Keywords: printed circuit boards, PCBs, active and passive components

APPLICATION NOTE 5800

PASSIVE COMPONENTS AREN'T REALLY SO PASSIVE (PART 3): PRINTED CIRCUIT BOARDS

By: Bill Laumeister, Strategic Applications Engineer

Abstract: Active components like transistors and integrated circuits change signals using energy from the power supply. However, passive components like resistors, capacitors, inductors, and connectors actually can, and do, change the signal in unexpected ways. This happens because all these passive components contain parasitic components. This application note, the first in a 3-part series, discusses printed circuit boards and the errors that can occur because passive components aren't really so passive.

A similar version of this article appeared January 9, 2014 in *Electronic Design*.

Introduction

Sometimes the best way to hide something is in plain view. Magicians use this technique along with some distraction to amaze an audience (**Figure 1**). It is simple actually: our experience leads us to expect certain norms and to see what we expect. Thus boxes are square, not squished parallelograms; spheres are symmetrical, not hemispheres or with elongated portions on the back where it is unseen. In that same sense, printed circuit boards (PCBs) seem straightforward. You think that you can see everything going on, but you are really only looking at the circuitry on the exterior surface. In fact, if you delve deep enough down to the board itself, you find complex layers and structures and a myriad of things that can go wrong here. When high-precision op amps and high-resolution data converters fail to perform as expected, we need to closely examine all the surrounding active and passive components, including the PCB. Into this context we also insert the PCB vendor who has an understated role that is, in fact, critical for IC performance.

This article is Part 3 of a series on passive components in ICs. In Part 1 we talked about capacitors. In Part 2 we looked at resistors and explained that they are not seemingly simple, benign, passive devices. Here in Part 3 we are going to discuss how PCB flaws and errors that are usually hidden, or at least disguised, can introduce passive errors into IC performance.

To understand how PCBs can introduce passive errors, we must first examine the composition of a typical board. Four examples of PCB problems and efforts to solve those hidden errors will help us appreciate the contribution that a good reliable PCB vendor makes to successful products.

We admit here that our articles on passives have generated some lively discussion about the definition of "passive." In our search for more knowledge and better-informed engineers, we are quite pleased about this. See our **Sidebar: Defining Passives** for some summary comments on this discussion.

Figure 1. A magician and his assistant provide distractions to help "sell" illusions.

Passive Viewing—Seeing What We Expect

Let's see how well, how carefully we viewed Figure 1. Did you notice the PCB assembly? Yes? No? It is in the shadows just to the woman's left side. Yes, we see what we expect to see. The same is true when we examine a PCB. When you look at a typical board directly (**Figure 2**), what do you see?

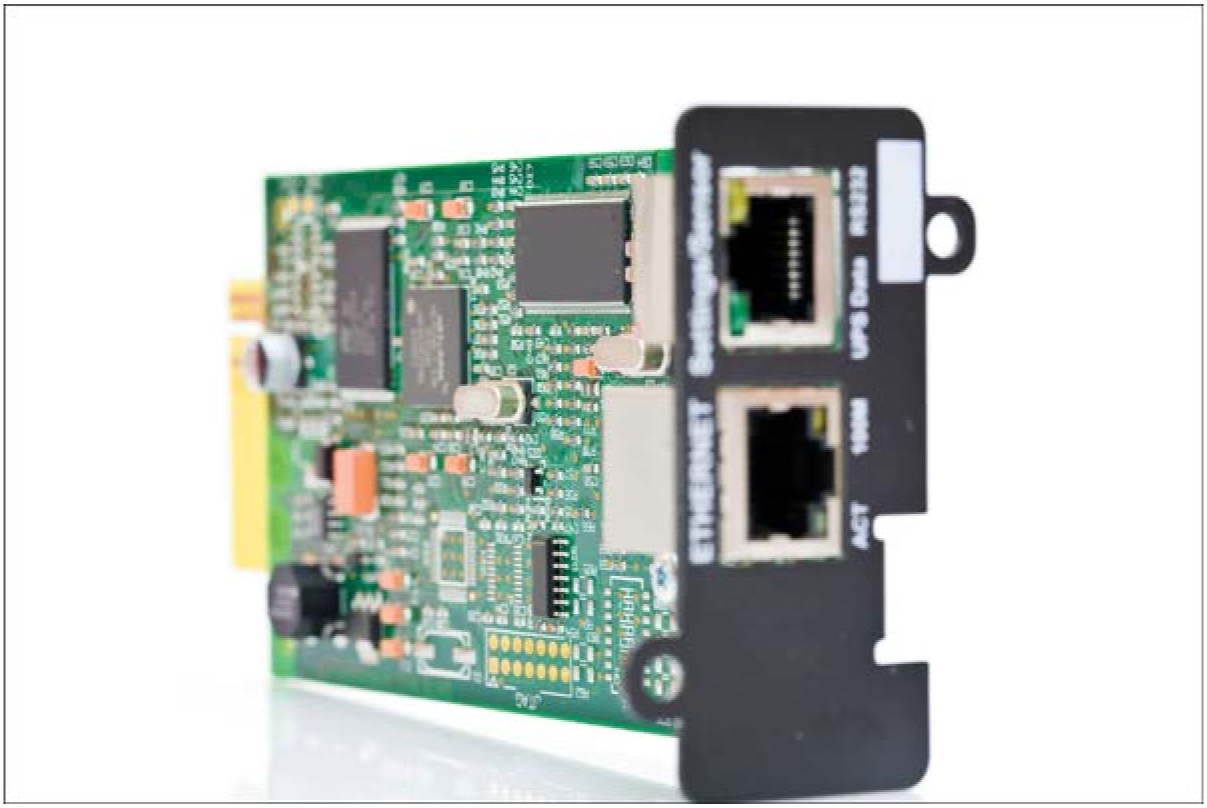

Figure 2. A PCB assembly with various components.

If you are like most of us, we see an Ethernet connector, another RJ-45 connector with the label “settings sensor”, “UPS data”, and “RS-232”. We see an inductor and electrolytic capacitors for a switching supply, several large-scale integrated circuits (ICs) and a bunch of decoupling capacitors. Put all this together and it is probably a digital board with several options because we can also see unstuffed components. Right? Yes, but we did not really see the bare PCB itself, and that is where this story starts.

As we said at the outset, myriad things can go wrong with something as complex as a PCB. Experience has taught us that a good, reliable PCB vendor is very important to us now. There are many choices in the materials, the density of the weave in the FR4, the polymer, via construction, minimum trace structure for a given etch method, tin plate and solder mask choices. We might specify a hard-to-find FR4 (a common fiberglass PCB) material because we prefer it, but lack of available FR4 materials could delay production and even double the board cost. Our respected PCB vendor will know about resources, what via construction methods are available, or what assembly methods are recommended for our application. There is definitely nothing passive about this relationship. When we tell the vendor that we care about board quality, he will reciprocate in like manner.

No Magic Wand Building a Board

Yes, the board—you start with fiberglass. The top and bottom layers (typically industry-type FR4) have copper on what will become the outside of the PCB. The center layer is copper with FR4 on both sides, thus comprising the two inside conductive layers. Prepreg is effectively the glue that holds the stack together; it can be just adhesive or it can be a combination of FR4 fiberglass and thermal-setting adhesive. During the fabrication process the stack in **Figure 3** will be compressed under heat and pressure to bond the layers together.

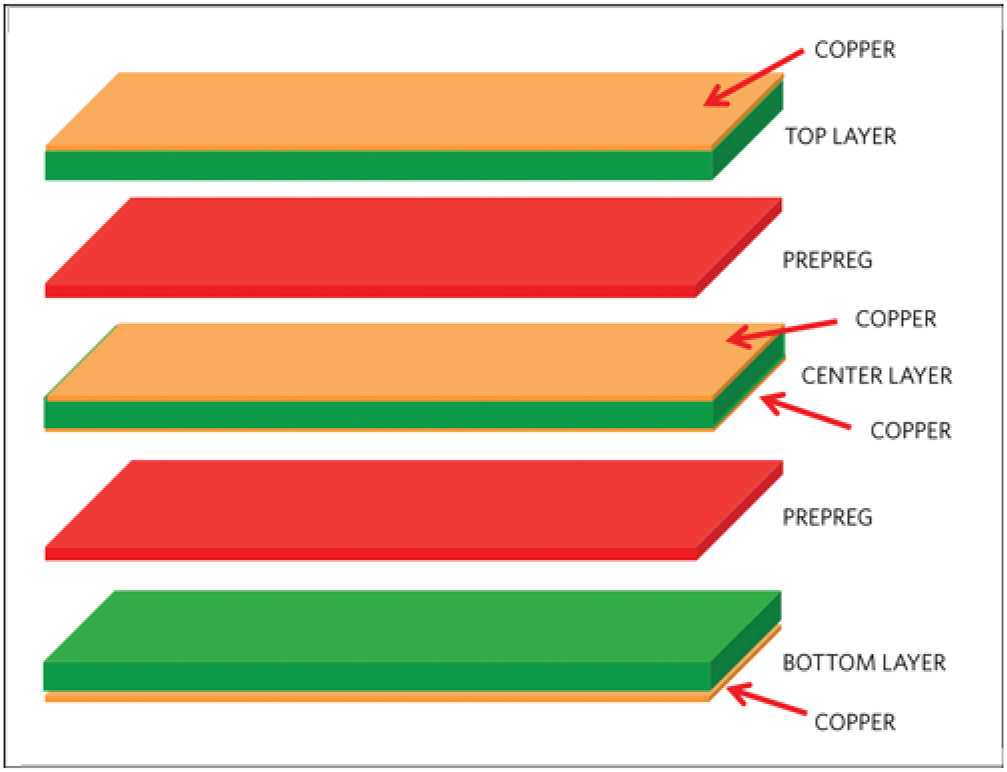

Figure 3. Is a typical four-layer PCB stackup.

The order of construction can differ depending on many things. Our favorite reference resource, the handbook that most engineers call the "PC Bible," is by Coombs.[1] He details the PCB fabrication processes, outlining literally hundreds of variations and possibilities. Just when you are thoroughly intimidated, you get to the Appendix. The knowledge in the Appendix is massive, a list of industry standards pertaining to everything PCB. It takes you from components including surface mount, general and passives, to printed boards, materials, design activities, then to component mounting and soldering, and through quality assessment, test methods, and repair. At this point we begin to appreciate and understand why we need our best board vendor to guide and advise us.

Still, mistakes do happen with boards, and it seems that they always occur just before a firm deadline. The four PCB examples below happened either before a bed-of-nails test of the bare board was available or after that test was eliminated to save time—always a bad practice that will punish us. Can you guess the errors that we found in each example?

Example 1: Over-etching

We received PCBs and assembled six boards. Oddly, the boards all had different issues. Normally when you fix one board that same fix applies to all the boards. But not this time, which was the key to understanding the problem.

We found that some of the errors were tiny shards of copper that shorted random things. Simultaneously, we were seeing a massive “passive” problem (at least we usually think that a PCB is passive) in the circuit’s performance. No circuit can function with dozens of random shorts. Because these shorts were random and different on each board, it was a troubleshooting nightmare. We sectioned the PCB and looked under a microscope. The board was over-etched, as shown in **Figure 4**.

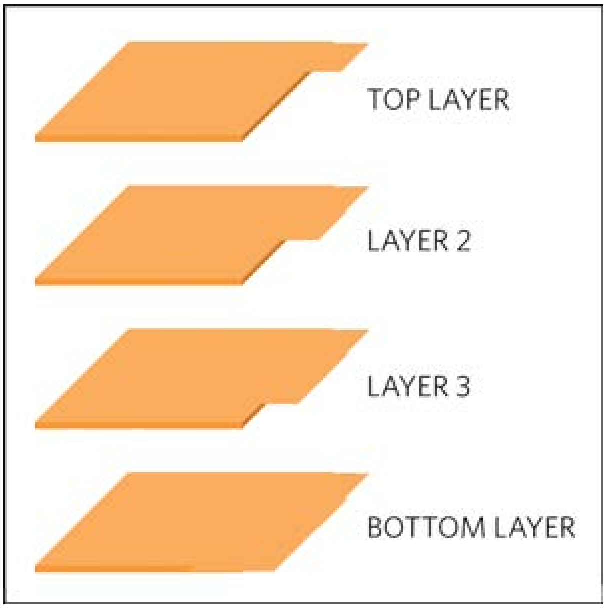

Figure 4. PCB section with over-etched, thin copper edges that break off as long thin shards and short to adjacent traces.

Figure 4A has flat sides under photo resist. If the chemistry and temperature are not correct or the board is in the etching solution too long, the effect is etching “around the corner” that undercuts the copper (Figure 4B). Long thin shards can break off the top edge, stay connected on one end, and short to the adjacent traces.

Looking closely at our board, we saw two abrasive scratch-mark patterns at 90 degree angles. The vendor had used a polymer grinding wheel with an embedded abrasive. They attempted to scrub off the shards by grinding the board in two passes on each side. They did remove the majority of the shards, but then they solder-coated the board, which made the remaining shards solid random shorts. Adding solder mask hid the shorts and most grind marks.

Example 2: Orientation

We received a two-sided PCB with solder mask and top silkscreen and assembled a board by hand with through-hole parts. Nothing worked. We had a serious problem with the so-called passive PCB: all three power supplies were shorted in multiple places. Nothing made any sense; not one circuit block out of dozens functioned at all. The technician tried, but finally called the engineer for help.

The technician managed to insert the parts in some strange ways. For example, the three leads of a transistor, which would normally form a triangle in the silkscreen outline, were distorted and twisted. Looking closely under the solder mask we realized that the silkscreen and the bottom side of the board were oriented properly, but the top component copper side of the board was a mirror image. The film used to make the topside image was upside down when the solder resist was exposed.

Example 3: Find Your Way
We received a four-layer board as in Example 2 above with similar issues. Again many traces were connected to the wrong things, power supplies were shorted at multiple places, and nothing (no circuit block) worked. Usually when there is a board error, at least some of the circuits function. We had implemented a complete bed-of-nails test and were confused when it did not identify the issues. Then we found out that the purchasing department skipped the bed-of-nails test to expedite board delivery. That test would have saved us days of effort. The wasted time was a costly error.

We found that the board layers were assembled in the wrong order. Many blind vias were attached to the wrong layers. As a result, we added an edge code (**Figure 5**) so we could inspect the boards.

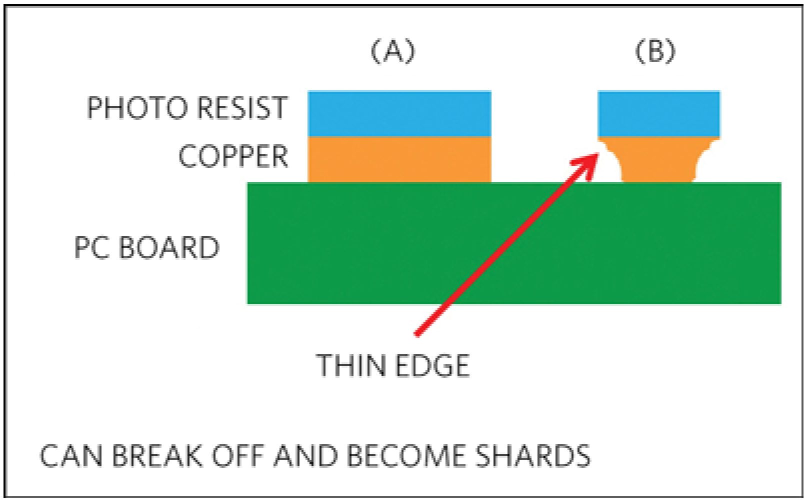

Figure 5. The copper layers of the PCB with a staggered edge code on the right. With the code implemented, we could quickly inspect the board layer order before we wasted time by populating the board with components.

The code of Figure 5 extends to the edge of the PCB. The boards are typically fabricated in larger panels made of many smaller PCBs to ease handling during fabrication. The individual boards are separated using a router, thereby exposing the Figure 5 code on the board's edge. A microscope lets us measure the copper spacing and see that it meets the board specification. This assures us that the stripline will be the correct impedance.

Example 4: the Right Thickness, but Not the Right Answer
We received a four-layer board. Most of it worked, but the striplines had huge ringing and reflections. Striplines are the equivalent of coaxial cables embedded into the PCB. A coax is a center conductor inside an insulating dielectric, surrounded with a circular ground shield. In addition to shielding the signal from external contamination, the coax and the stripline provide a known impedance signal path that, when terminated in its characteristic impedance, does not reflect energy. If the PCB is not constructed properly, the impedance change causes reflections and ringing which destroys analog signals and can even confuse digital signals.

Sectioning the board permitted us to measure the thickness of the various layers. We found that the vendor had a shortage of some thicknesses of the board material. Their untrained employee tried to meet our delivery deadline and substituted extra prepreg layers from something in stock, thus making the total thickness correct. This might sound like a good “fix,” but it definitely was not so. Look back at Figure 4. Let’s say that the center layer with copper on both sides was substituted with thinner material. The capacitance between those two layers will rise because the dielectric is thinner. To keep the layout and the final board the same total thickness, we can compensate by increasing the upper prepreg layer thickness. This will lower the capacitance between the top copper and the nearest copper layer in the center. Note, however, that this also assumes that the prepreg has the same dielectric constant in both cases, which may not be true. Thus, the change in capacitance changes the PCB and stripline impedance, and our supposedly “passive“ PCB is now ringing. You can say “Good bye” to signal integrity.

PCB Problems Cause Passive Failures

Clearly an unseen, taken-for-granted, whatever-you-want-to-call-it PCB exerts considerable influence on precision circuit performance. Moreover, we cannot take anything for granted nor assume that passive IC problems are unrelated to the PCB itself. Common IC performance problems and errors caused by a bad PCB include voltage drops and impedance in ground vias, planes or foils; leakage resistances and moisture absorption; and stray capacitance, with welcomed and dielectric absorption or soakage.

Voltage Drop

Voltage drop in ground vias, planes or foils is a commonly overlooked issue. Adding to the complexity of the problem, voltage drops at both DC and high frequencies require different remedies. Recall Coombs Handbook,[2] Chapter 10 for trace versus capacitance and crosstalk and Chapter 13 for voltage and ground thickness versus sheet resistance. For via impedance we look to Sayre:[3]

$$L = 5.08h\,[\ln(4h/d) + 1]$$

Where:
L = inductance of the via, nH
h = length of the via, inches
d = diameter of the via, inches

Using h = 0.0625 inch and d = 0.020 inch gives us a via inductance of 0.666µH. How can we reduce this inductance? Place two, four, or more vias in parallel.

This is a good first-order approximation and is useful in thinking about signal integrity below a few hundred megahertz. For more details and consideration of the current return paths, we turn to Howard W. Johnson and his “Black Magic” series.[4]

Leakage Resistance

The leakage resistance[5] of the PCB can disturb sensitive high-impedance circuits. The sources of leakage include improper selection of a laminate material, fingerprints, skin oils, human breath, residual fabrication chemicals, improperly cleaned solder flux, and surface moisture and humidity. If this is a problem for your circuit, consider surface and subsurface contamination and moisture absorption to be everywhere, on, in, or under the solder mask; on, in, or under conformal coatings; on or in active or passive components.

When troubleshooting an existing PCB, remember an experienced engineer blowing on the board through a soda straw. The straw localizes the moisture to help identify the sensitive area. Thorough board cleaning with the proper solvents is important. The wrong solvent, for example, cleaning a water-soluble flux with a polar solvent, can leave salts on the board. If deionized water is used to clean the boards, bake the boards to dry them. Even now you may not be done. Even the cleanest board may still cause problems. A PCB with a very sensitive circuit such as an op amp with a high impedance input and high gain likely needs additional attention. It might be necessary to guard or surround the sensitive pins on all board layers with a driven low-impedance circuit matching the DC level of the guarded pins.[6]

Stray Capacitance

Capacitance is usually a problem when it is stray and unavoidable. It reduces bandwidth and slows high-speed signals. It is bad when dielectric absorption or soakage[7] causes hooking, slew-rate errors, or under-/overshoot. However, capacitance is welcome when it is high-frequency power decoupling. We can specify a thinner than normal dielectric (even thin FR4) between the power and ground planes to increase the capacitance. Discrete capacitors smaller than 10pF (self-resonant at ~2GHz in surface mount) are easily compromised by trace and via inductance. Where the capacitance is distributed between power and ground planes, it has low series inductance and is repeatable if, yes, if we have a "golden" excellent PCB vendor.

Summary

Let's think back to our opening magician's mysterious box with hidden tricks. We expect certain norms and see what we expect. We simply cannot be that blind when it comes to the potential problems in a PCB. The manufacture and assembly of a board is far more complex than it appears to a casual examiner and in that complexity lies the potential for PCB flaws and errors. Now, most importantly for our discussion, those flaws and errors can introduce passive errors in ICs. We only examined voltage drop, leakage current, and stray capacitance, but the list of potential passive errors is indeed longer. Solving these passive problems inevitably means fixing the PCB, and each situation will demand its own solution. Finally, within this context we can all appreciate the contribution that a good reliable PCB vendor makes to our successful products.

References

1. Coombs, Clyde F. Jr., *Printed Circuits Handbook*, sixth edition, McGraw Hill, ©2008.
2. Coombs, *op cit.*, Chapters 10 and 13.
3. Sayre, Cotter, *Complete Wireless Design*, McGraw-Hill, IBSN 978-0-07-154452-8, page 421 in first edition.
4. Johnson, Howard, *High Speed Digital Design: A Handbook of Black Magic and High Speed Signal Propagation: Advanced Black Magic and Via Inductance*
5. Maxim Integrated tutorial 4429, "Murphy's Law and the Risks of Designing 'Off Data Sheet,'" Figure 3 and the following text.
6. Maxim Integrated application note 3264, "Compact DWDM laser Temperature Control with the MAX8521," Figure 2 and surrounding text.
7. Laumeister, Bill, *Passives Aren't Really So Passive: Part 1, Capacitors*, **Electronic Design**, June 4, 2013 article.

Sidebar: Defining Passives

When we started talking about "passives", we stirred up a hornet's nest? Several engineers[1] inside Maxim Integrated and in the larger engineering community immediately challenged the definition of "passive." We are still trying to find a short, accurate definition that is universally accepted. The most common definition is simply “not active.” Thus, a typical active device uses power to do something like create gain. But there are always exceptions. For example, an emitter follower is active, uses power, converts impedance, and has a gain just less than unity. The goal of these articles on passives, therefore, has been to warn people that what we think is a passive, can and does cause nonlinear responses that can change the signal. Thus resistor voltage dependence or capacitive absorption (soakage) can cause harmonic distortion. Hydroscopic PCBs can change offset.

How does one define a passive component? It is also a tough question. Engineers in a chat room had some good suggestions. The IEEE® dictionary[2] defines:

> Passive device, A device that does not require power and contains no active components.
> Passive Electric Network, An electric network containing no source of energy.

Davor Vujatovic in the Encyclopedia of Life Support Systems (EOLSS) suggests a passive definition:[3]

> A passive component denotes a component that is unable to deliver more energy to an external circuit than it initially stores. To determine whether the component is passive, the total energy absorbed by it must be greater or equal to zero. In other words, a component that absorbs more energy than it delivers is passive. If the total energy delivered by the component is greater than the total absorbed energy, the component is active, i.e. the active component is capable of delivering energy to the outside world.

The chat room engineers also suggested the Wikipedia entry under, “Passivity (engineering)”.[4] It has an interesting perspective in the first two paragraphs:

> “Passivity is a property of engineering systems, used in a variety of engineering disciplines, but most commonly found in analog electronics and control systems. A passive component, depending on field, may be either a component that consumes (but does not produce) energy (thermodynamic passivity), or a component that is incapable of power gain (incremental passivity).
>
> A component that is not passive is called an active component. An electronic circuit consisting entirely of passive components is called a passive circuit (and has the same properties as a passive component). Used without a qualifier, the term passive is ambiguous. Typically, analog designers use this term to refer to incrementally passive components and systems, while control systems engineers will use this to refer to thermodynamically passive ones.”

Then depending on one’s engineering discipline; Wikipedia says,

> Thermodynamic passivity
> In control systems and circuit network theory, a passive component or circuit is one that consumes energy, but does not produce energy. Under this methodology, voltage and current sources are considered active, while resistors, capacitors, inductors, transistors, tunnel diodes, glow tubes, metamaterials and other dissipative and energy-neutral components are considered passive.”
>
> Incremental passivity…In circuit design, informally, passive components refer to ones that are not capable of power gain; this means they cannot amplify signals. Under this definition, passive components include capacitors, inductors, resistors, diodes, transformers, voltage sources, and current sources. They exclude devices like transistors, vacuum tubes, relays, tunnel diodes, and glow tubes.”

The Wikipedia article really sums it up in the second paragraph: “Used without a qualifier, the term passive is ambiguous.”

We included the wording “seems to be passive” in our article definition in an effort to “weasel word” the definition to allow nonlinear distortion from something that we expect to be “inert” or “benign.” “Seems“ used above drew lightning for the engineers, so now adding inert or benign will probably add more fuel to the fire. We are still trying to find a short, accurate definition of “passive” that is universally accepted. The most common definition is “not active” and it is not sounding so bad after all.

References

1. www.electro-tech-online.com/threads/capacitors-more-active-than-you-think.135643/#post-1137754
2. The IEEE Standard Dictionary of Electrical and Electronics Terms, sixth edition, IEEE Std. 100-1996; terms are ”Passive device” and ”passive electric network.”
3. Vujatovic, Davor, Electronic Engineering Vol I –Active Networks, ©2009 Encyclopedia of Life Support Systems, EOLSS Publishers Co Ltd, ISBN-13: 978-1848269774).
4. Wikipedia: http://en.wikipedia.org/wiki/Passivity_(engineering)

IEEE is a registered service mark of the Institute of Electrical and Electronics Engineers, Inc.

Related Parts		
MAX44244	36V, Precision, Low-Power, 90µA, Single/Quad/Dual Op Amps	Free Samples
MAX44245	36V, Precision, Low-Power, 90µA, Single/Quad/Dual Op Amps	
MAX44246	36V, Low-Noise, Precision, Single/Quad/Dual Op Amps	Free Samples
MAX44248	36V, Precision, Low-Power, 90µA, Single/Quad/Dual Op Amps	Free Samples
MAX44280	1.8V, 50MHz, Low-Offset, Low-Power, Rail-to-Rail I/O Op Amp	Free Samples
MAX44281	Ultra-Small, Ultra-Thin, 4-Bump Op Amp	Free Samples

More Information
For Technical Support: http://www.maximintegrated.com/en/support
For Samples: http://www.maximintegrated.com/en/samples

Other Questions and Comments: http://www.maximintegrated.com/en/contact

---

Application Note 5800: http://www.maximintegrated.com/en/an5800
APPLICATION NOTE 5800, AN5800, AN 5800, APP5800, Appnote5800, Appnote 5800
© 2014 Maxim Integrated Products, Inc.
The content on this webpage is protected by copyright laws of the United States and of foreign countries.
For requests to copy this content, contact us.
Additional Legal Notices: http://www.maximintegrated.com/en/legal