suit this need. If the trace was connected by large vias trace probably could serve as sense resistor. On outer layers where copper thickness is 35 um, sense amplifier with gain of 100 could be used to scale voltage into useful range.

As with the inductors, interesting question is the tolerance of embedded resistor, especially inside same batch of PCBs. Exact value of resistance isn't of big importance, it can be measured and measurement can be used as a calibration value if boards from same batch have reasonably little variance.