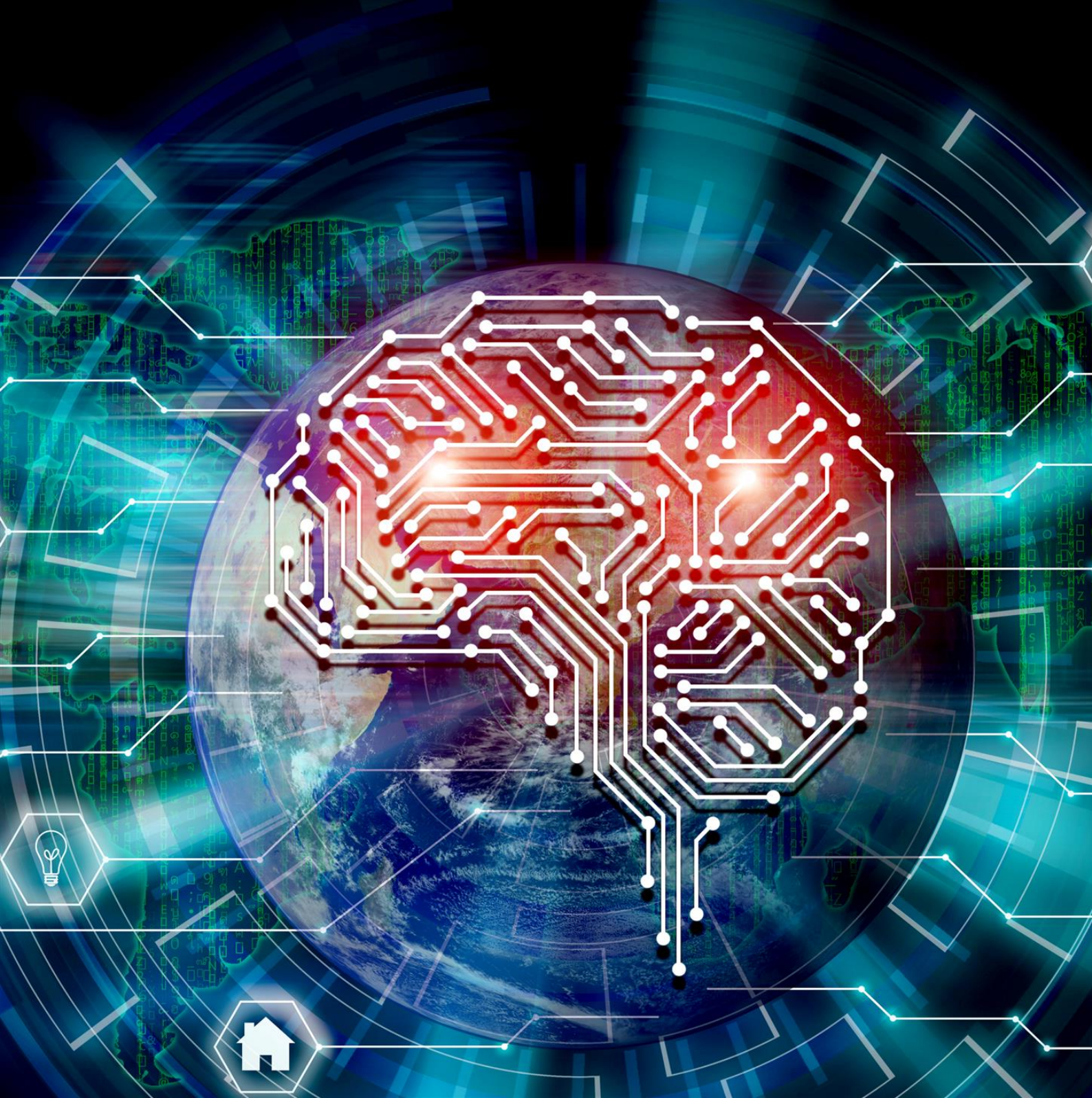


Altium[®]

Bill of Materials Control





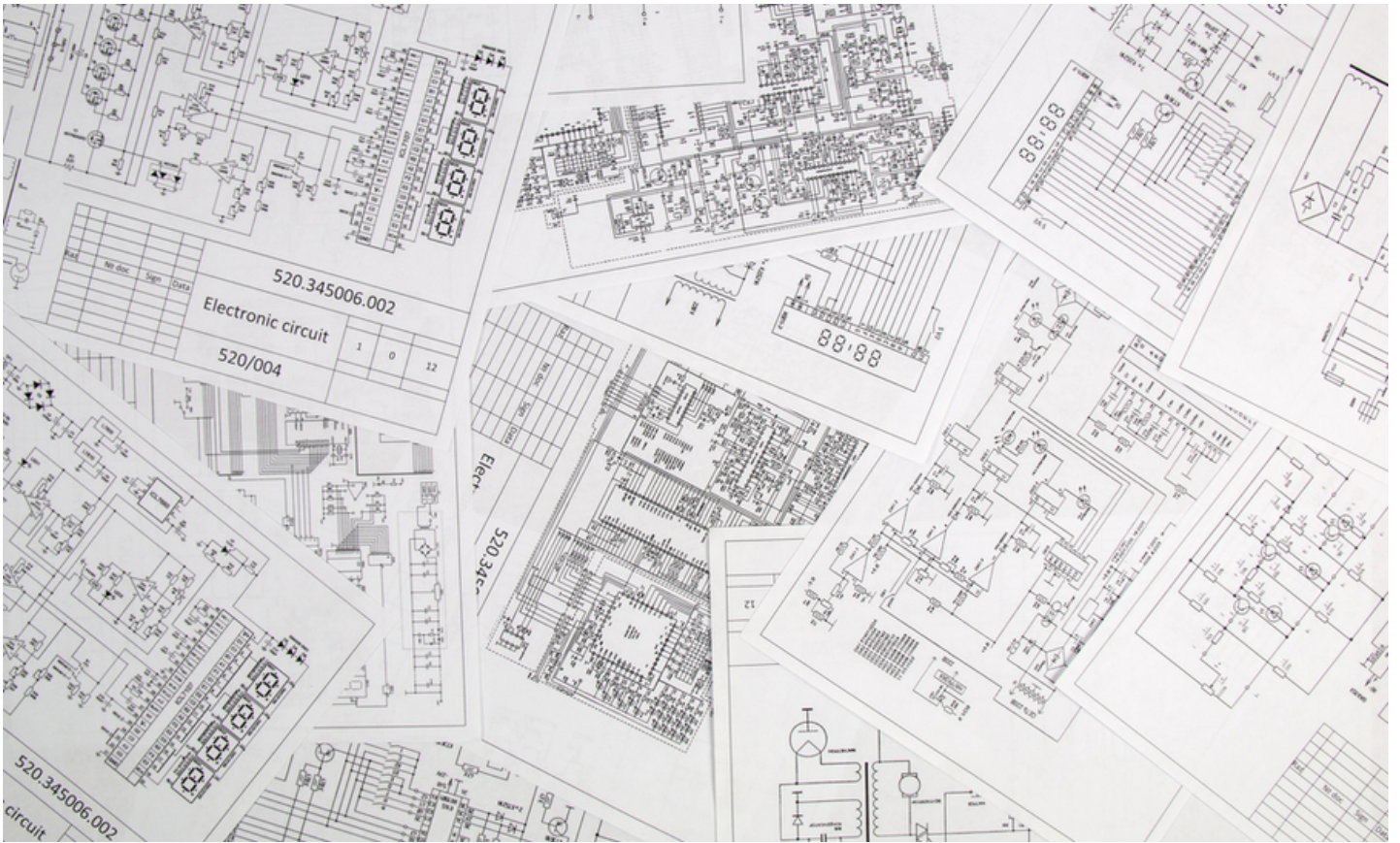
BILL OF MATERIALS CONTROL

In the past, many PCB designers were confined to tools that dictated at which point in the design process that the Bill of Materials could be produced. But we know from years of experience that the Bill of Materials, or BOM, can change throughout the design process and the iterative change management is its own challenge. We believe the decisions should be up to you, while the repetitive aspects should be automated as much as possible, which is why we strive to offer tools that embrace flexible and agile options for PCB designers and other engineering professionals involved in the decision process.

Join us as we explore a variety of topics about Bill of Materials management, including:

- Optimizing Your BOM for Rapid Prototype Manufacturing
- How to Create a Bill of Materials That Meets Your Needs
- PCB Designers for SpaceX and Beyond: Yesterday's Fiction, Tomorrow's Reality
- BOM Control: Setting up your PCB Documentation for Success

OPTIMIZING YOUR BOM FOR RAPID PROTOTYPE MANUFACTURING



With how interconnected the world is, I'm still in awe of all the people knowing and learning multiple languages. Luckily as an engineer, we all are able to speak a common language when it comes to any engineering subject: numbers. No matter where you live zero is zero, pi is pi, and $2+2$ will always equal, hold on let me grab my calculator, 4.

In our current design and manufacturing ecosystem, we are often tasked with working with manufacturers, part suppliers, engineers, and a plethora of other folks outside of our in-house design team. Of all these various moving parts involved with creating the end product, you're lucky if you only have three different languages to be aware of by the time you look at the bill of materials (BOM).

A word that will be one of your best friends as a PCB designer, though, is prototyping. Prototyping is thought to be a quick and dirty way to get your design mocked up via schematics, drawings, and BOMs; however, the speed of your prototyping process can sometimes be anything other than 'rapid'. The disconnect between your prototyping manufacturer and your design can cause speed bumps in this process. But how can a smart BOM help this?

BEFORE YOUR BOM, HOW DO YOU START WITH RAPID PROTOTYPE MANUFACTURING?

So you've gotten to the stage in your design where you are just about ready to unveil your design to some investors at a local event, but you'll need a few prototypes quickly manufactured in order to give them something to hold and feel. You find a manufacturer overseas who has done some great work in the past and you decide to give them a shot. Let's now make our way through some tips in order for you to avoid some common pitfalls and present them with a fully optimized and ready to rapidly produce BOM.

BE THOROUGH, VERY THOROUGH

Growing up, my folks constantly preached over-communication. This, of course, was a late night conversation when I was staying out far past curfew without a whisper of where I was, but the same can be conveyed in all aspects of your BOM. It's far better for you to be overly descriptive in your efforts to describe your design. Sending revisions of your BOM back to your manufacturing takes the rapid out of your rapid prototyping very, well, rapidly.

Your BOM should over-communicate your design and leave no questions for the manufacturer to ask ensuring a higher quality **first pass check**. Don't leave it up to the manufacturer's creativity to assume anything in your BOM, especially if there exists a language barrier between you.

- Did you list the max voltage on your capacitors?
- What about the % tolerance on your resistors?
- And packaging parts, even the stickers?

With these in mind, your manufacturer can have a very clear and concise understanding of what your design entails. Now comes the question of design variability.



BILL OF MATERIALS CONTROL

Ensuring your document is thorough can increase first pass quality in your rapid prototyping stage.

LABELING VARIABILITY

Many parts of your BOM don't have to be the **exact part numbers** listed, rather, they just need to perform within a range of specs. In these cases, you'll have a part in mind but if the manufacturer doesn't have the specified part a substitute will do. This variability in your BOM should be labeled as such. Even the use of a simple color-coded legend will do.

Red labels could mean the part is critical to the design, no substitutes allowed. Yellow might mean P/N changes allowed but to be approved prior to implementation. Green could mean P/N changes allowed without approval as long as specifications met. Now, this is not an industry recognized method of variability, but implementing a simple system into your BOM could save you an arm and a leg when sourcing parts that may (or may not) be suited for your design.

If you have been following along well, then you have over-communicated and color-labeled each part and specification on your BOM. But what if you have a few changes that you plan to implement before you send it to the manufacturer? Should you be tracking these changes? Here's a spoiler for you: of course you should.



Color coding your BOM is a great way to convey variability.

TRACKING CHANGES IN YOUR BOM

It's very rare that you are the only one referring back to your design, and even more your BOM. Being the most relevant document to your manufacturer, your BOM is your playbook of your design, telling all the players involved (engineers, sourcing agents, manufacturers) what parts go where and why.

BILL OF MATERIALS CONTROL

Folks who will be touching your BOM more than a time or two are bound to have to deal with a handful of changes that are inevitably going to happen, and if there is no rhyme or reason to why changes were made, your team may be left scratching their heads and some explanations will be expected later on.

Instead of leaving folks wondering why changes were made, let them know as they happen via change logs, version documents, and revision histories. Anything in your BOM that is revised should be listed within the spreadsheet itself so each team member is able to follow the breadcrumbs of your changes. It is a small act which will go a long way to encouraging efficiency in your rapid prototyping.

The first lesson to keep in mind is that communication, and over-communication, are pivotal in keeping any of your design processes stable and on-track through any product variations. The barriers that exist between overseas and in-house manufacturers do not have to be inhibitive if you remove the chances of misunderstanding. And the easiest place to begin is in your smart PCB design software, like Altium Designer with its ActiveBOM tool to keep your lists organized.

If you would like to further discuss methods that will increase first pass quality when optimizing your BOM for rapid prototyping, talk to an Altium expert today.

HOW TO CREATE A BILL OF MATERIALS THAT MEETS YOUR NEEDS



Sometimes when pressed to answer a question that I'm not prepared for like, "what do you want for your birthday", I'll respond flippantly; "small stacks of hundreds please!" I don't even remember where I got this quote from, but I tend to use it a lot. It often kills the conversation and in the case of my wife, it earns me an immediate "annoyed wife glare" in response. Wouldn't it be nice though if just for once someone actually honored my request?

Well, it'll probably never happen but there are many other things in life that we wish for that are much more likely. Take a PCB design for example. There are tools and functionality available to us today that we could only dream of 20 years ago. Today there are tools and functionality that we are hoping for that are closer to actually happening than you may realize. One of those wishes that I have is for a PCB bill of materials (BOM) tool that better fits my needs.

I'm not an extravagant engineer, I don't need much except for my BOM to work well. I don't like to have to look around through hundreds of suppliers and manufacturers, and would rather have a clean communication system between them all. Perhaps you may be experiencing some of the same pain points that I am and we can commiserate together. But then you need to understand: these BOM needs can be met today.

THE FUNCTIONALITY OF MY DESIGN TOOLS CREATES MY BILL OF MATERIALS

When working with PCB BOM data, here are some of the needs that I have that would be really helpful to have software solutions for:

1. Not being constrained by inaccurate or inaccessible parts: When I'm trying to get all of my parts set up for a BOM, it is exasperating to bump into parts that don't have accurate and detailed information or may not even be available to me in the library.
2. To know the availability and details of new parts in the design: When I've specified a new part in the design based on my limited research, it is really frustrating to find out after-the-fact that the part that I was expecting to use is not actually available.
3. Access to library parts that doesn't take a magic wand to figure out: When I'm trying to organize my BOM, I want to be able to find the library parts that I want to use in my design without having to search through multiple systems using multiple browsers. There's got to be a better way.



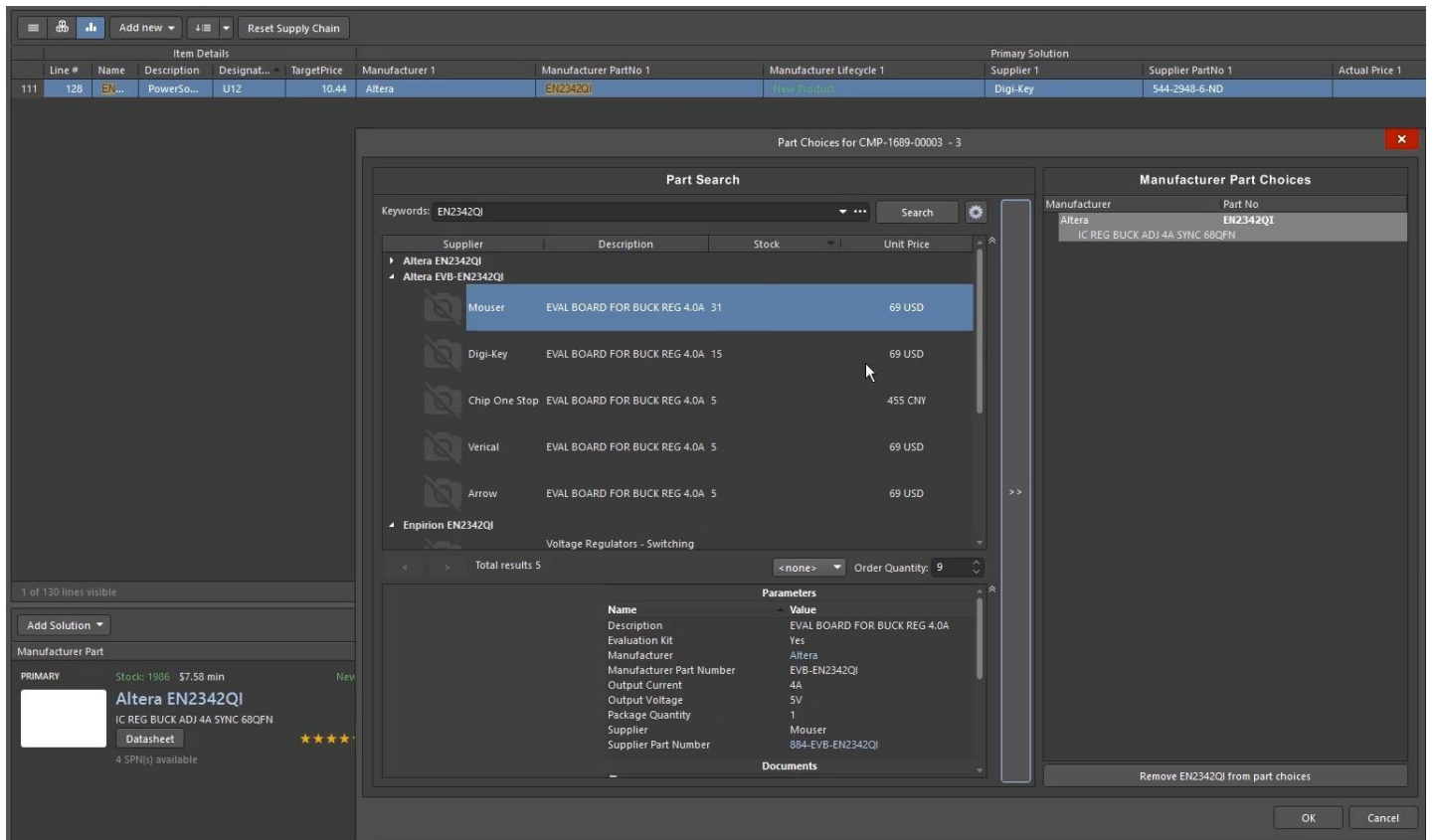
In a perfect world, my BOM management tool would interact with my layout tool

THE MOVING PARTS OF A PERFECT WORLD

In a perfect world, these needs would be taken care of through my design software. I would be able to conduct part vendor research while working on my BOM without having to go through a lot of different search engines. This research would allow me to have immediate access to the part data that I need, and it would also easily tie into the CAD libraries that I am using.

BILL OF MATERIALS CONTROL

Above all this system would be intuitive and productive to use. It would be part of my entire design system and configured to work together with the other design tools. I would be able to have access to my BOM data as well as work with that data while working in the other design tools. This would be a vast improvement over the BOM only consisting of a report that is generated when the design is complete as other design systems have it now.



Managing your PCB BOM as another design tool will give you more power and flexibility

CREATE BILL OF MATERIALS MANAGEMENT SOFTWARE THAT MEETS YOUR NEEDS

Keep in mind, the ability to manage your bill of materials with your design software is already here. With BOM management software you will be able to work with the bill of materials information as part of the overall design. This BOM manager is another tool alongside your schematic capture and PCB layout tools in the same unified design environment.

When you select a part in the BOM management tool and make a change, the same parts in the schematic and the layout will select and change as well. Not only is the BOM manager going to be useful to you for regular design functions, but it will be a great tool to use during design reviews as well. You can also conduct your part research through this BOM manager, and get up-to-date part details and availability information.

Since the BOM manager is part of the same unified design environment as the rest of your design tools, it fits seamlessly into the

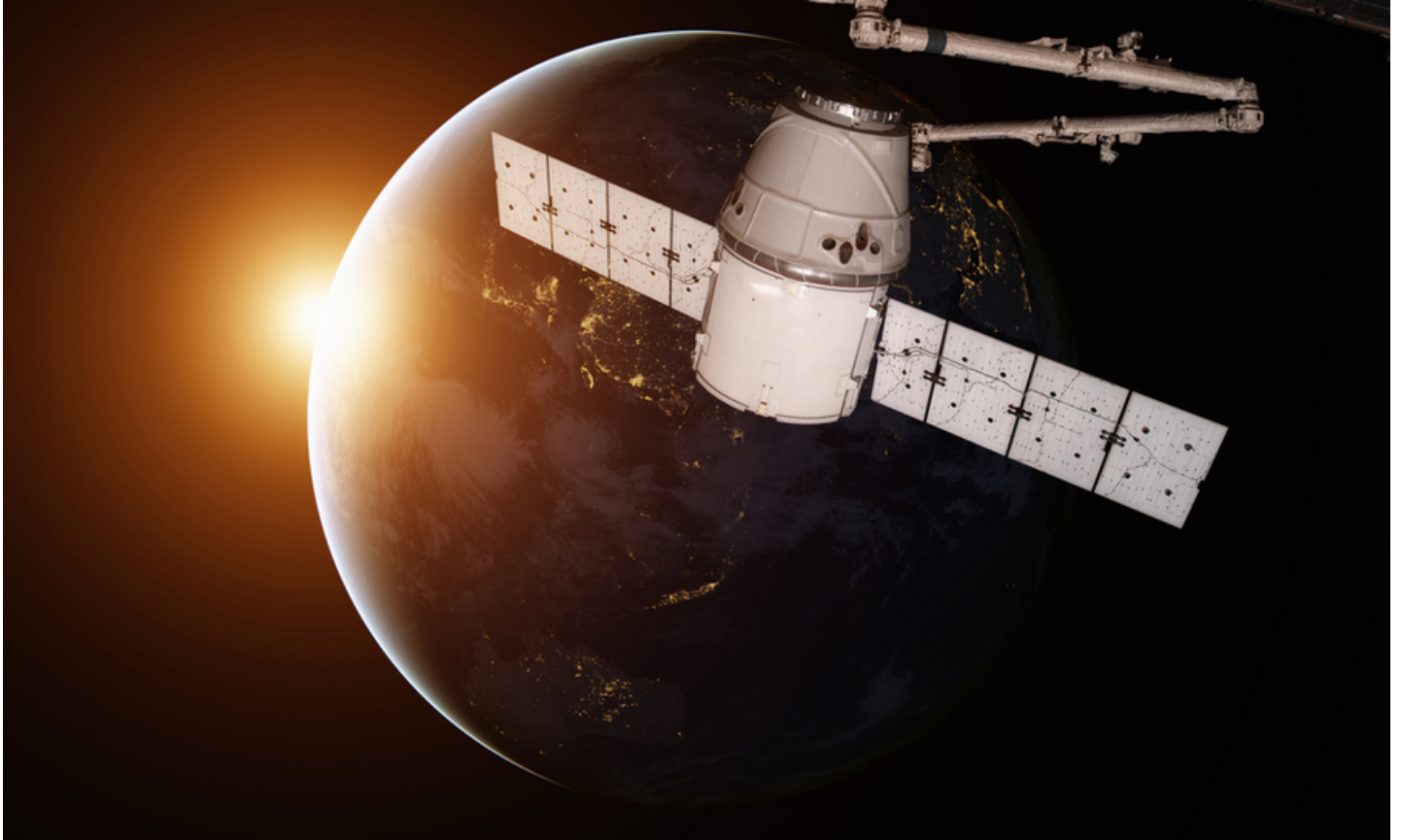
BILL OF MATERIALS CONTROL

design flow along with the rest of your design tools. In addition, being part of the same design environment means that you will have a direct tie to the library parts as well. This will allow you to create a bill of materials that meets your needs, as well as the needs of everyone else in your design flow.

PCB design software, like [Altium Designer](#), has the advanced bill of materials management software that I have been talking about here. [Active BOM](#) will give you better access and control over the parts in your design like you've never had before.

Find out more information by [talking to an expert at Altium](#).

PCB DESIGNERS FOR SPACEX AND BEYOND: YESTERDAY'S FICTION, TOMORROW'S REALITY



I've wanted a career in engineering for as long as I can remember. There were people who could MacGyver something out of nothing and I wanted to be one of those people. I'm sure that watching science fiction on TV added a lot to that. I mean, who wouldn't want to manipulate technology and control immense power like "Scotty" on Star Trek? I found out that I'm not alone in those dreams—apparently, *Scotty was the inspiration* for much interest in engineering.

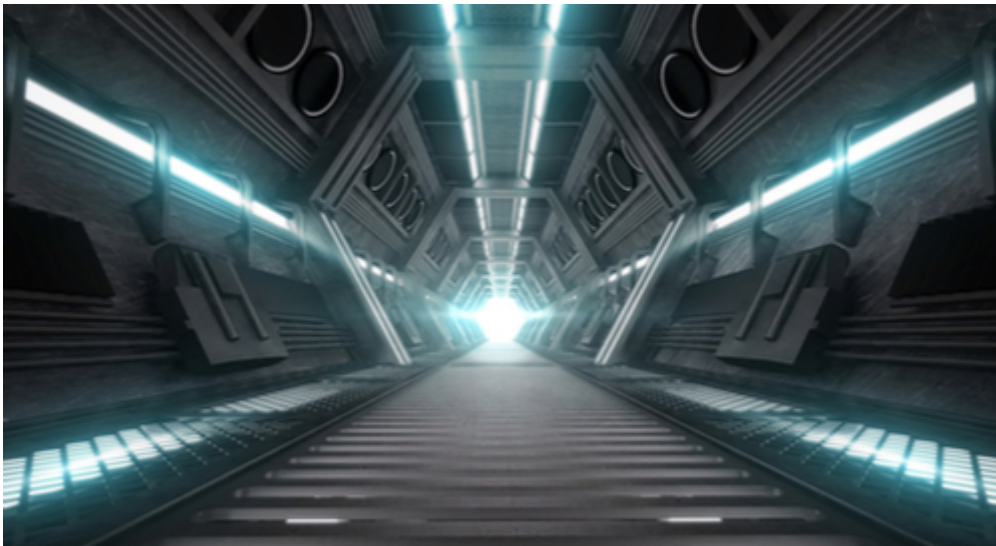
Back then science fiction was not the staple of entertainment that it is today, my parents only tolerated my interest in it. Who would have guessed that those of us who dreamt about spaceships would one day be on the forefront of designing them? Here we are though, and to say "the sky's the limit," has never felt more achievable. Many amazing technologies are being designed now such as driverless cars, robotics, IoT devices, and spacecraft.

It takes more than dreams to design this level of technology though. To design tomorrow's technologies in today's industries is going to take designers who can step up into new challenges. A thorough knowledge of their field is an obvious requirement, but these designers also need to be flexible enough to not be boxed in by that same knowledge. It will also take designers who can use the tools of the trade to be able to make one small step a reality.

DESIGNERS WHO ARE KNOWLEDGEABLE

SpaceX, one of the most recognizable brands designing tomorrow's technologies, is a good sample for the kind of knowledge and experience that PCB designers should be equipped with when reaching for the atmosphere. Designer positions look for designers with experience in RF design as well as designs utilizing high pin-density components. Designers with experience in PCB CAD libraries and library configuration management are also highly sought after.

Further experience being sought after includes mixed analog and digital designs, experience with power supplies, different high-speed design techniques, and understandings of design for manufacturability (DFM), specifications, standards, assembly, and testing principles. What an earful! It sure sounds like they are wanting the moon, but I suppose that's not a bad thing if you are in fact aiming for the moon with your products.



Designing tomorrow's technologies will require out of the box thinking

There's much more that goes into PCB design than simply having the knowledge and experience to make your designs function, though. Oftentimes, design problems will force you to think of creative solutions to problems you've never encountered directly before. Being able to communicate effectively and problem-solve will keep your designs rocketing forward.

DESIGNERS WHO ARE FLEXIBLE ENOUGH TO THINK OUTSIDE OF THE BOX

Obviously, designers of emerging technologies will need a wide range of knowledge and experience; however, the distinguishing feature of many of these designers tends to be in their abilities to think and act proactively. Letting a simple error in your design

BILL OF MATERIALS CONTROL

hinder the testing and prototyping of large-scale projects can be easily avoided with the right foresight and design for stability.

PCB design methodologies are what you'll need to be focusing on most, here. Knowing exactly how your tools function, and how well you can utilize everything available to you for a quick and solid design without having to backtrack, reach into separate tools, or try to manage the organization of multiple software functions.

For any of these emerging technologies and especially for space flight, PCB designs will need to get smaller, more robust, and yet handle more functionality. This will require different materials and higher density designs. To build these designs will require advancements in the fabrication, assembly, and test processes. It is the PCB designer who will need to be able to adapt to and champion these new methodologies and processes in order to create their next level designs.



Advanced design tools will be needed to conquer the design challenges of tomorrow's technologies

THE NEXT GENERATION OF PCB DESIGN TOOLS WILL BE NECESSARY FOR THE CHALLENGES OF SPACE

We already know how design cycles are being compressed in order to meet the overall product needs, and that is a trend that will only intensify. With the need to co-design multiple board projects for these emerging technologies, the need for tools that can provide design solutions is becoming more and more important. A designer whose fluent in the latest design tools and can understand how to optimize their methodology for speed, security, and potency is pivotal.

To help keep the design on schedule and the engineering team focused, PCB CAD tools now feature the ability to work within a unified design environment. This allows the design group to work on all phases of the design within the same environment without having to interface between different schematic and layout tools. CAD tools also now give you the ability to design multiple system boards at the same time within the same environment.

BILL OF MATERIALS CONTROL

Enhancements in PCB design software will give you the edge that you need when working on tomorrow's technologies, and for features like a unified design environment and multi-board designs, Altium Designer 18 has just what you need.

Don't let the wrong software environment keep you slow and trudging along. Upgrade your workflow and start building for tomorrow by talking to an expert at Altium.

BOM CONTROL: SETTING UP YOUR PCB DOCUMENTATION FOR SUCCESS



Years ago I ventured into the world of remote control helicopters. Not the ready-to-fly-out-of-the-box drones that they sell today, but a gas-engine, Bell-Hiller mix, remote-controlled helicopter. What the salesman didn't tell me though was about all the other items that I would need to purchase in order to complete it. I was going to have to purchase the engine, muffler, radio components, servos, starter, battery, as well as much more.

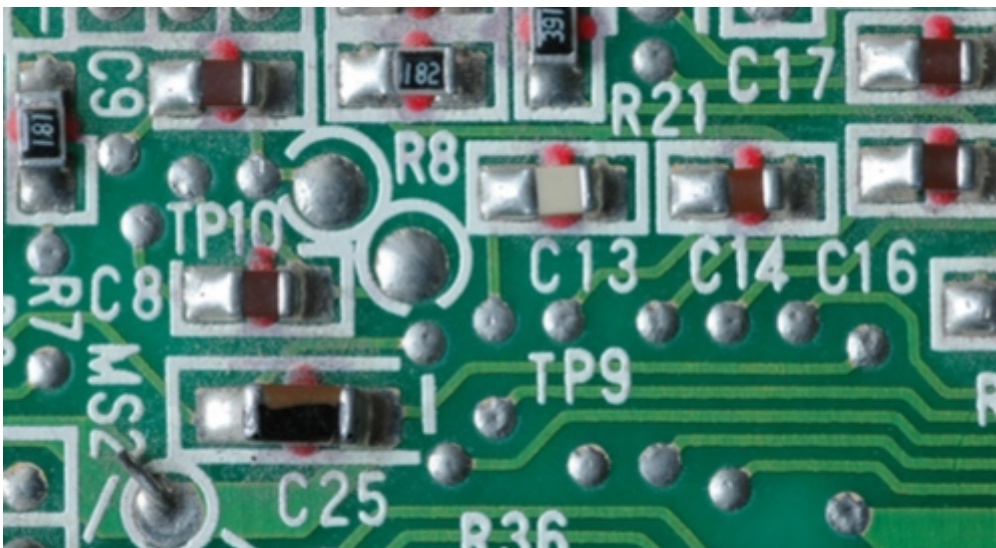
As you could imagine this didn't go over very well with my spouse, and I can't really blame her. A 3X cost overrun due to unplanned missing components is bound to make anyone a little cranky. In the same way, we also can't blame our manufacturers when they get irritated at us for giving them a PCB bill of materials that isn't complete enough to build a board with. An incomplete BOM can cause a manufacturer a host of problems that will eventually come back to us in the form of delayed schedules, cost overruns, or bad builds.

The key is to make sure that your BOM is ready to build your board before you send it out to the manufacturer. In this way, you won't get surprised by missing or incorrect data. There are many opinions, and some debate, as to what is the most important data to include in the BOM. The important thing to remember though is that you must provide enough information for your manufacturer to complete the assembly of your board without any problems.

THE CRITICAL ELEMENTS OF A PCB BOM, AND WHY THEY ARE IMPORTANT

To ensure the success of your PCB BOM, you will want to make sure that the basics are covered. To start with, the BOM needs to have all of the components of the design included in it. Each component will be listed in the BOM along with the following basic information:

- **Part Number:** This is the **identification number for each component** used in the design. Most companies use their own internal part numbering system. This will give them the ability to list the manufacturing part numbers as well as alternate manufacturing part numbers all under the same company part number.
- **Reference Designator:** This is the unique identifier for each individual component in the design.
- **Description:** This is a brief description of the component and can include information such as values and tolerances if they are not listed elsewhere in the BOM.
- **Package Type:** This list the case or package type for that component.
- **Quantity:** For those designs whose BOMs are formatted to list each individual part as a separate line item, this field is either going to be a "1" or completely omitted. For BOMs that are listed by part number, this line will contain the total number of those components used in the design.
- **Comments:** This is **your general "catch-all" field** in the BOM to cover any further information that wasn't detailed in the description field.



Correctly assembling components like these requires an accurate PCB BOM

ADDITIONAL ELEMENTS OF A PCB BOM

There are also many other elements that can be included in a PCB BOM depending on the needs of the BOM. For instance, a board that is being prototyped may not rely on an internal part number and instead default to a manufacturer's part number instead. Here are some of those elements that may or may not get included that you should be aware of:

- **Manufacturer's Name:** The name of who makes this specific component.
- **Manufacturer's Part Number:** The manufacturer specific part number.
- **Component Data:** Some manufacturers want complete component data listed in the BOM instead of being part of the general part description. Here is a partial list of some of that data:

Value

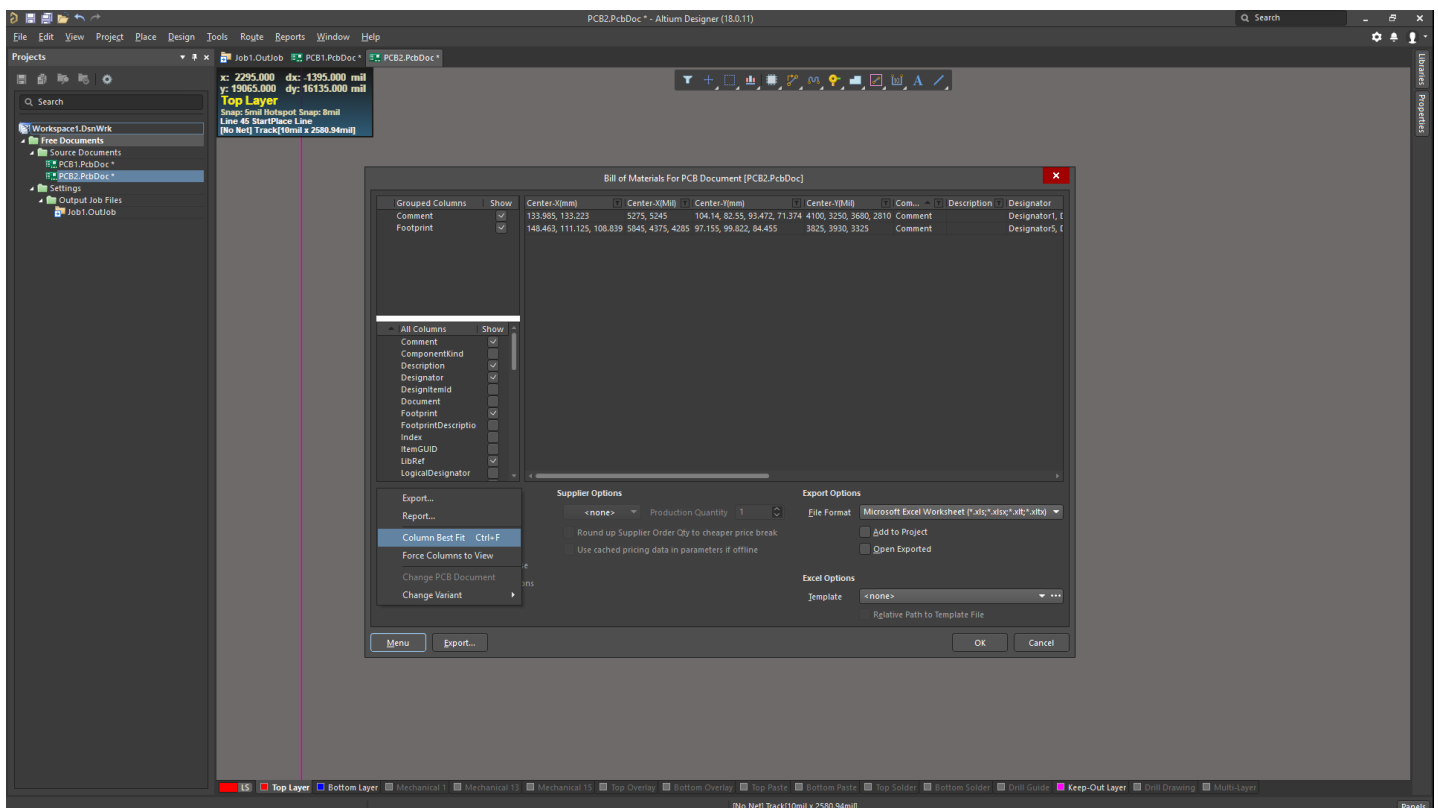
Voltage

Current

Power

Tolerance

- **PCB CAD Footprint:** The CAD model used in layout for that specific component.



BILL OF MATERIALS CONTROL

Make sure that your PCB BOM includes all required mounting hardware

MAKE SURE TO INCLUDE ALL OF THE DATA FOR COMPLETE BOM CONTROL

The data that the PCB layout tools provide to the BOM should give you all of the regular electrical components. There may be some other components though that must not be forgotten to make sure that your PCB BOM is complete.

The PCB may require mounting hardware that isn't included as a regular electrical component inside of the CAD system. Also, you may find out that items such as labels and manufacturing stickers need to be added as well. Does your board have a battery on it? If so then the battery clip may be included as part of the BOM data from the CAD tools, but was the battery itself included? These are a few of the "gotchas" that can cause your PCB BOM to get stuck in the manufacturing process.

Fortunately, there are now [PCB design tools](#) that can provide solutions for making sure that your BOM is complete and ready to go. These tools give you the ability to interact with the BOM as part of the design process in the same way that you interact with the schematic or the layout. Instead of waiting for the design to be complete, you can work from within the BOM tool to get access to part data, and you can cross-probe from there into the schematic and layout.

These BOM tools are much more than simple report generators, they are part of the complete unified design environment as the schematic and the layout. For the advanced BOM functionality that we have been talking about [PCB design software](#) like [Altium Designer](#) will provide the experience you need. [Active BOM](#) has been created to give you more control and power with your design.

If you'd like to know more about how your design software can make your whole design and production process more intuitive, begin by [talking to an expert at Altium](#).

ADDITIONAL RESOURCES

Thank you for reading our guide on Bill of Materials Control. To read more Altium resources, visit the Altium resource center [here](#) or join the discussion at the bottom of each original blog post:

- [Optimizing Your BOM for Rapid Prototype Manufacturing](#)
- [How to Create a Bill of Materials That Meets Your Needs](#)
- [PCB Designers for SpaceX and Beyond: Yesterday's Fiction, Tomorrow's Reality](#)
- [BOM Control: Setting up your PCB Documentation for Success](#)