Altium

Design for Manufacturing

DESIGN FOR MANUFACTURING

When it comes to manufacturing, organization is critical. With so many different players involved in the design process (engineers, sourcing agents, manufacturers, etc.), design for manufacturing entails a level of detail that demands continuous communication and information exchange with the different parties involved.

Join us as we discuss a variety of topics to help you with Design for Manufacturing, including:

- Bill of Materials Management Strategies: Automation is Your Friend
- Step Up Your Game: Multi-Board Versus Multilayer Designs
- How To Test Electronic Components During PCB Design
- Properties Of Semiconductors- Electrodeposition And PCB Devices
- The Advantages And Disadvantages Of Designing With Breadboards

BILL OF MATERIALS MANAGEMENT STRATEGIES: AUTOMATION IS YOUR FRIEND

When I was a boy I really enjoyed reading the "Alfred Hitchcock and the Three Investigators" mystery books. These books were a contemporary to "The Hardy Boys" mystery books and I filled a lot of my free time reading about "The Secret of Terror Castle", "The Mystery of the Screaming Clock", and many other frightful enigmas. A few years ago I ran across some of these books, and now I collect as many as I can find.

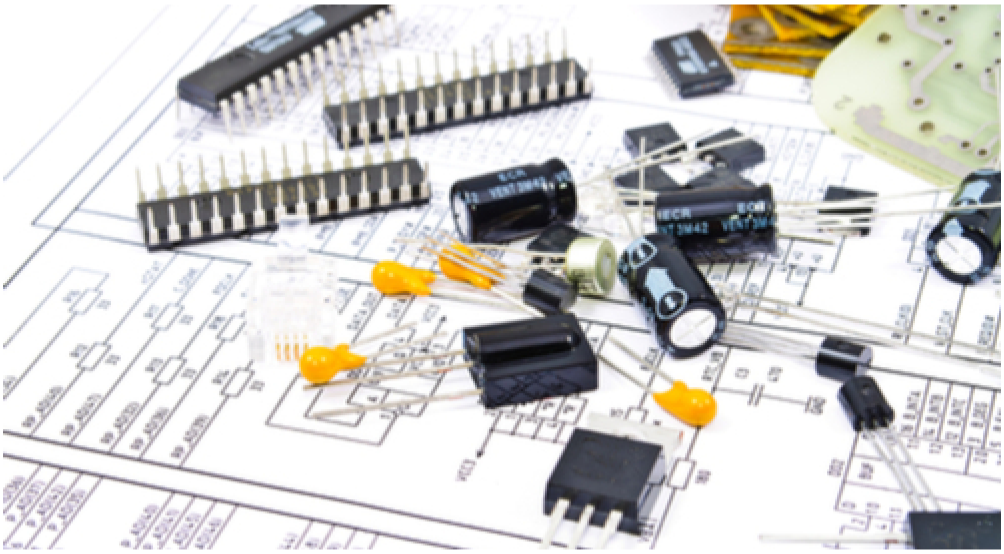
Another avid fan I know said it well when he said; "finding these books is like being reacquainted with an old friend." As a boy, I trusted these books to make me part of their marvelous world. Believe it or not, I've developed this same trust with my PCB design tools, especially the ones I use less frequently but still trust immensely. This trust is that they will get me through what I need to accomplish. And for better or worse, the tools have gotten me through quite a bit.

One aspect of my PCB design activities that was something less than a trusted friend was the process of creating a bill of materials (BOM). This used to be a frustrating manual process that often was plagued by discrepancies and errors. Now with bill of materials management tools, it has become something that I can trust again due to the automation that I am able to use. With automation, my BOM process has earned itself my trust again.

THE NOT SO FRIENDLY MANAGEMENT PROCESS OF CREATING A BOM MANUALLY

It used to be that the PCB bill of materials was nothing more than a text file that was generated from the CAD tools. In order for this document to become useful to the other departments, it had to be manually verified and corrected. It would also have to have any additional parts not represented in the CAD database added in as well. After that, it had to be reformatted per the needs of whoever was going to be using it and then distributed.

Because this was a manual process, every time the design was changed we would have to go through the same steps to make sure that the BOM was correct. If someone didn't receive their messages that day they might miss out on the latest update and continue working from an outdated BOM. There was also the very real possibility that with all of the manual changes that were made to the document that it might get out of sync with the actual design database.

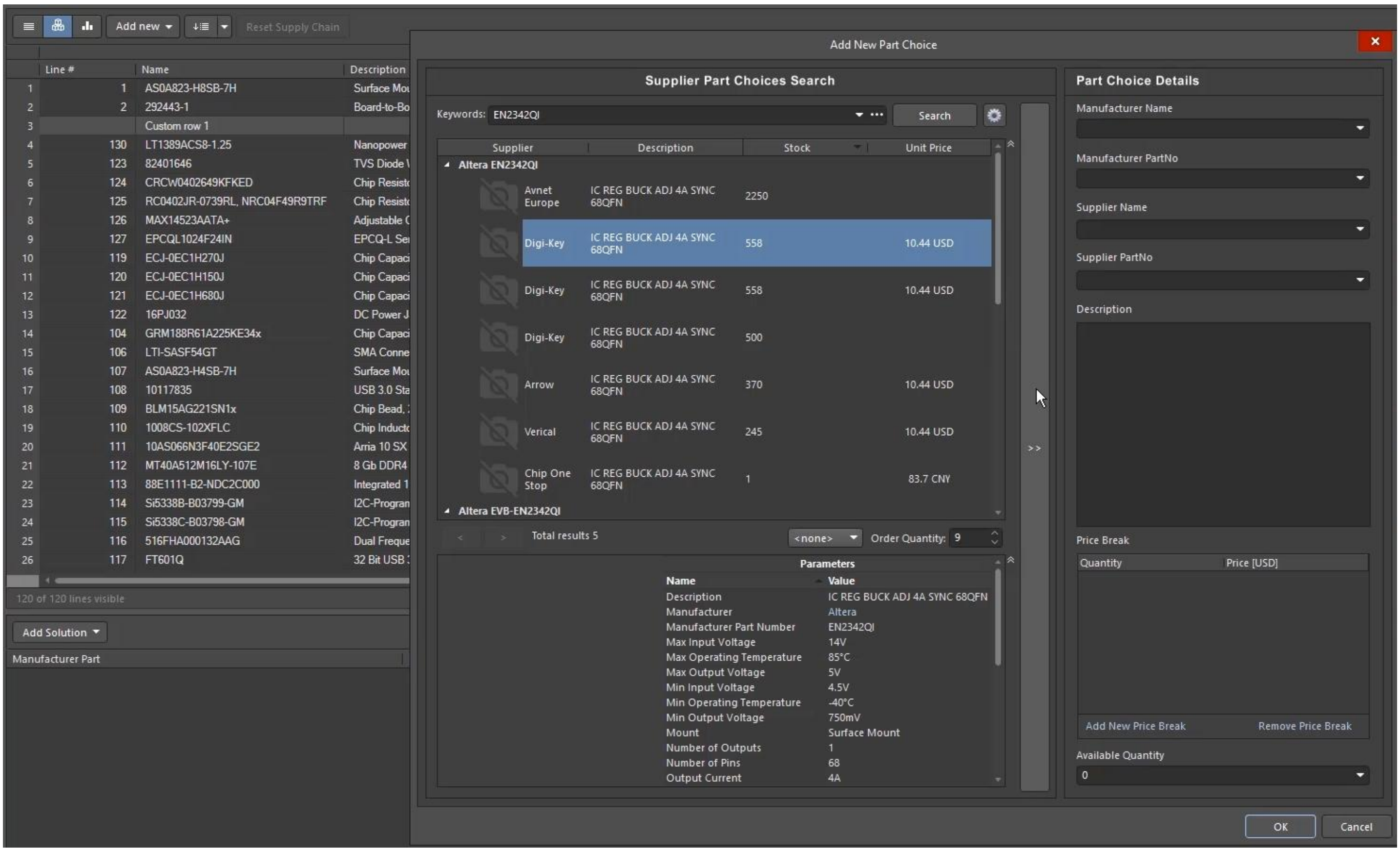


Creating and recreating a PCB BOM in order to capture design changes can lead to errors

THE PCB BOM CREATION PROCESS THAT WAS DESIRED

What I was looking for was a PCB BOM whose creation and update processes was something that I could trust. I wanted a BOM that was more than a simple text document and that would work globally for all departments. I also wanted a BOM that was available during all phases of the design instead of it only being available after the layout was completed. It would be really advantageous, for instance, to query parts from the library while still working in the schematic.

Most of all I wanted the BOM to be something that I could work with that was part of the overall design process instead of it being just an output file. I wanted to be able to interact and work with the BOM entries the same way that I work with parts on the schematic. And on top of all of that, I wanted to be able to do all of this easily without having to jump through a lot of different hoops to get there. I know, it seemed like I was asking for a lot.



Bill of materials management from the design tools will give you more control over your design

BILL OF MATERIALS MANAGEMENT AUTOMATION THAT GAVE ME WHAT I WANTED

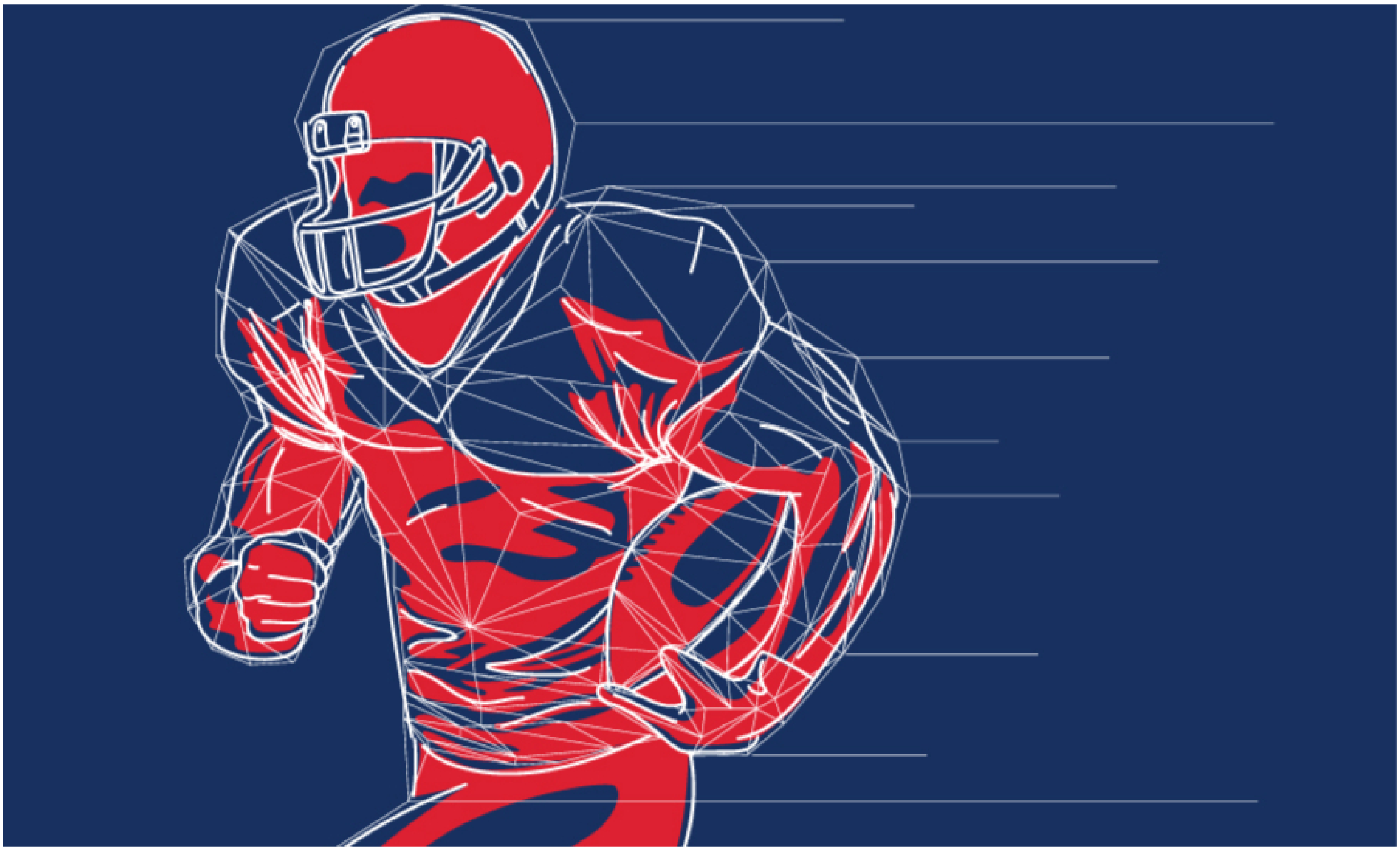
Fortunately, I have found what I was looking for. The BOM management tools that I am now using have given me the level of automation that I was hoping for. The days of manually compiling a BOM are gone. Instead, the BOM data is managed throughout the entire design as another part of the design toolset instead of it being a simple output file. This allows the BOM to interact with every other design aspect from schematic to layout to manufacturing drawings.

The BOM management tools that I am using have given me the ability to select parts within it to highlight the same parts in either the schematic or the layout. Not only has this been very helpful to me during design, but in design reviews as well. I am also able to synchronize accurate BOM information with other departments.

The management tools allow me to research components for detailed part information and then pass that data on through the BOM to purchasing and manufacturing. This has helped us to make sure that the correct parts are ordered and ready for the assembly line. Finding user-friendly automation that we could trust to solve problems in our manual BOM creation has been a major relief. Software like Altium Designer's Active BOM may be the answer that you are looking for too.

If that's the case and you're interested in how finding software with tools you can trust will help your design process, find out more information by talking to an expert at Altium.

STEP UP YOUR GAME: MULTI-BOARD VERSUS MULTILAYER DESIGNS



I don't remember much about playing football in high school. I'm not sure if that's because of wanting to forget the endless grueling practices to prepare us for getting pummeled in the game, or actually getting pummeled in the game. Either way, I tend to vaguely associate high school football with lots of hard work and pain, punctuated by the occasional moments of glory. If there is one thing that I do remember very vividly, it was the coaches encouraging us to always "step up our game".

Our coaches knew that the goal wasn't a matter of arriving at a plateau, but instead constantly advancing to the next level. To prepare us, therefore, for whatever was coming next, they pounded "step up your game" into our heads every moment that they could, and I'm actually grateful for that. It helped me through my teenage year and continues to help me today. In my career of designing circuit boards, I have had to step up my game many times. I've seen advances from thru-hole technology to surface mount, and from wide open designs to high density interconnect (HDI) designs. Now the next step has arrived: moving up from designing single multilayer boards to designing multi-board systems.

Those of us in the design world are familiar with designing a single printed circuit board, but designing multiple boards for a system is probably something new to many of us. Fortunately, there are new and enhanced PCB design CAD tools that can help with this task. Whereas designing PCBs for a multi-board system used to be a clumsy manual process, these new tools provide a design solution that is eloquent and actually fun to use. Keep reading and you'll see what I mean.

Today's PCB designs often require designing all of the system boards together

REMEMBERING THE OLD WAYS

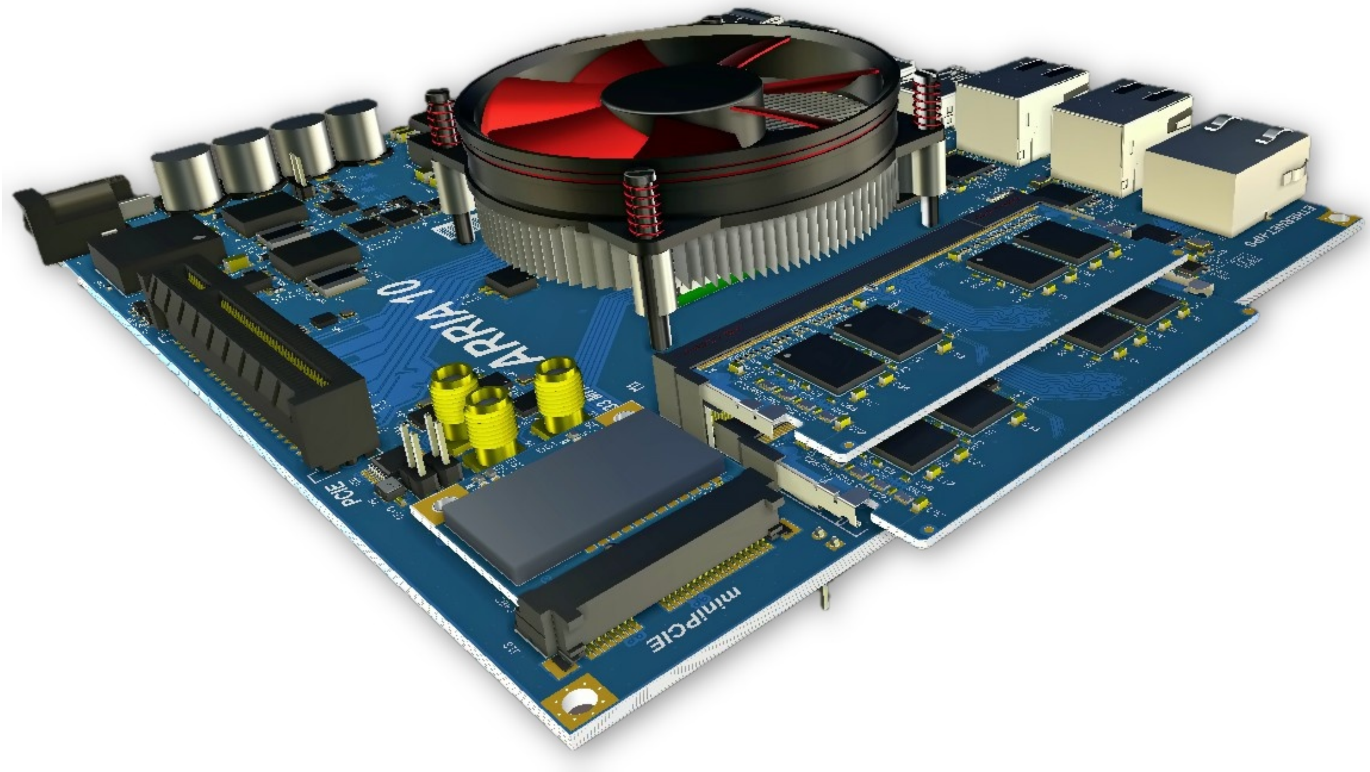
System level design is nothing new, but how we design the different circuit boards that make up a system is. For years, PCB design was limited to designing one board at a time as there wasn't a CAD solution that would design multiple system boards together. The mechanical design of the system would dictate the size and shapes of the different circuit boards that would fit into it, but the boards would be designed individually.

After individual PCBs were designed, they would be built as prototypes. Once these prototypes were up and running, they would then be put together as an entire system. This would allow system level electrical and mechanical checking to be run. Any problems discovered would have to be sent back to the design engineers for another spin of the boards. I have worked on designs in systems like this that required major changes to board outline shapes and component placement in order to make the board work with the rest of the system. It was a tedious process, to say the least.

A STEP IN THE RIGHT DIRECTION

As 3D mechanical CAD systems grew in their abilities, they became an important part of the design process. They could create the board outline shape with the appropriate keepout and mechanical clearance zones, and send that data to the PCB design CAD system. Once the printed circuit board was designed, they could then read the PCB design placement and library files in order to accurately depict the populated circuit board within the system.

Mechanical CAD systems were a major step forward towards total system design. Design teams could now check for physical conflicts between the board and the rest of the system before building prototype boards. However, there still wasn't a way to verify electrical connections, and all changes and corrections still had to be sent back to the individual PCB designs.



Multi-board design within a 3D PCB CAD system

BEYOND MULTILAYER DESIGNS: MULTI-BOARD DESIGNS AT THE PCB DESIGN LEVEL

Today, system level design is a whole lot easier due to the ability to create multi-boards at the PCB design level. Where PCB design systems in the past could only create individual designs, you can now link together multiple board designs in one multi-board project. This makes it possible for individual board designs to be treated as child objects within the main system assembly.

Interconnections between the boards are identified for electrical checking by assigning multi-board properties to connecting nets on the schematic. In this way, the PCB design system knows how the system boards are connected just as if they are physically connected together in the actual system. This facilitates electrical checking to verify that intra-board connectors have the correct pin assignments.

In the 3D environment of the PCB layout tools, the different system boards can be brought together within their enclosure. This allows for physical conflict clearance checking as well as checking for the correct location of interconnecting components such as plugs and connectors. Any changes that are required can be made at the 3D system level in order to verify that all connectors are aligned correctly and any physical conflicts have been resolved.

Multi-board design in today's PCB design CAD tools is not just a step up in your game; it is a complete game changer. You will save time and money previously spent on system prototypes by confirming beforehand that your circuit boards are system ready.

Altium Designer 18 is a top-of-the-line PCB design software that is made for multi-board design. It has a 64-bit architecture and other performance enhancements that will enable you to create your different system PCB designs all within a single multi-board design project.

Would you like to find out more about how Altium can help you to move into a multi-board PCB design system? Talk to an expert at Altium.

PRACTICE MAKES PERFECT: HOW TO TEST ELECTRONIC COMPONENTS DURING PCB DESIGN

When I was younger, well much younger, I had dreams of one day being a pro athlete. I wanted to be the guy who scored the winning touchdown, knocked in the winning run or hit the buzzer-beater. Although I never made to the professional level, I did have some success and I learned a lot. The most important thing was the results you achieved were strictly tied to your preparation. In other words, if I wanted to do well in the game I had to test myself by practicing.

I'm still using those skills, though: in order for your PCB to function as intended, you need to test your design. By testing the electronic components of your board before manufacturing you can be sure that it will perform as you intended with your design.

The most successful teams that I played on had coaches who stressed the importance of practicing as if you were playing in an actual game. It is one thing to shoot baskets alone, but it is something else to make them when you are being defended. The same difference exists between simulating your circuit's schematic without the physical and material restrictions of the actual board and testing your electronic components during PCB design.

IS TESTING PCB COMPONENTS DURING DESIGN NECESSARY?

Although this is a very important design step, schematic simulation does not guarantee the functionality, reliability or even manufacturability of your design. Applying good design for manufacturing (DFM) practices will help you ensure that your design can be manufactured. However, there are additional steps you can take to test functionality and reliability during design.

The reasons for performing a schematic simulation of your circuit are to:

Determine Component Parameters: Primarily voltages, currents, and impedances.

Verify Circuit/System Operation: Check inputs/outputs, and signal levels.

Make Changes and Correct Errors: Replace parts, or make design changes.

Save Costs: By not having to buy components or materials for testing.

But what if you knew that one of your board's electronic components would fail at some point once it is in the field? You would either replace the component or change your design to remove the source of the failure before manufacturing the board. A design tool like a power distribution network analyzer (PDNA) allows you to evaluate parameters of your PCB as if you were using electronics testing equipment on the actual board.

You can condense your design rule checking into a smooth operation within your layout software.

Performing tests on your PCB electronic components during design will not only make sure that your boards will function properly and be reliable but also save costs if changes have to be made.

HOW TO TEST PCB COMPONENTS DURING DESIGN

The use of a PDNA provides a united design and analysis environment for your PCB design. You can perform power distribution analysis over the entire board or only specify predefined areas. Additionally, voltage gradient overlays give the power to look at voltage drops on the ends of all traces; including vias. You can also look at current density on your traces to make sure there are no potential undercurrent/overcurrent issues.

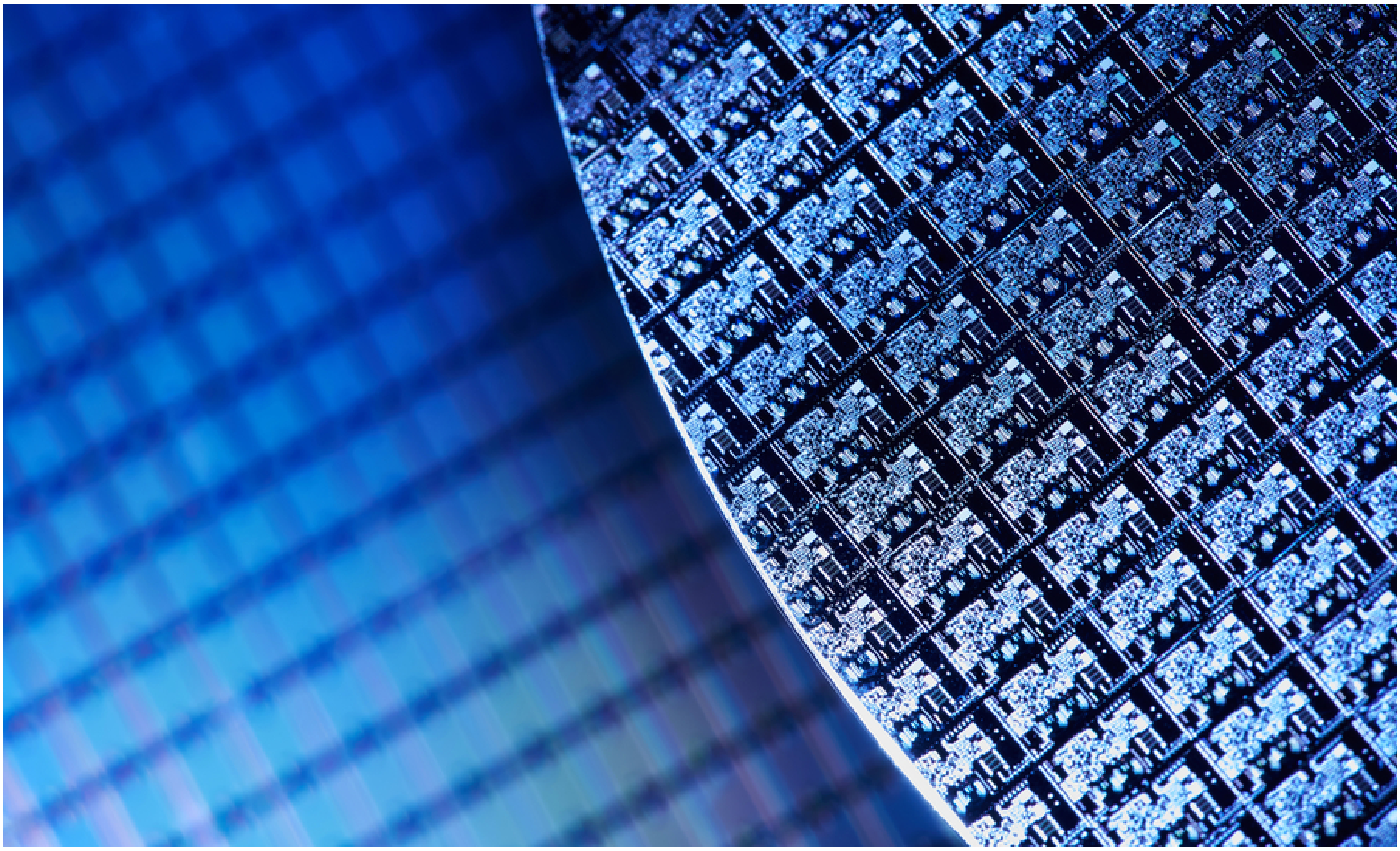
This ability to look at a component's incoming and outgoing traces can be used to determine the component's specific parameters. Employing these capabilities during design is akin to a successful practice where you can follow it up by moving to fabrication and assembly confident that your board will perform as designed.

Successfully practice your PCB design without ever leaving your layout software.

The ability to gather thermal, signal, and power integrity, current density, and trace impedances during design is a significant benefit to the PCB design process. Not only are you able to perform design for manufacturing (DFM) checks, but you can also test the board's electronic components, both of which can save development time. With the right PCB design software such as Altium designer, PDNA and other smooth design methodologies are available to you.

For more information and tips on testing electronic components during PCB design and how to make corrections that will avoid post manufacturing problems, contact an Altium PCB design expert.

PROPERTIES OF SEMICONDUCTORS: ELECTRODEPOSITION AND PCB DEVICES



I started working with gas sensors 10 years ago—my team had several ideas on how to improve the performance of our devices by integrating semiconductor nanomaterials into our PCB-based sensors. We were all masters of electrodeposition and masters of gas sensors in our own right, and we had the idea to integrate the two technologies. Wouldn't you know, though, that a different kind of gas sensor would cause us all more trouble than anything the materials did.

One of the toilets had been phantom flushing for some time, and we decided, collectively, it would be an easy fix. Google said that it was probably just a leaky flapper, so we ordered the part and when it arrived started to the bathroom to take apart the toilet. Long story short, it took six masters of electrodeposition and gas sensors four hours to change this part on the toilet, but we definitely learned a bit about the internal workings of a toilet in the process.

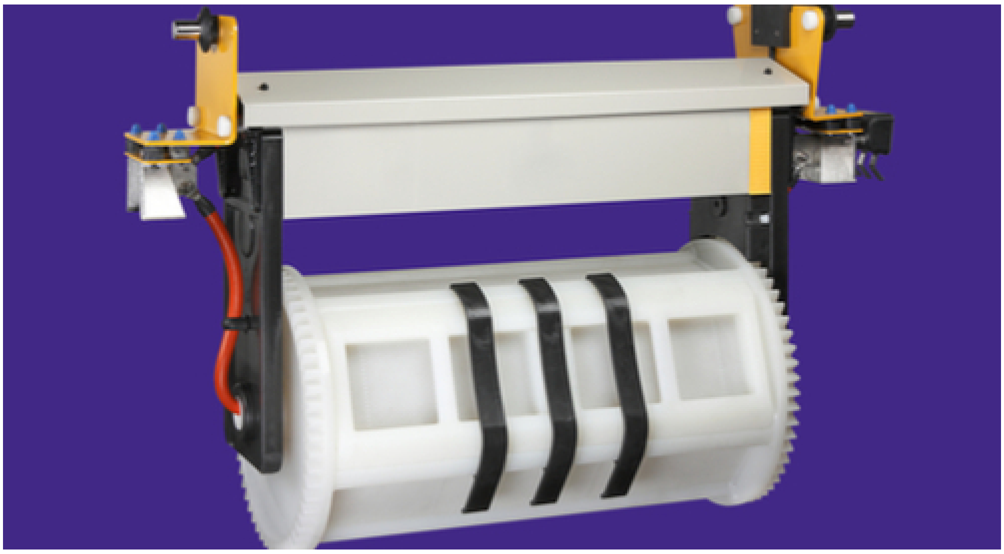
Regardless of plumbing or PCB designing, it's important for you to have a solid understanding of the medium you're working in, and the tools you'll be working with. Electro-depositing required materials directly into our PCB circuit enabled the continued miniaturization of our devices. Learning more about electrodeposition might be of service to you the next time you're trying to find a seemingly impossible solution to a rigorous manufacturing demand.

ELECTRODEPOSITION PROPERTIES: FOR SEMICONDUCTORS AND BEYOND

Electrodeposition, electro-plating—all of these methods are electrochemical processes that are used to deposit some material onto a substrate. Electrodeposition is an electrochemical process that can be used to form a variety of semiconductor films and nanostructures on conducting or semiconducting substrates. The precursor compounds are dissolved in a chemical bath, and the films deposit via an electrochemical reaction.

Certain nanostructures have unique optical and electronic properties that will enable development of new electronic devices. Integrating the nanoscale components with the supporting electronics is required if your unique piece of technology is to proliferate and become wildly successful.

Electrodeposition also enjoys other advantages over typical semiconductor deposition processes. The precursor materials tend to be less expensive, the process can be conducted in ambient conditions, and the equipment tends to cost less than physical or chemical vapor deposition (PVD or CVD), or plasma deposition systems. Electrodeposition also runs at low voltage in contrast to plasma deposition.



Oftentimes there are solutions you may have never considered for a voltage or current problem.

MASTERING ELECTRODEPOSITION AND OTHER CAUTIONS

Anyone that is new to this area should be warned: electrodeposition is easy to perform but difficult to master. Slight variations in the deposition conditions can drastically change the geometry and roughness of the resulting nanostructures. The primary factors influencing the structure of the resulting film are the pH of the chemical bath, its conductivity, temperature, the current density, and the presence of impurities.

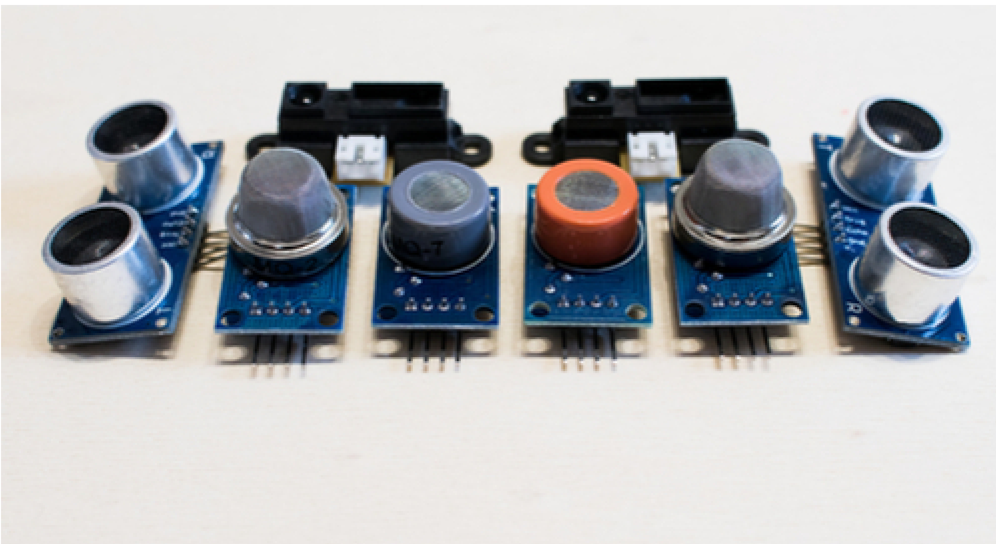
Controlling the deposition conditions requires careful attention and practice. As some of the reactants are used up, the conductivity and pH can change thus resulting in a change in current density. Maintaining the conductivity and pH of the chemical bath requires continuously adding a small amount of electrolytic solution to the chemical bath while monitoring the pH and current density. This will help maintain the deposition conditions throughout the reaction.

APPLICATIONS: WHERE DOES THIS KNOWLEDGE GO?

One notable application can be drawn from the realm of biochemical sensors. Many electronic sensor platforms that are currently available are based on strain rheometry, surface acoustic wave (SAW) resonance shifts, or fluorescence spectroscopy. These devices see greater benefits by incorporating high aspect ratio nanostructures after perfecting electrodeposition processes to different semiconducting materials.

Take, for example, SAW sensors for detecting biological markers or hazardous chemicals. SAW biosensors typically use gold or silver nanoparticles with an attached ligand to detect their target analyte. The attached ligand binds to the analyte, and this increases the mass of the active sensing layer. If instead, a high aspect ratio semiconductor decorated with the metallic nanoparticles were used in these devices, the larger surface area preserves sensitivity while enhancing the detection range and usable lifetime of the device. More information on these devices can be found here.

Another example from this field relates to chemisorptive and physisorptive gas sensors. These devices typically use a polymer film deposited on a pair of interdigitated electrodes as the active sensing layer. If a nanowire film can be deposited directly on the electrodes, the increased surface area enhances both sensitivity and lifetime.



Gas sensors are just one of the many applications that you can innovate in your designs with.

TRY YOUR OWN INNOVATIONS

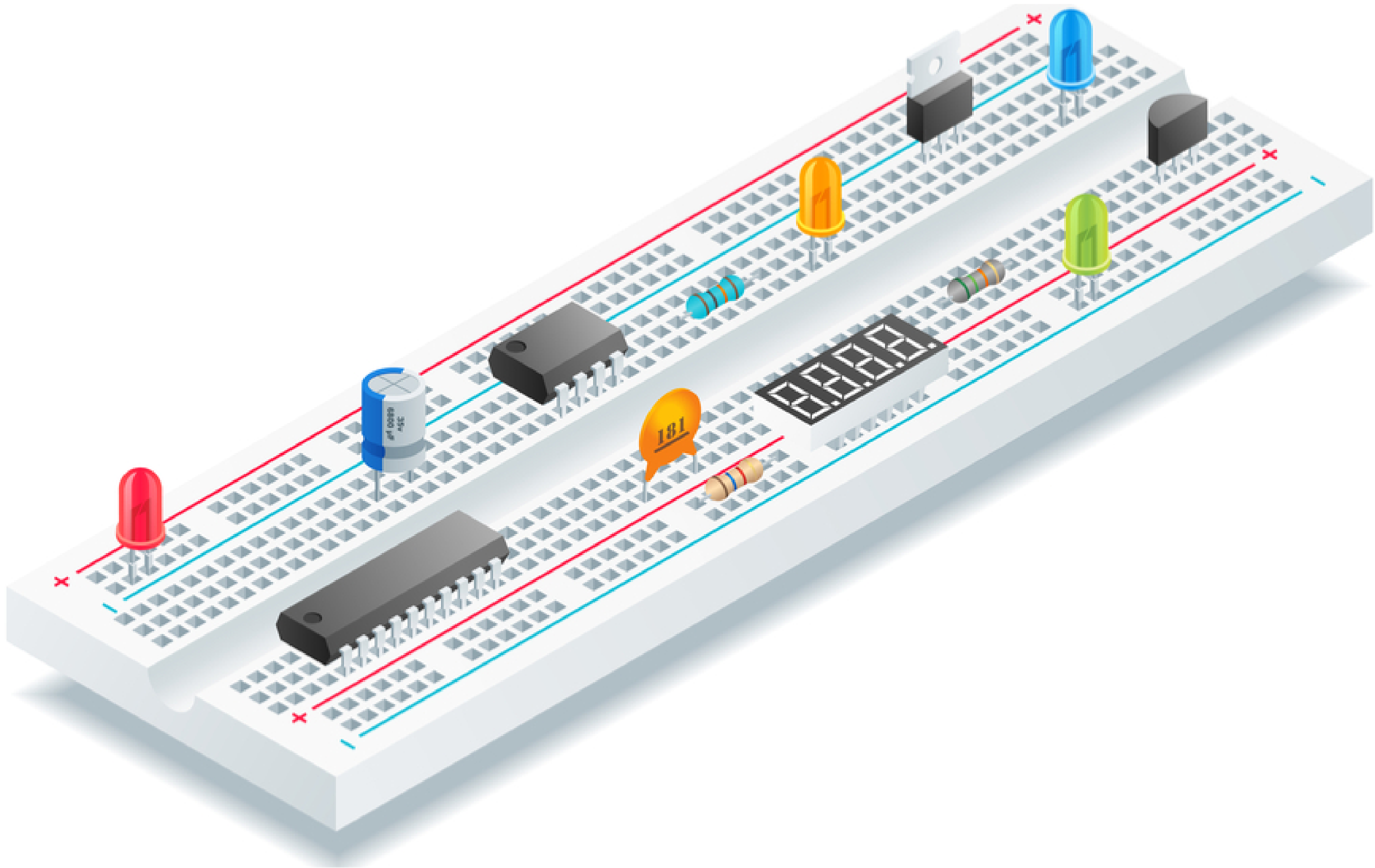
Integrating nanostructured components onto PCB boards via electrochemical techniques may be the first step required to bring new devices to market that have been typically relegated to the academic literature. These techniques also enable efficient control over the device footprint and allow designers to use unconventional and unique materials.

An electroplated PCB is an ideal substrate for integrating your piece of nanotechnology with conventional electronic components. The metal contacts on the PCB can easily take solder, which allows them to be integrated into larger scale devices and allows external components to be attached as needed. The metal contacts also form the substrate for electrodeposition of your semiconductor nanostructures.

If you're looking to take your PCB designs to the next level, you'll need PCB design software which can keep up with your own innovation. With a unified design environment, intuitive rule checking, smart manufacturing output files, and most importantly, a power distribution network analyzer for measuring your voltage and currents, Altium Designer is capable of making your design software methodology make the transition from idea to innovation flawless.

To find out more about how to get the most effectiveness out of your PCB designs, consider talking to an expert at Altium today.

THE ADVANTAGES AND DISADVANTAGES OF DESIGNING WITH BREADBOARDS



Breadcrumbs, breadboards, and breadwinners. Two of which will provide you with a delicious and nutritious meal enough to feed a family, one will provide you with the foundational playground to which your PCB design will spawn from. I will save you time, effort, and many awkward encounters by telling you that if you go around trying to stick jumper leads into breadwinners around town, your design will most likely not succeed.

Similarly, you will likely not maintain proper signal integrity amongst other issues if you try to route your CPU through a mountain of breadcrumbs. Let's loave the questions behind conclude that breadboards are what you use for your PCB design. All jokes aside, breadboards are likely going to be in the desk drawer of every PCB designer and will likely be the first place you will turn when in need of a sandbox style domain for you to test your new designs on.

But some questions will remain even for the seasoned designers. Is it appropriate to use a breadboard for every design application? Are there limits to what these wonderful contraptions can do? Can similar prototyping challenges be accomplished via computer simulation? By further understanding the capabilities of a breadboard, you can determine if using one is best for your design needs.

WHAT ADVANTAGES DO BREADBOARDS HAVE?

It may appear that the limits these boards have are enough to send you running for greener pastures, however, the advantages the boards carry will likely be ample for any designer to at least consider.

The ease with which you can change a breadboard is one of its biggest benefits. Imagine you have a rock-solid design all drawn out and soldered up, only to find that there is a bug buried deep within the system. A long and messy road of desoldering and debugging is in your future. Now if you were to wire up your design with the removable and easily adjustable jumper leads, this design fix would be a seamless event of unplugging and replugging.

Additionally, if your rock-solid design was to be soldered up, ready to go, but then failed in a catastrophic way (cue mini explosion noises), your entire circuit may be in jeopardy and you'll potentially be out of otherwise perfectly functional components. With the breadboard acting as a sort of shield for your (hopefully non-) failing systems, you can ensure that portions of your design will remain intact.

Originally, breadboards were just that; boards used to cut bread on.

WHAT LIMITS DO BREADBOARDS HAVE?

Originally, breadboards were in fact as they sound to be; a board in which bread was cut on. These wooden boards were easy-to-access, and inexpensive way to mount your electrical projects on. As they began to develop the first true electrical boards, the term was just too commonly used with electronics to be abandoned, thus remained the term "breadboard." So aside from the fact that modern breadboards aren't viable tools to cut your delicious sourdough on, there are a few other limitations that exist that must be taken into account when determining your next prototyping steps.

Breadboards will not be a good tool when designing circuits with high voltage. Although most PCB applications deal in low-voltage applications, if you find yourself in the area of 50V, pay attention and consider the fact that the board connections are not suited for this voltage application. Similarly, the boards will have difficulty in keeping up with high current applications. When heading into 20mA's or greater, you will likely find your design not performing as well.

In addition, when dealing with frequencies in the higher, more precise reaches of the spectrum, your signal integrity will certainly be in jeopardy and will not easily travel through the board. The reason for most of the errors you will run into with signal integrity (as well as the aforementioned disadvantages) is due mainly to the physical connection strips within the boards. These strips carry a very high resistance and can, in turn, carry stray capacitance that may wreak havoc.

Computer simulations can sometimes be a better alternative to a physical breadboard

WHAT ALTERNATIVES ARE THERE TO BREADBOARDS?

We've moved from the original boards back in the days of cutting our bread atop our circuit designs to physical copper strips integrated within the board, but what's the next step in breadboard prototyping? Some simulated programs mimic what breadboards aim to accomplish but will not run you the labor of setting up the circuit design (in a physical sense). Limitations of these programs are not nearly as lengthy as a physical board.

As long as the ability is written into each simulator, you can test and simulate to your heart's desire. The only caveat of these programs is the learning curve associated with each, and each likely has its own degree of difficulty.

Aside from the benefits of augmenting my physical board arrangements via interactive visual CAD simulation software, I personally enjoy seeing a physical mock-up of my circuits prior to any finished soldering of a design. It gives me a real world, real functioning mock-up for me to physically hold and feel. This, of course, is my own preference and will highly depend on what my design aims to accomplish, but nothing beats a circuit constructed with my own two hands.

A program that encompasses features such as Design Verification which allow you to simulate your design under a vast array of variables in order to test for flaws and inconsistencies without ever setting foot in a lab setting. A very powerful tool that may save you many headaches (from the plume of blown caps your rock-solid design may run through). If you're looking for a PCB design software with this and other prime features, consider Altium Designer.

Knowing what we know now, where does your design stack up to when considering your use of a breadboard? If you are still questioning the benefits, limitations, and simulated alternatives relative to your design, talk to an Altium expert today.

ADDITIONAL RESOURCES

Thank you for reading our guide on Design for Manufacturing. To read more Altium resources, visit the Altium resource center here or join the discussion at the bottom of each original blog post:

- Bill of Materials Management Strategies: Automation is Your Friend
- Step Up Your Game: Multi-Board Versus Multilayer Designs
- How To Test Electronic Components During PCB Design
- Properties Of Semiconductors- Electrodeposition And PCB Devices
- The Advantages And Disadvantages Of Designing With Breadboards