

***Altium***<sup>®</sup>

# BOM System Improvements





## BOM SYSTEM IMPROVEMENTS

In the past, many PCB designers were confined to tools that dictated when 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. We believe the decision should be up to you, which is why we strive to offer tools that embrace flexible and agile options for PCB designers and other professionals involved in the decision process.

Join us as we discuss a variety of topics to help you with BOM System Improvements, including:

- Cross-Probing Electronic Components in PCB BOM Software
- Don't Stay in the Tower: BOM System Communication and Component Pricing Costs
- No Icebergs Here, Captain! Board Rule Checking for Bill of Materials Management
- Your PCB Bill of Materials Report Can Be So Much More Than a Simple Statement
- Bill of Materials for PCB: Part Numbers and Organization, or Packing Tetris?

### CROSS-PROBING ELECTRONIC COMPONENTS IN PCB BOM SOFTWARE



I have a friend who once bought a 1968 Bel Air that was in immaculate condition inside and out. There was one problem though: the engine ran really rough. Since my friend was pretty good at working on cars, he bought it for a very low price thinking that he could simply tune it up. It didn't work though, and my friend came to realize why this beautiful car had come with such a cheap price tag. Then he took a closer look under the hood and was amazed to find that the spark plug wires had been routed for the shortest distance instead of being correctly routed to their assigned plugs. With the spark plugs miss-wired like this, it was amazing that the engine ran at all. Once he corrected the problem he had a car that not only looked great but ran great as well.

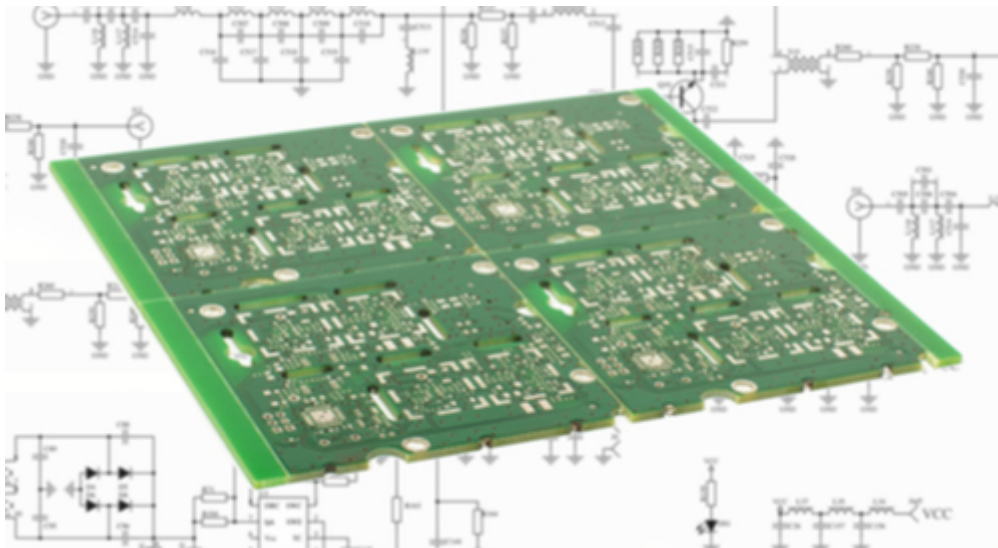
Not only do I wish that I had been lucky enough to find and buy that car myself, but that story also reminds me of the importance of being able to see what we are connected to. My friend had to really search before he noticed that the plug wires were connected to the wrong plugs. In printed circuit board design, we don't always see how our components and nets are connected to the schematic either. Fortunately, our layout tools can help us with this by cross-probing between the schematic and the layout.

Cross-probing will automatically select an object in the schematic when you select the corresponding object on the layout side, and vice versa. You can select parts in the schematic to help organize your layout, and when looking for a component on the schematic you can simply select it in the layout. Slip behind the steering wheel and let your cross-probing from your layout and schematic into the bill of materials turn into a cruise.

### CROSS-PROBING ADVANTAGES IN LAYOUT

Just like my friend and I could give you all the details which make that Bel Air shine, I can also give you the traits which makes cross-probing so convenient. When you are laying out your PCB, the ability to cross-probe will give you real benefits:

1. **Component organization:** If you want to organize your layout without using the cross-probing, you will need to find the components on the schematic, note their reference designators, and then search for those same parts in the layout. This is a very cumbersome process. Cross-probing gives you the ability to select the components you want in the schematic, and have the corresponding parts in layout automatically select. This allows you to easily organize a pile of unplaced components in the layout.
2. **Placing parts according to the schematic flow:** This is really an extension of the first point, but it is an important one. After you organize your components you can then create circuitry patterns on the layout according to the logic flow in the schematic. Being able to cross-probe allows you to do this by selecting the components in the schematic to work with on the layout side.
3. **Net selection:** In the same way that you can select components in the layout by cross-probing from the schematic, you can also select nets. This allows you to quickly find critical nets such as clocks or differential pairs. If you need to add multiple nets to net classes on the layout side, you can do this easily by selecting those nets in the schematic so that they are then automatically selected on the layout side.

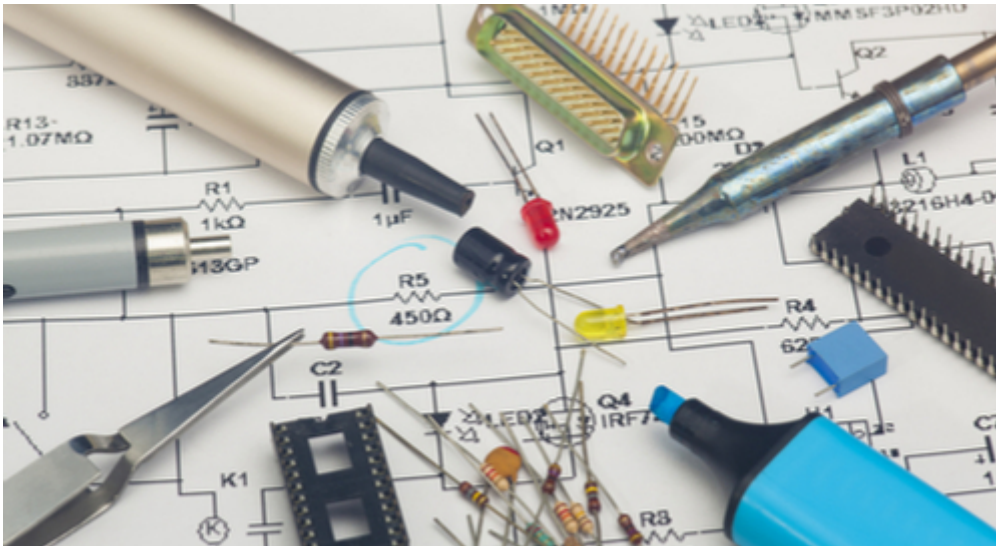


Cross-probing allows you to see what's connected between the schematic and the layout

### CROSS-PROBING ADVANTAGES IN THE SCHEMATIC

Cross-probing between the schematic and the layout works both ways, so anything that you select on the layout will automatically select in the schematic. This gives you the ability to trace back a circuit from the layout to see how it is represented in the schematic. Test technicians will also benefit from cross-probing by having the layout tools open as they debug the board. By finding and selecting the part in the layout that they are currently testing, the schematic will automatically select the corresponding component for them.

I have found that cross-probing also really benefits a design review. With the schematic and the layout presented side-by-side, participants in the review can check from either direction. Once an area that they have concerns about is selected, the cross-probing will then select and show them the corresponding circuitry on the other side.



Being able to cross-probe helps all aspects of the design including manufacturing and test

### CROSS-PROBING INTO THE BILL OF MATERIALS

With the ability to cross-probe from the schematic or layout into the BOM and back again, a whole new world of functionality has been opened up to the PCB design team. Designers are able to find a specific part by looking at a sorted BOM first without having to search through the schematic or the layout first. This is extremely helpful in looking for where specific parts are being used in a design, especially when those parts need to be updated or changed.

Design reviews also benefit greatly from cross-probing into the BOM. When one schematic part is questioned, design teams can immediately cross-probe over to the BOM to find out how many of those parts are being used in the design. This will give the team immediate access to the reference designators of those parts, and ability to select and display them as well.

## BOM SYSTEM IMPROVEMENTS

---

Cross-probing is a great way to keep your wires straight and to know what you are connected to. Additionally, the ability to cross-probe into the BOM now provides you with yet another powerful tool that will help you with your PCB design.

PCB design software, like [Altium Designer](#), has the cross-probing functionality between the schematic, layout and the BOM that we have been talking about here. [Active BOM](#) connects the bill of materials with the rest of your design through the unified design platform giving you the control over your design that you need. Find out more information on how Altium Designer's Active BOM can help your next PCB design by [talking to an expert at Altium](#).

# DON'T STAY IN THE TOWER: BOM SYSTEM COMMUNICATION AND COMPONENT PRICING COSTS



Once upon a time, way back in the dark ages of the 20th century, there were three kingdoms: design, manufacturing, and purchasing. The royal design engineer's process would look like this: they'd design the product, take their carriage to the manufacturing department kingdom where the designer would submit the design quickly and leave, trying to minimize as much chatter and cross communication as possible. The manufacturing engineers who receive the design would devise a parts list and pull straws to who had the displeasure of dealing their parts list to the purchasing kingdom. Finally purchasing would draw up the town suppliers and have them scramble over who had the smallest pleasure of deciphering the exact parts specified by poorly kept and shoddily handwritten lists. Then a dragon would swoop down and light all three kingdoms ablaze. I promise that's how it worked.

Things have changed, though—now, it's the 21st Century. Marketing remains ever-alert to changing trends, and are constantly pressing new requirements onto designers. The design kingdom grew too large for one territory and are now separated by continents, so in addition to responding to marketing, they have to know, instantly, what changes every other design team is proposing. Similarly, the purchasing people have to be ready to hit a moving target, and suppliers, too, need to be nimble.

Nobody has the time anymore to track information through hordes of archived emails, text messages, or phone calls and voicemails. The necessary design information needs to be tracked live in a real-time updating cloud-based format that is available to all. Thankfully, we've developed the bill of materials (BOM) management software to do all this and have consistent component pricing availability. The dark ages are over for your PCB designs—bypass the walls of our processing kingdoms to have the smart communication we need in the BOM software that's available.

### BOM SYSTEM MANAGEMENT: TALK TO EACH OTHER, NOT THE MIRROR

A modern BOM management software facilitates communications between all members of the team. The key to the process is that the engineer can instantly see the modifications that the parts supplier makes on their screen, and the parts supplier is also immediately aware of the engineer's changes. Up-to-the-moment component pricing is an immediately applicable feature for a BOM conversation. Below is an example of the steps in a typical exchange between the designer and a parts supplier.

1. The **design engineer** will share the early-stage design to the parts supplier. This gives an idea of the parts recommendations and enables a degree of futurity to the project.
2. The parts supplier suggests recommended parts changes based on their knowledge of inventory, parts interactions, or pricing changes through interacting with the BOM on their screen.
3. The engineer can then accept or pass any of the supplier's suggestions, making any necessary changes to the layout of the design in the process to show updated component necessities.
4. This process can then be repeated as many times as necessary to supply the right components for your production and pricing needs.

All the while, similar exchanges take place between other parties to the project. The two main caveats are that everyone involved is immediately made aware of every change, and nothing should ever be erased—all too often, somebody's bright idea will turn out to be less effective due to component supply inventory, unreasonable pricing, or design flaws. If backtracking ever needs to be accomplished, then archived projects and BOM lists will be allow backtracking to a point before a design has gone wrong quickly and easily.



## BOM SYSTEM IMPROVEMENTS

---



Don't inhibit your own designs by refusing to communicate with your process.

## KEEP THESE TRAITS IN MIND WHILE YOU MODERNIZE YOUR BOM

Emails and spreadsheets get sent, but did you notice the subsequent email that changed the whole game? You got that spreadsheet, but did everyone else? Most importantly, did management agree with the proposals? And don't forget the wizard's advice that your high component density would run into soldering problems during manufacturing. In addition to faster communication for all, a [properly applied BOM management software](#) eliminates confusion with essential features:

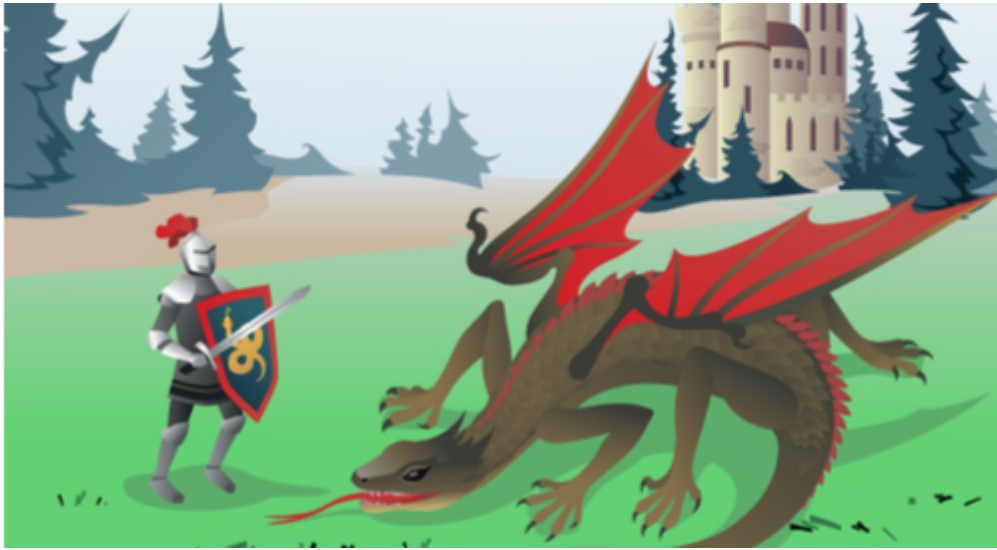
**Access Control:** With proper BOM management software, you can make sure that the various hands in your cauldron can only see and interact with what they need. Different stakeholders can be given access to view and make changes based on the specific privileges they've been granted.

**Universality:** It doesn't do much good if your BOM solution doesn't work with some CAD systems. Ensure that your BOM software can interact with the mechanical, electronic, and software specifics that you need.

**Availability:** Often you will find yourself interacting with design team members, manufacturers, suppliers, and field servicers who are all over the globe. It is important to be able to leave real-time notes, have [accurate component and inventory control](#) at all hours, and maintain an open-access platform for any team member's needs. After all, nobody wants to be woken up from a call from an international team member just for BOM access. A cloud-based solution will be available at all hours to ensure you can rest easy.

**Buy In:** Your divergent teams across the disciplines will all be using a multitude of software systems to accomplish their tasks. Your BOM management software must be able to interface and incorporate technological and software innovations in modern systems.

Fortunately, a BOM management software doesn't need to be centered around one of those ancient kingdoms anymore—it is fully capable of meeting the needs of designers, manufacturers, suppliers, customers, and all of the interested parties.



The only dragon you should face in your PCB design process are the design challenges themselves.

## YOU DON'T NEED A GREAT WALL

With smart BOM software, you don't need to scale and tackle an immense wall between the different facets of your design process. Instead, clear and consistent communication practices are all you need to enforce in order to maintain inventory and manufacturing goals. Leaving detailed notes, and making sure to check updated information is a click of a button instead of three email threads, fourteen web-browser tabs and a quest to find the holy grail.

Configure your BOM software to provide the information you need from your current supplier's part status, availability and any present alternative parts that can save you time and money. Keep your design moving and avoid holding up any part of *your pcb design software's* process. Easy inventory management takes a load of stress and frustration out of any conversational interaction between manufacturers, designers, and suppliers—Altium's BOM software keeps things simple.

If you want to move out of your communication dark ages and jumpstart progress into the 21st century, *talk to an expert at Altium today.*

# NO ICEBERGS HERE, CAPTAIN! BOARD RULE CHECKING FOR BILL OF MATERIALS MANAGEMENT



You know the story of RMS Titanic and how just before midnight on April 14th, 1912, it struck an iceberg in the Atlantic Ocean and sank. What you may not know though is how one small item may have saved the ship from sinking. During the inquiry, one of the surviving crew members who was a lookout reported that they did not have a pair of binoculars to use in the crow's nest. When asked if having the binoculars would have made a difference or not, he replied that they would have given "enough time to get out of the way."

The tragic thing is that the binoculars were there, they just weren't accessible to the crew members who needed them. According to one story, another crew member who had been replaced just before the ship sailed accidentally took the key to the binocular locker with him. Without the key, the lookouts couldn't get the binoculars and therefore weren't able to see the iceberg in time for the ship to steer around it.

It is often the little things that we forget to check that end up causing the biggest problems. This is why we have processes to check for those small things that may end up sinking our boat or causing our printed circuit boards to be non-manufacturable. Traditionally you will check for errors related to how the circuit board will perform by using design rule checks (DRCs), and the manufacturer will check for errors related to the manufacturing of the board by using manufacturing rule checks (MRCs). We will see here how these checks need to be used together in order for the finished PCB to make it through manufacturing without errors and match up with its bill of materials.

### MANUFACTURING RULES CHECKING

The PCB manufacturer has always checked the board for manufacturing rules violations, and for a long time that was the only place that those checks happened. The older layout tools that we used would run the traditional DRCs, but they weren't set up to run all of the checks needed to make sure that the PCB could be manufactured. Once we finished the design and confirmed that the board had passed its DRCs, we would then turn it over to the manufacturer and they would run MRCs with their own specialized tools.

I was always frustrated when one of my designs was rejected by the manufacturer because of a problem that I couldn't check for with my layout tools. Some of these unchecked items included searching for solder mask slivers, acid traps, or silkscreen to silkscreen clearances. A lot of time was wasted waiting for the manufacturer to check my design and then report back to me so that I could make corrections.



Full design rule checking is the best way to get your board through manufacturing with no errors

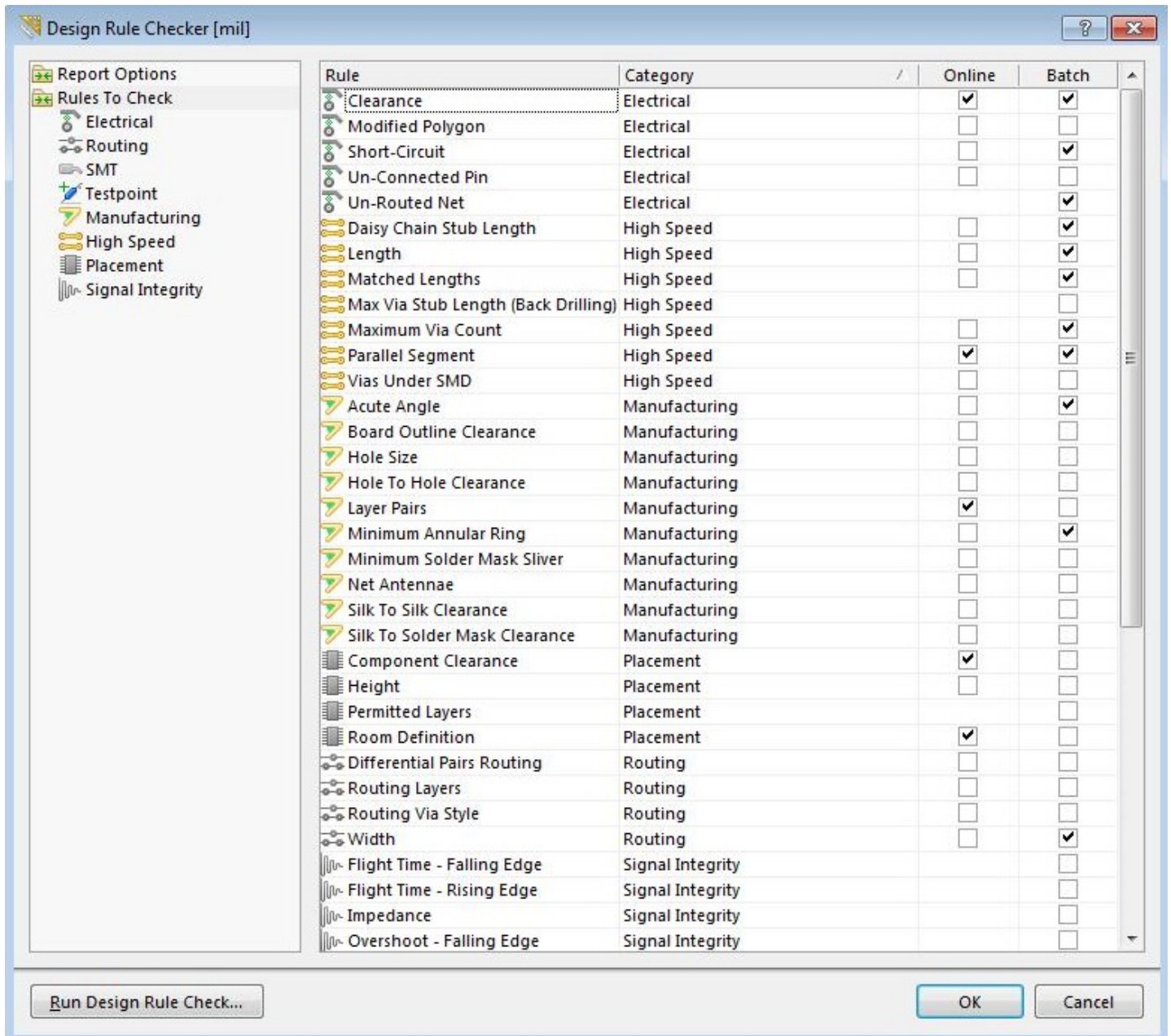
### DESIGN RULES CHECKING

DRCs, on the other hand, are those violations on a printed circuit board that will affect its electrical functions as opposed to how the board is manufactured. For instance, an incorrect size of the conductors on a PCB, such as trace widths or the spokes of a thermal relief pad, can cause the board to not work as it should. Objects like these are among those DRC items that typically don't have any effect on the manufacturing of the board. There are however other DRC items such as object clearances between traces, component pads, or area fills that will affect both the electrical functionality of the board as well as its manufacturability.

DRCs are an essential design tool and should be available to you as an online function while the board is being laid out. Prevent your design errors from occurring due to components being too close together, or net shorts due to traces touching each other with real-

# BOM SYSTEM IMPROVEMENTS

time rule checking in your layout tool. In addition, at any time during the design, a batch DRC pass can be run as well. You can use it to catch any DRC errors that were ignored during design, or if the online DRC was not used.



The layout check design rules menu contains a complete set of DRCs and MRCs

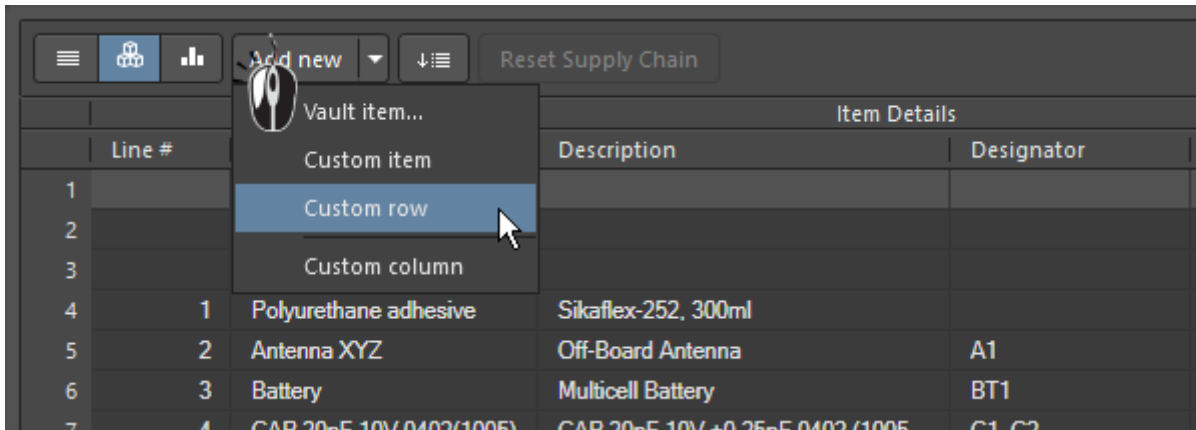
### YOUR BILL OF MATERIALS WILL ALSO BENEFIT FROM RULE CHECKING

The best checking solution is enabling you to check all aspects of the PCB design at once. This includes both the traditional DRCs and the MRCs as well. Fortunately, the MRCs that used to be only available as part of the specialized tools owned by the manufacturers, are now included in the DRC options of advanced PCB layout tools. Circuit board designers are able to detect MRC errors along with DRC errors and correct them all at the same time.

With the PCB design entirely checked for errors and ready for manufacturing, you are able to create the bill of materials (BOM) with confidence in its accuracy. CAD tools today have functionality that allows the creation of the BOM with different formats, groupings and other customizations that make your BOM part of the design process instead of a simple report. Even finding a specific part on the schematic can be easily accomplished by selecting the part from the BOM.

If you're looking for both advanced rule checking functionality and advanced BOM creation utilities, then consider [PCB design software](#) like [Altium Designer](#). It has [Active BOM](#) which is more than an independent or auxiliary document, it is part of the entire unified design platform giving you more control and power with your design. Find out more information by [talking to an expert at Altium](#).

# YOUR PCB BILL OF MATERIALS REPORT CAN BE SO MUCH MORE THAN A SIMPLE STATEMENT



I just finished going through my bank balances and I am very grateful for the online utilities that my bank provides. It used to be that you could only get an account statement in the mail from your bank that you had to reconcile with your checkbook. Today's online banking systems are far superior with different utilities to help you track and organize your finances. With these tools you can move money around, create and monitor budgets, as do many other banking functions as well. It is a vast improvement from those simple paper statements.

Interestingly enough though, in the world of PCB design we still often settle for working with simple printed paper statements. I'm talking about the bill of materials (BOM) reports. These reports are typically generated from our PCB CAD tools during the design process. Like a simple account statement from your bank, these reports will give you only the basic information about your design that the manufacturer needs to build the circuit board. As you know though, there is so much more data involved with PCB design. It is about time that our BOM tools step up their game and allow us to work with all of the design data.

Like a present that has been wished for and then suddenly appears, PCB design teams now have the gift of advanced BOM management tools available to them. These tools do much more than just report basic design information, they are packed full of advanced functionality that let you interact with the BOM as part of the design. Let's look at some of the things that they can do and how these utilities can help you.

## BOM MANAGEMENT TOOLS ARE AN ACTIVE PART OF THE DESIGN

BOM management tools are much more than the simple BOM report generators that CAD systems have traditionally been equipped with. BOM management tools are integrated into the PCB CAD system and are therefore able to offer a greater array of benefits:

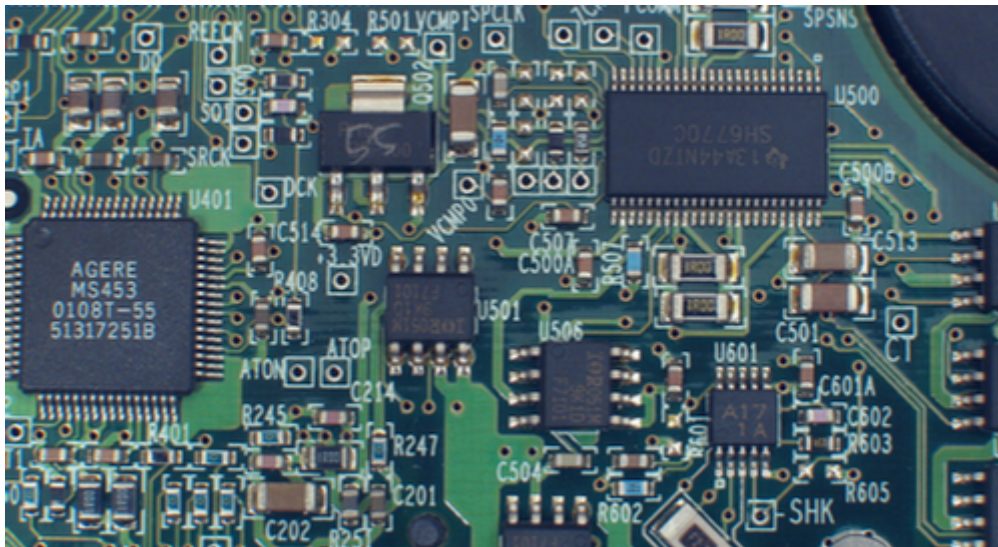
- **Live connectivity to the design:** BOM management tools are connected to the design in the same way that the schematic or

## BOM SYSTEM IMPROVEMENTS

---

layout tools are. Each of these tools performs a specific design task, and they are all interconnected.

- **Full feature menu:** Unlike a report generator, whose limited functionality is activated with a few menu commands, BOM management tools are operated with a full featured menu.
- **Design changes are automatically updated in the BOM:** Since the BOM management tools are live, integrated parts of the design database, any changes to the design are immediately reflected in the BOM. This eliminates the need to generate a new BOM in order to see these changes.
- **Cross-probing:** As part of the design database, BOM management tools allow **cross-probing** functionality as you would normally find with the schematic and layout. By selecting a part in the BOM management tool application window, the corresponding parts in the schematic and the layout will also be selected.
- **Supply chain management:** With the live connection to external data sources that the BOM management tools provide, the designer has access to component manufacturers for part lifecycle information.



BOM management tools will give you better control over the parts used on your PCB design.

## BOM TOOLS ARE CONFIGURABLE FOR THE PCB BILL OF MATERIALS REPORT

Although BOM Management tools will give you a tremendous amount of new design functionality, at the end of the day you will still need to generate regular bill of materials reports. These reports will go to different manufacturers, test engineers, field technicians,



## BOM SYSTEM IMPROVEMENTS

---

or anyone else that has a need to see a parts list for your PCB design. These various groups tend to want to see their BOM data formatted differently. To accommodate this, design teams have traditionally used tools such as spreadsheets to sort the BOM information; however, editing data in external tools from the design environment can lead to errors.

BOM management tools give you multiple ways to sort and report your bill of materials data. In addition to that, you will be able to add custom items to the BOM as well. Items that aren't usually added to the schematic such as mechanical parts, mounting hardware, glue, and identification labels can all be easily added into the BOM management tools. This will give you the ability to configure your bill of materials report to satisfy the needs of your end users, all while still in the design environment.



BOM management tools will allow you to customize your BOM reports

## BOM MANAGEMENT TOOLS ARE PART OF YOUR DESIGN ENVIRONMENT

BOM management tools allow you to work with your bill of materials data in the same way that your schematic editor allows you to work with connectivity data. Schematic editing, board layout, and the BOM management tools all work together within the same unified design environment. This environment gives the BOM management tools the ability to do some functions that a simple report generator could never do:

- **BOM checks:** You will have the ability to check your bill of materials data against the complete design data to ensure that it is correct.
- **BOM data is available for output files:** In addition to being able to generate a standard bill of materials report, you will be able to funnel BOM data into other output files as needed.

# BOM SYSTEM IMPROVEMENTS

---

- **BOM data available for manufacturing drawings:** Data from the BOM management tools can also be used in the creation of fabrication and assembly drawings. If your company's drawing standards include BOM data, the BOM management tools will help to make that task a lot easier by inputting the data directly into your design tool's drawing editor.

In the same way that [an auto-router](#) will enhance your ability to design higher levels of PCB technology, BOM management tools will help to elevate your design abilities as well. Just as I discovered a whole new level of functionality by switching from paper bank statements to online banking tools, you too can discover some incredible benefits with modern BOM management tools.

[Altium Designer](#) is [PCB design software](#) that has all of the benefits that I've described here. [ActiveBOM](#) is an advanced BOM management system that is part of the entire Altium Designer unified design platform. Find out more about how ActiveBOM can help you by talking to [an expert at Altium](#).

# BILL OF MATERIALS FOR PCB: PART NUMBERS AND ORGANIZATION, OR PACKING TETRIS?



It would be great if spending a day at the beach didn't require gathering towels and chairs, fixing a lunch and making sure there was enough sunscreen to go around. Swimsuit selection, packing up and traveling are also unpleasant chores. It would be nice to just think "beach" while sitting on the couch, and suddenly be comfortable working on a tan with your toes in the sand. That's the thing about organization though; it's what really gets us where we want to be.

It would also be nice to just think my design and have it made while I'm relaxing on my couch. Unfortunately, I don't think technology has gotten that far yet. Unlike the last-minute details for what you need for vacation, though, you should never have to last-minute detail what you need for your PCB design. Through carefully planning in your approach to your Bill of Materials for PCB manufacturing you will get closer to that overdue vacation.

## COMMON BOM VALUES, AND PART NUMBER ORGANIZATION

There are a lot of options to consider when deciding what should be included in a BOM. Creating a template with everything that might be needed is a good place to start—it's always easier to narrow a wide pool than it is to expand a small one. Some items are pretty much universal such as component designator, value, quantity, and price. Others such as part number assignment are less definitive but can make a big difference in terms of easily organizing important functions within the company and with its suppliers.

Depending on the size of a business, whether in-house board assembly is used, and how inventory is managed, your inclusions to a

## BOM SYSTEM IMPROVEMENTS

---

BOM will vary. Figuring out how to manage part numbers is one of the most important and difficult chores. If your strategy is correctly organized there will be less time wasted on unnecessarily long documentation, or worse, on confusion. This means more moments for being barefoot on a beach.



Don't resort to organizing your parts with an unclear or unoptimized system.

## UNDERSTANDING THE TYPES OF PART NUMBERS

At the end of the day, the goal of any organizational system is to make your job easier now and in the future. That's exactly the mindset you should be thinking about while getting to know your part numbers, and choosing which organization systems are best to include in your practices. There are at least four types of part numbers that should be considered for inclusion in any Bill of Materials:

**Manufacturer Part Numbers:** The manufacturer part number enables engineers to determine the device's capabilities and specifications, can help contract manufacturers to find alternate suppliers and is a good starting place for ensuring compliance. Make sure to have a plan in place to define and prioritize alternate components. Changing the manufacturer part number of a component in a BOM involves significant risk and should be carefully orchestrated. Do not wait until a component becomes impossible to source before you organize and test your system for dealing with alternates.

**"As Ordered" Part Numbers:** Another common part number is the "as ordered" or supplier part number. This is the number that a purchasing agent requires to be able to order the correct device. These sometimes include all or part of the manufacturing part number, and usually includes whether or not the part requires packaging that allows for automated assembly. A contract manufacturer (CM) may not reference this number for purchasing, though it might help with error checking and allow for quick answers to questions they may have. For internal builds such as proof of concepts and early prototypes, this part number will be important.

## BOM SYSTEM IMPROVEMENTS

---

**Generic Part Numbers:** Generic part numbers are definitely more useful than sand in your socks. One way these are used is when a designer is searching for a standard component for a new design. For example, they may search for LM317 to find an already defined and readily available adjustable regulator, and use whichever manufacturer's part is convenient. Also, Technicians looking at a BOM can read "BC171" much more readily than a typical manufacturer part number, and quickly identify that part as an NPN transistor may expedite a troubleshooting process. For many intents and purposes, generic part numbers can often be left off; though, the occasional convenience makes them nice to include if it doesn't take too much effort.

**Internal Part Numbers:** Organization of a company's physical inventory and digital part library can be greatly aided by the use of internal part numbers. Companies having small inventories can sometimes supplant this requirement by using a manufacturer part numbers along with a "part type" or "library reference" field associated with each BOM component. An internal part number strategy might be to make all jacks start with 100- and all switches with 200- and so on. Alternatively, they could be made searchable by defining a variable named "JACK" and "SWITCH" for the respective groups. Both can be useful, especially if there is a large inventory, and many types of components who's internal part-number prefixes may be difficult to always remember.



Know which numbers organization will be most helpful and convenient to you and your team.

## SOFTWARE SETUP FOR BOM ORGANIZATION

Given the number of options available for part numbers alone, it is obvious that having flexible software is important when it comes to organizing bill of materials for PCB design. Your bill of materials needs to be able to have comprehensive templates, clear walkthroughs for every step of the BOM production, and easy outputs. Furthermore, the best BOM software out there will give you real-time **component obsolescence management**, enabling you to avoid pesky manufacturing errors that will cost you time and money.

## BOM SYSTEM IMPROVEMENTS

---

Industry leaders will go above and beyond the basic level of organization for BOMs, and Altium's ActiveBOM delivers nothing short with real-time pricing, availability, supplier shortage alerts, and the ability to provide solutions for difficult BOM problems such as alternates and variants.

Quit fooling with that silk-screen, and get ready for some sunscreen! Give Altium a try today and [contact an expert at Altium to find out more.](#)

## ADDITIONAL RESOURCES

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

- [Cross-Probing Electronic Components in PCB BOM Software](#)
- [Don't Stay in the Tower: BOM System Communication and Component Pricing Costs](#)
- [No Icebergs Here, Captain! Board Rule Checking for Bill of Materials Management](#)
- [Your PCB Bill of Materials Report Can Be So Much More Than a Simple Statement](#)
- [Bill of Materials for PCB: Part Numbers and Organization, or Packing Tetris?](#)