ALTIUM®

Bill of Materials Management Guide

BILL OF MATERIALS MANAGEMENT GUIDE

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 Bill of Materials management, including:

- What is a BOM in PCB Design?

- PCB Design and BOM Management: How to Ensure Profitability in Low Volume Manufacturing

- BOM Management Tools and Real Time Part Access: How It Will Help Your PCB Design for Manufacturability

- How PCB BOM Management Can Reduce Delays in Manufacturing

- How PCB BOM Management Tools Facilitate Efficient Communications Between Departments

- PCB BOM Creation as a Living Document

- Obsolescence Information Management for PCB BOM Maintenance

- How PCB BOM Management Tools Fit into a Paperless Manufacturing BOM System

- How Online BOM Management Can Get Help Build Teamwork

- How Multi-Level BOMs Can Organize Your Next System Design

Altium.

WHAT IS A BOM IN PCB DESIGN?

I spent many years designing circuit boards in a service bureau. I built library parts, I placed parts and routed traces, and I created Gerber files and drawings. In all that time though, I never worked with a PCB BOM. It probably was because those customers created their own BOM's and we worked only with the data that they gave us. Whatever the reason though, I was in the dark when it came to the BOM and I had a lot of questions.

Since then I've worked with plenty of BOM's in the different jobs that I've had, and I usually don't give them much thought anymore. So, I was surprised then when I recently found myself fielding questions about BOMs. The most popular being, "what is a BOM and what it is used for?". This got me thinking that maybe my BOM-less experience in the service bureau wasn't as unique as I thought it was. Perhaps there are many others out there who need some basic information on PCB BOMs.

If that describes you, then you've come to the right place. It will take just a moment of your time to cover the basics and then you'll be set to go.

THE BASIC ANSWER TO "WHAT IS A BOM?"

The "Bill of Materials", commonly referred to as the BOM, is simply a list. In the case of a PCB design, the bill of materials is a list of all the parts needed to build that specific printed circuit board.

BILL OF MATERIALS MANAGEMENT GUIDE

A PCB BOM is generated from PCB design software. In order to do this, the part information to generate the BOM must already be available in the CAD system. The information for each part is usually stored in its specific CAD library part. As each part is instantiated in the design, the part information is pulled along with it from the library. Once in the design, the part information is then available for use in the different elements of the BOM.

What is a BOM?

THE ELEMENTS INSIDE OF A BOM

There can be many types of information in a PCB BOM, but the BOM must have a core set of elements to start with. Here is a list of some of the more common core elements that you will see in a PCB BOM:

1.   **Comment**: Each part type on the PCB needs to have a unique identification which is listed as a comment in the BOM. Typically a company designated part number is used as the comment, but this is not a requirement. Vendor part numbers or other designations can be used as instead. An example of a comment could be the company part number "27-0477-03".

2.   **Description**: This is a basic description of the part. In the case of the comment 27-0477-03 listed above, the description could be "CAP 10uF 20% 6.3V".

3.   **Designator**: Each individual part on the board will have its own unique reference designator. In the case of the 10uF capacitor, it could be "C27".

4.   **Footprint**: This is the name of the physical CAD footprint used by the part. For instance, C27 could be using a CAD footprint named "CAP-1206".
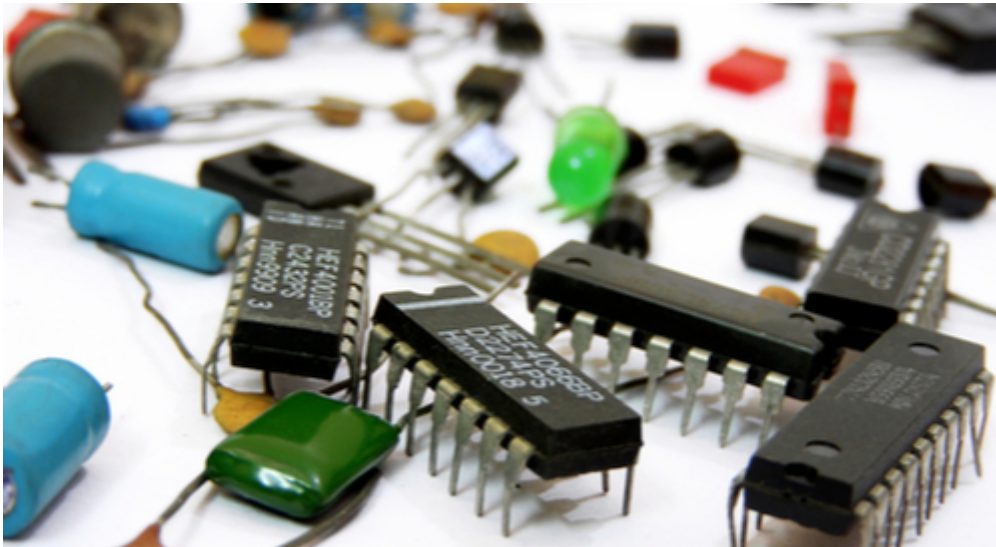
Usually the BOM is organized by the value of the comment element. The other core elements that are associated with it would then be included on the same line. As an example, let's take the core element values from above, and demonstrate a small one-line BOM

Altium.

report:

Comment:	Description:	Designator:	Footprint:
27-0477-03	CAP 10uF 20% 6.3V	C19, C26, C27, C31, C46	CAP-1206

Depending on the requirements of the company, the BOM can also include additional elements of information. These can include tolerances and values, or any other type of information that is associated with the part. The only downside is that your BOM report can get very long depending on the amount of elements that you include in it.

The PCB BOM will report the types and amounts of the components used on your design

ADVANCED USES FOR A BOM

Up to this point we have been talking about the basics of the BOM. There are also some more advanced features that we should mention as well. PCB BOM's can be organized by elements other than the comments, and they can also be used to display some advance design data. Here are some examples:

Change the format of the BOM: You can change how your BOM looks by organizing it differently. Typically you will group the BOM by the comment information, but you could also group it by footprint information instead.

Include components that are not fitted: You have the option to include parts that are designated as being un-stuffed (not fitted) in
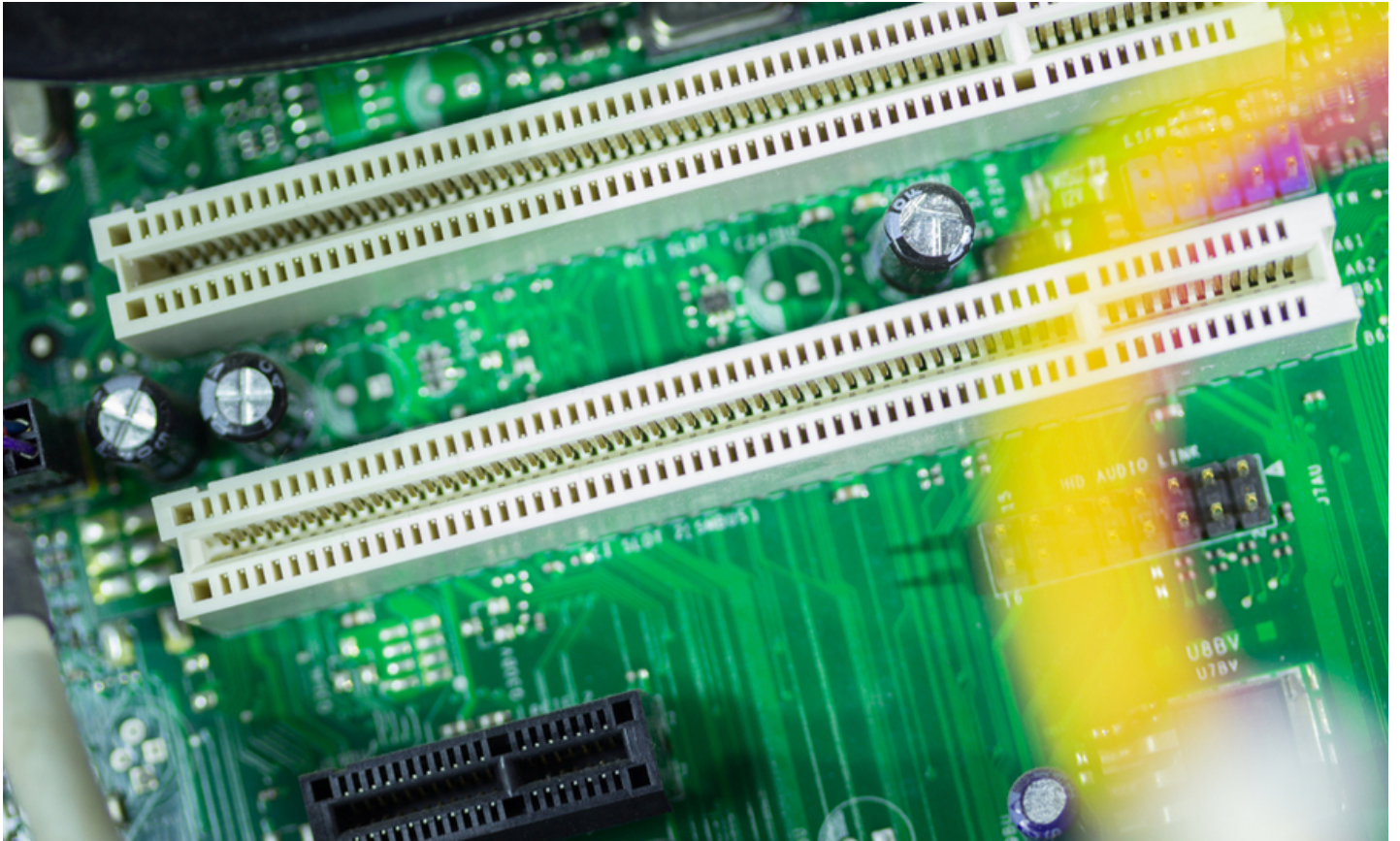
the BOM.

Generate BOMs for different variants of the board: When you have created different stuffing options (fitted or not fitted) for parts in different variations of the board, you can generate different BOMs for those different variants.

I hope that this has answered the basic question of what is a BOM, and given you a lot of additional information as well. A lot has changed since I was asking these same questions in the service bureau. Back then the PCB BOM was prone to errors as it was all managed on paper. It's a lot different today though with online tools designed to manage the creation, updates, and distribution of the BOM throughout your entire company.

Does the idea of BOM management tools that can help your design and manufacturing departments work better together sound helpful to you? If so then Altium's BOM tools might be the answer that you are looking for. Find out more information by talking to an expert at Altium.

PCB DESIGN AND BOM MANAGEMENT: HOW TO ENSURE PROFITABILITY IN LOW VOLUME MANUFACTURING

What's the difference between Angry Birds and SpongeBob? I'm probably at an age where I shouldn't be caring about these things, but choosing the wrong character for my son's 5th birthday cake could spell disaster.

As a designer, you probably can't relate to my need to carefully consider the repercussions of choosing SpongeBob. However, you might understand it better knowing that I apply the same level of caution when designing for low volume manufacturing. Even worse than ruining a child's birthday party, a single mistake in low volume manufacturing can wipe out the product's profit margin, along with the smile on my supervisor's face.

HOW SMALL DECISIONS CAN DRASTICALLY INCREASE THE COST OF PCB PRODUCTION

When only a couple hundred of a specific PCB model is produced a year, each design iteration due to poor design decision can impact the profitability of the product. So does having to create an unnecessary stencil setup that increases the PCB assembly cost by 30%. It's also important to consider the availability of each component you choose for your design. I once opted for a connector

that was more popular at the time without considering its availability in my region. This resulted in a longer lead time, higher shipping costs and having to build up an excessive inventory on site to mitigate these factors. Here are some tips on how to avoid these situations.

8 IMPORTANT CONSIDERATIONS FOR PROFITABILITY IN LOW VOLUME MANUFACTURING
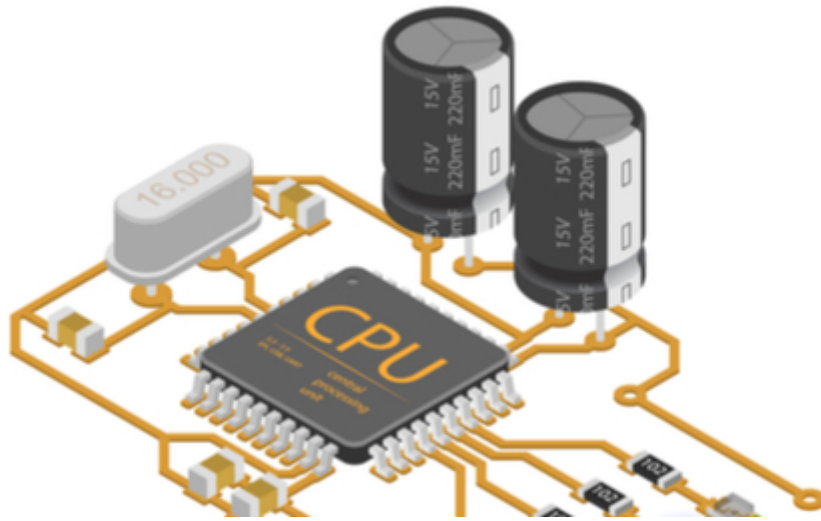
1. Use Commonly Available Parts

Besides cost and functionality, the availability of a particular component is important when it comes to low volume production. Certain projects may require you to commercialize your design quickly, and waiting for months for a single component could jeopardize your entire project. Before you decided on a particular component, be sure to check that they are easily available from your local suppliers.

2. Avoid Components With Short Life Cycle

In my early career as a design engineer, my supervisor wouldn't allow us to use components that were also heavily used in automotive and mobile applications. This was because the mobile and automotive industries were evolving faster than our industry in terms of electronics and so these electrical components were more likely to have a short life cycle. As a result, we avoided having to revise our designs as often due to component obsolescence.

3. Place All Surface Mounted Components On The Same Layer

The cost of stencils used in surface mounted components assembly is significant in low quantity manufacturing. Rather than trying to save costs by placing components on both the top and bottom layers of your PCB to minimize your board size, you can place all of the surface-mounted components on a single layer and save the cost of one stencil. Depending on your board, this can reduce your assembly costs by 25%.

BILL OF MATERIALS MANAGEMENT GUIDE

Having all SMD components on the same layer means you only need one stencil.

4. Panelize Your Design

Panelizing your design is a cost effective measure that can also shorten the lead time of the assembly process. Your supplier may charge a setup cost to panelize your PCB but the overall cost savings makes panelization a preferred option in manufacturing your PCB.

5. Flying Probe Test

You can't cut costs by skipping the post fabrication electrical test for your PCB. However, you can choose the best method of testing based on the nature of your project. If there is a high chance that your design will need to be revised in the future, choose the Flying Probe Test since it does not require a setup cost.

6. Include Assembly Friendly Design Features

Omitting fiducial markers and dummy strips from your design can prove to be costly. Assembly machines need fiducial markers to precisely mount surface mounted components. Dummy strips are essential for leaving space for machines to clamp onto the PCB. Forgetting these important features could incur unnecessary post-assembly costs.

7. Establish A Systematic Post Assembly Test Procedure

Having a lower volume of PCBs doesn't make it any less important to quickly test and isolate defects. A properly constructed test procedure could save precious time in quality control and ensure a timely delivery. You don't want to keep your client waiting while your team struggles to make sure that the PCBs are functional.

8. Prepare An Accurate BOM List

The last thing you would want when manufacturing a low volume of PCBs is to miss a crucial component and delay the assembly

---

Altium.

process. Omitting a component in your BOM list and having to resort to manually placing the part on your own is something that can be avoided. Trust me, you don't want to have to solder one hundred connectors by hand because you forgot to add it to a spreadsheet. Bottom line, always ensure that your BOM list matches the sample that you are sending into your PCB assembly supplier.
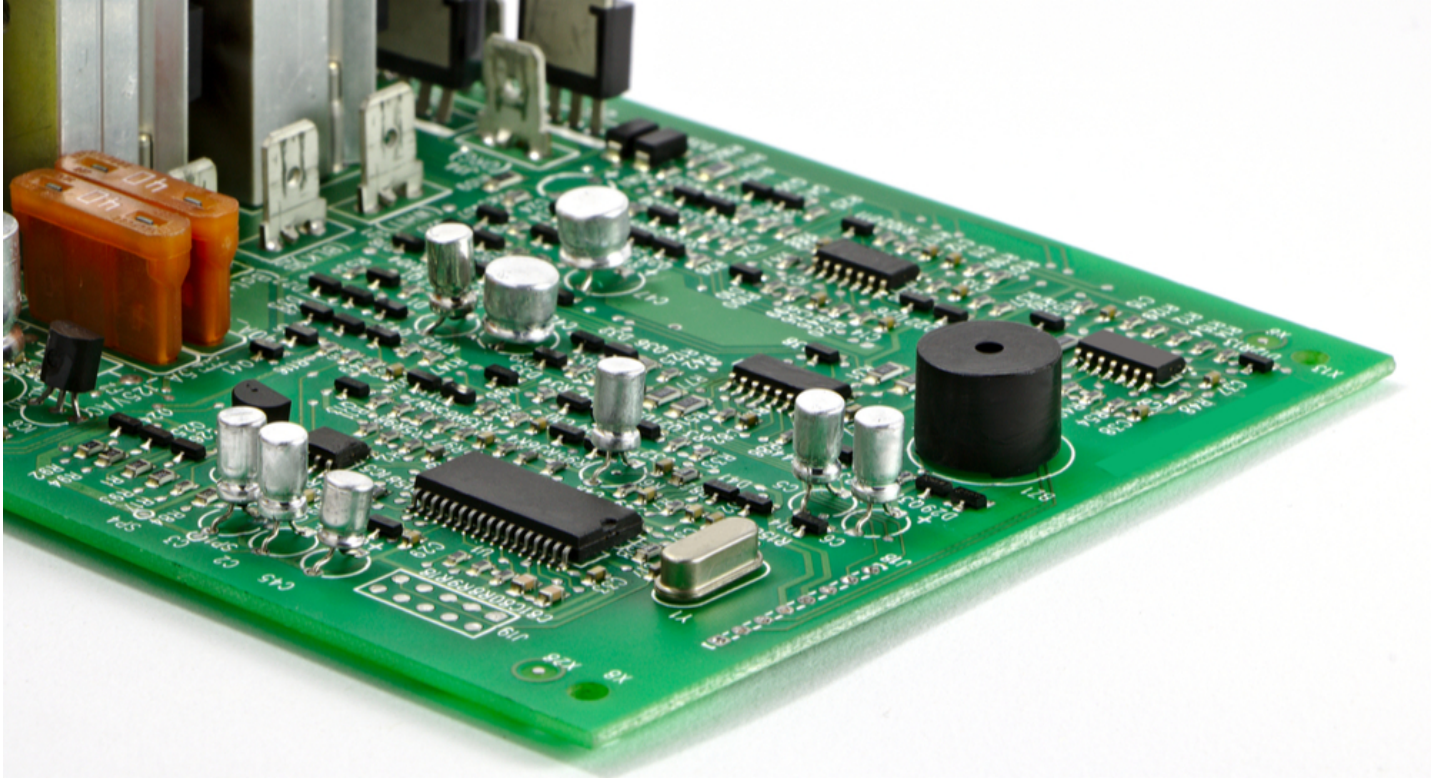

Mess up on your BOM list and you may end up with loads of manual rework.

The burden of making the right decisions to ensure a healthy profit margin for low volume PCBs is on us, the design engineers; however, it doesn't need to feel like one. Altium's BOM tools can make your life easier by automatically creating and updating your bill of materials inventory. That way no components are accidentally omitted and they can be monitored for obsolescence.

Have a question about component management? Contact an expert at Altium.

Altium.

BOM MANAGEMENT TOOLS AND REAL TIME PART ACCESS: HOW IT WILL HELP YOUR PCB DESIGN FOR MANUFACTURABILITY

Have you ever been so ready to take off on a new project that you can't wait to jump into it? Of course you have. As engineers that's part of what makes us tick. We've got so many ideas running around in our head that we would rather forego sleep or food in order to create a design. Then it happens, some kind of delay forces us to lock up the brakes on our creative process. There are many scenarios that can do this, but one of the worst is when we get stopped by a fouled up part selection process.

A delay due to a poorly managed manual part selection process can completely break the creative flow that you initially started out with. The possibility of forgetting an important aspect in the creation of the design while you wait for the delay to be resolved is a real concern. Then, once the problem is resolved, there will be a lot of wasted time as you have to go back and reacquaint yourself with all of what you were thinking and doing before the delay. On the other hand, if you had immediate access to your part inventories instead of going through manual processes to find the parts you need, this is one obstacle that will no longer stand in your way.

A fouled up manual part selection process can kill design creativity

THE FRUSTRATIONS OF A MANUAL PART SELECTION PROCESS

Traditionally, the availability of parts for engineers to select from is manually managed by using a process of spreadsheets and other paper documentation. Here are some of the ways that this process can fall apart if it isn't managed efficiently:

1. The design engineers may not know what parts are available in stock or which ones need to be re-ordered: An un-stocked part can at best slow down the production of the board while waiting for it to be reordered. At worst an un-stocked part can force a redesign of the board or even kill it completely if the un-stocked part is no longer available.

2. A manually controlled part selection process is prone to human error: Errors are par for the course when you are Manually compiling part information from different sources into a master inventory. If there are part detail errors in the list of stocked parts, engineers may unknowingly specify an incorrect part. Imagine a part that mistakenly has a slightly different value than what it should have. The part will look and fit correctly on the board, but the board itself may have intermittent errors in its operation or perhaps not work at all.

3. Design cycle delays: Engineering may have to wait for an unacceptable amount of time for new parts to be researched and qualified. This causes delays in the design cycle.

4. Temporary parts used by engineering may cause problems later on in the design cycle: Engineering typically won't wait for a new part to be qualified, so they will often use a "dummy" part instead as a placeholder on the schematic. This dummy part will then be used until the actual part data is available for this new part request. This can result in a lot of wasted time as eventually the schematic and even the board end up getting re-worked in order to replace the dummy parts with the real ones.

Altium.

If you're new to PCB design then you might not have run into these difficulties, but some of these issues might have struck a chord in those of you with more experience. Now, let's talk about solutions.

BOM Management tools can help you focus on the creative process

BOM MANAGEMENT TOOLS TO THE RESCUE

The source of the frustrations that design engineers experience with a manual part selection process can usually be narrowed down to two areas: the overhead of managing all of the documentation and human error. To combat this, BOM management tools have been created specifically to help to help with the heavy lifting of the part selection process.

Engineers will no longer have to be concerned if their part inventory lists are correct and up-to-date, nor will they have to worry if the details of the part listings are actually correct. BOM management tools will give the engineers real-time access to the part inventories without the worry that human error has caused a discrepancy on the master list.

With BOM management tools connecting different functions within the design process (engineering, quality, layout, purchasing, and manufacturing), engineers will find that new part qualification will have less of an impact on their design cycle time. Since all concerned are plugged into the same system, new part requests can be processed much faster. Additionally, access to new part information through the BOM management tools will give engineers accurate part data immediately. This will eliminate the difficulties associated with working with dummy parts that may not be correct.

Could BOM management tools help reduce your frustrations with your manual part selection process? For easy to use BOM software, check out Altium's BOM management tools. In addition to streamlining and taking the worry out of your part selection process, there are many other benefits to it that you can explore. Talk to an expert at Altium to find out more.
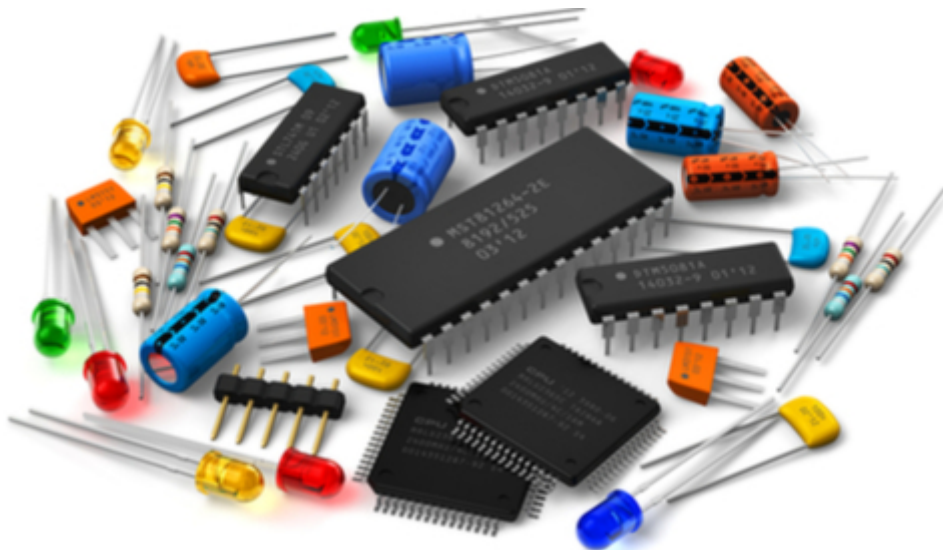
HOW PCB BOM MANAGEMENT CAN REDUCE DELAYS IN MANUFACTURING

Cafeterias in colleges, summer camps, and schools are often known for serving up food that has a consistent quality and similar ingredients. For the kids who are tired of eating spaghetti with meatballs, this approach to food production is a downer. However, from a manufacturing point of view, using standard ingredients with minimal variety makes for efficient production. This is also true in circuit design. When you're designing your PCB for manufacturability (DFM), you'll want to use generic parts and materials and minimize your variety of components.

In theory you should always abide by these rules; however in practice, there are times when new and non-standard parts are required for your designs.

These parts need more time to qualify, order, and stock. They may also require a change to the assembly process. In other words, the introduction of new and non-standard parts to the PCB manufacturing process will likely result in production delays. However, good PCB BOM management can help engineering and manufacturing work together to reduce or eliminate these potential delays and streamline the assembly process.

Parts for a Printed Circuit Board

POTENTIAL PROBLEMS WITH NEW AND NON-STANDARD PARTS

In a design flow that involves multiple departments, each group has tasks to perform when a new part is introduced. The procurement department typically qualifies a new part for price and availability and engineering the part for functionality. In manufacturing, a new or non-standard part will have to be evaluated for the following assembly criteria:

1.   Will it work with the current assembly processes, including pick and place machines and part-handling equipment?

2.   If not, will it require a change to the current assembly process, such as different programming for the assembly line equipment or hardware adjustments, or will it require entirely new equipment?

3. Will it require assembly by hand, and if so, will additional labor and workstations be required?

Any changes to the assembly line will take time to implement. In addition, time documenting and training assembly line staff could also lead to unexpected delays.

Handling new and non-standard parts can impact the manufacturing process. This is because manufacturing needs additional time to organize, stock, and prepare the parts for assembly. If the parts are not available on time, then the assembly line will have to stop and wait until they're ready.

Real-time solutions with BOM management tools

BOM MANAGEMENT TOOLS OFFER REAL-TIME SOLUTIONS

BOM management tools can help resolve the problems that arise from using new and non-standard parts.

BOM management tools synchronize component handling in procurement, engineering, and manufacturing departments, by providing real-time access to the BOM. When engineering specifies a new or non-standard part, both procurement and manufacturing can begin their qualification processes at the same time. Without BOM management tools, manufacturing would have to wait until the last minute to receive a manually created BOM that has been processed by both engineering and procurement. You can see how BOM management tools give manufacturing more time to evaluate the BOM. This extra lead time is necessary for fixing any issues with the selected components. For example, if a component is no longer viable, engineering will have time to consider a redesign solution, instead of forcing manufacturing to make last-minute changes to the assembly process.

Real-time access with BOM management tools also allows manufacturing to see when new and non-standard parts are being ordered. BOM management tools give manufacturing access to the part specifications and order information, so they'll be ready to handle and stock the parts correctly when they arrive.

With part inventories pre-stocked and the assembly processes prepared in advance, your manufacturing department will be able to work at their full potential to build a product. This will eliminate the time and money that gets wasted when the assembly line experiences delays due to unexpected problems with new and non-standard parts. For easy-to-use BOM management software, check out Altium's BOM management tools.

Would you like more information on bill of materials management solutions? Talk to an expert at Altium.

Altium.

HOW PCB BOM MANAGEMENT TOOLS FACILITATE EFFICIENT COMMUNICATION BETWEEN DEPARTMENTS

Many years ago, I was a junior PCB designer learning the ropes of the trade in a large company. To help me understand the entire product production process, my boss took me on a tour of the manufacturing floor. The people that I met there were not rude per say, but I noticed a subtle attitude of, "Oh, you're one of them" that puzzled me. The reason for their annoyance I found out later was that they would routinely suffer delays because of bad information in the bill of materials that came from our design department.

It was not fun then to have a target painted on my back, nor is any more enjoyable today, yet inter-departmental communication problems still exist. Engineering gets frustrated when the parts they want are not approved for purchase, while purchasing gets frustrated when the parts they researched and ordered are not in the final bill of materials. Manufacturing gets frustrated when the BOM doesn't match the parts that they have on hand, and everyone is frustrated at the delays getting a product to market because of the communication breakdowns.

Fortunately, there are now PCB BOM management tools that can help resolve these communication problems between departments.

Altium.

Manually distributed documentation can result in communication breakdowns

COMMUNICATION PROBLEMS WITH MANUALLY CREATED BOMS

In a traditional product development cycle, the bill of materials is created in engineering in conjunction with the development of the design. Usually, the BOM is developed in a spreadsheet and this document evolves as the part requirements change during the design process.

At a point determined by the project management, a copy of the BOM is distributed to the different departments or personnel involved in the product development. This is a preliminary copy of the BOM that will get updated and replaced as the design continues to be worked on. Purchasing will research the parts for cost and availability, and manufacturing will confirm that the parts are ordered, stocked, and that they won't pose any difficulties for the assembly line.

A manually created BOM that is continually updated and physically distributed is vulnerable to communication breakdown problems. Here are some of the possible pitfalls that this process can fall into:

Word of mouth: One common scenario of this problem is when an engineer makes a value change to a part and verbally asks a co-worker to record the change on the BOM spreadsheet. A verbal request like this can be misinterpreted, recorded incorrectly, or even forgotten. Mistakes like this may not show up until much later in the design cycle causing extra time for a design correction.

Version control: Another communication problem can occur with how updates to a manually created BOM are handled. If versions are incorrectly tracked on the BOM, important part decisions may be made from outdated information. This could cause manufacturing delays to find and correct the problem, or even circuit failure on the assembled board if left undetected.

Time: Waiting for updated preliminary BOMs to be distributed takes time. As the amount of preliminary BOMs increases with the evolution of the design, the time taken by waiting for the next distribution adds up as well.

Delivery failures: If the BOM distribution list is incomplete, or staff changes are not accounted for, the BOM may not get delivered to

key personnel. Action items such as part requests and part orders may not get completed in a timely manner causing a slowdown on the assembly line.

BOM management tools can improve inter-departmental communications

COMMUNICATION SOLUTIONS WITH PCB BOM MANAGEMENT TOOLS

BOM management tools can provide the solution to many communication problems. By virtue of being "online" with your coworkers, you will see real-time changes and actions with the BOM.

When a part change is requested, the requester can see the change happen real-time on their own workstation. This will allow them to confirm that the change is correct. Better yet, the requester can make their own change in the BOM tools without taking someone else's time ensuring that the change is correct.

Version control will no longer be a problem as the parts list in the BOM will be "live". As parts are added, changed, or eliminated in the design, the parts list in the BOM will be appropriately updated in real-time. Since every team member in the product design process will have access to the same real-time information, a manually created BOM to track the latest changes is no longer needed.

You will save all the time that currently gets wasted while waiting for preliminary BOMs to be distributed. Instead of team members breaking their concentration to work on other projects while waiting for the next version of the BOM to be delivered, they will have access to BOM updates real-time.

And lastly, manually delivered BOM failures will be a thing of the past. PCB BOM management tools are accessible to every team member, whether they are in engineering, purchasing or manufacturing. This means that instead of waiting for a new version of the BOM to be delivered, everyone will already have immediate access to the same information.

Do these PCB BOM management tool solutions sound like they would be helpful to you and your team? Then Altium's BOM tools might be the solution you are looking for. If so, find out more information by talking to an expert at Altium.

PCB BOM CREATION AS A LIVING DOCUMENT

The other day my son bought himself a new phone because the navigator in his old phone was defective. Every time that he used the old phone for directions, he would end up getting lost. Although taking the time to get a new phone wasn't in his plans, he could no longer afford wasting time trying to figure out where he was and where he needed to go next.

His difficulties reminded me of the problems that I've seen in some PCB design groups where the design staff struggle with the creation of a bill of materials. Following the right steps for creating a BOM should be as straightforward as following a map to your intended destination. But if the process has problems, the efforts to create a correct BOM can be as frustrating as going west on the freeway when you were trying to go east.

Let's look at what a bill of materials is and how it is traditionally created in the PCB design process. Next, we'll examine how a PCB BOM created as a living document is different from the standard method of creation. Then we'll discuss why creating a BOM as a living document is a huge advantage to the design process.

WHAT IS A PCB BOM AND HOW IS IT CREATED?

The PCB design that you create will need a list of parts to identify what and how many components are needed to be attached to the board during manufacturing. This list is known as the "bill of materials", or "BOM".

The BOM document is created as the PCB is being designed according to these steps:

Altium

BILL OF MATERIALS MANAGEMENT GUIDE

1. As design engineers add parts to the schematic, those parts are added to the BOM by the engineers or other members of the design team.

2. As parts are changed or deleted on the schematic, the BOM is continually updated by the design team.

3. When the schematic is ready for layout, the BOM is updated with the component reference designators from the schematic.

4. After the layout is complete, the schematic is updated with reference designators that have been renumbered in the layout. Those updates are then updated to the BOM.

5. At any point in the design process when design updates require part changes, those changes are updated to the BOM as well.

6. Once the design is finalized, the BOM is compared to the layout and the schematic to verify that it is complete and correct.

As you can see, this is a complex process that is dependent on error-free manual input. The use of BOM management tools can eliminate the manual input and automate the creation of the BOM as a living document.

Bill of materials can be better managed online as a living document

AUTOMATED BOM CREATION AS A LIVING DOCUMENT

BOM management tools automatically compile the bill of materials data as a living document while the design is in progress. This gives you immediate access to the current BOM at any point during the design process.

In addition to an automated bill of materials report, BOM management tools will give your company many other advantages. Engineering and purchasing will benefit from online part requests and access to real-time part information. Your manufacturing group will avoid delays by having new part inventory on hand and ready for assembly. In fact, your entire company will profit from the improved communications that these tools provide.

All of these benefits are tied through the same system directly into the creation of a bill of materials. Engineering part requests get automatically transferred to the BOM, as well as the quantities and values used in the schematic. Reference designators are also sent to the BOM and then are automatically updated after design changes.

The creation of a bill of materials by BOM management tools replaces many different manual tasks with one automated system. Here is how that system can help you and your design process.

PCB BOM's created as a living document can help assure your team's success

THE ADVANTAGES OF PCB BOM AUTOMATICALLY CREATED AS A LIVING DOCUMENT

The traditional method of manually creating a BOM, along with all of the edits and updates that go along with the process, are prone to error. There can be mistakes with data entry, design changes could be forgotten, or final verifications get missed. With automatically creating a BOM as a living document, these potential problems can be eliminated.

Altium.

1. Parts that are added, changed, or modified on the schematic are automatically updated to the BOM in real-time without human interaction.

2. Component reference designators are automatically added to the BOM and are continually updated to reflect changes made in the layout.

3. Final verification to check for human data entry error is not required as the BOM has been updated automatically from the schematic and layout data.

Just as a faulty map can slow your trip down by getting you lost, a faulty BOM creation process can slow down your design and manufacturing processes with costly errors. With the BOM created as a living document within the BOM management tools, you will free yourself from these errors that can happen with a manually created document.

Whether you're looking to optimize the way your organization handles information or to update your design software, Altium Designer will be sure to fit your needs. By using Altium Designer's ActiveBOM technology, you'll have some of the most advanced BOM solutions at your fingertips.

OBSOLESCENCE INFORMATION MANAGEMENT FOR PCB BOM MAINTENANCE

In my home laboratory, the garage, I have many bins. One contains small springs, another gear-motors, yet another small bolts... and so on. Despite my girlfriend's skepticism, these bits are actually very useful for fixing assorted gadgets and household appliances, and they also provide fodder for my weekend tinkering habits. Keeping track of what I have and where it is uses up quite a lot of mental capacity in itself. At work, the sheer volume of components makes it necessary to have good software for managing of this information. Aside from freeing up brain power for other tasks, there are other important benefits. Allow me to demonstrate.

MANAGING SMALL CHANGES CAN TAKE LARGE AMOUNTS OF TIME

Oftentimes, component suppliers are kind enough to send you notices that a part is going obsolete once you've ordered a few. This can be very handy for preventing the type of panic attack that occurs when your contract manufacturer stops your biggest client's most important production run due to shortages on the eve of your boss's anniversary party. Given this information, it is necessary to find a replacement for the part and document the change in every BOM in which it's used. But it doesn't stop there. Every new component needs to be vetted as part of its final assembly before a change is released. To do this in a timely manner, you'll need to be proactive and organize how your information is managed.

FINDING ALTERNATE PARTS

Naturally, a good first step is finding the new part. For most passive components, finding a suitable replacement part is not too difficult. The component must have the same footprint and value, and have good or better specification values. For example, tolerance, power dissipation, voltage rating, frequency response, etc. For active components, or for passives like transformers with multiple pin-outs, close attention must be paid to pin assignment. A comparator that meets all of the performance requirements may still not be pin-compatible, so be sure to double check those pin assignments. The result of using the wrong parts can sound like popcorn but doesn't smell nearly as nice. To avoid that scenario, most modern CAD software comes with component libraries to manage this type of information.

Using the wrong parts can sound like popcorn.

CONSIDERING THE BOTTOM LINE

The cost of the replacement is another consideration. Often a direct replacement will have a similar cost, so no sweat. Still, strange things happen in the market of supply and demand and costs can change drastically. On these occasions, making the right decision means considering alternatives such as whether or not there is head-room in the design to use a cheaper component with different specifications. If not, and the cost becomes prohibitive or a replacement can't be procured, you might be looking at board re-work to use an alternate pin-out or even a re-spin. BOM revision control and rework documentation can quickly become complicated to track in these instances, though good software can help.

If a direct replacement was not available but a similar part was, it is important to consider how the component affects the design, and why the original was selected to begin with. For example, a component can be chosen in order to reduce the number of part-types required in a design. Sometimes such component's specifications will exceed the requirements of the circuit. When cost is

Altium

critical, these decisions may also need to be re-evaluated. Good notes that are clearly associated with the schematic will help to figure things out more quickly.

Consider how things are organized.

WHERE DID I PUT THAT?

Once the selection of the replacement has been finalized, finding where the original part was used becomes the next task du jour, or du-week (as it were) depending on how many designs it was used in. Finding every instance of where a component was used in every product of the company can prevent some very unwelcome surprises. When it comes to things that machines do much, much better than humans, this is an important one.

Using Altium, I have the benefit of using BOM Management tools to solve many of the above problems. Aside from helping immensely by comparing component specifications and availability across suppliers while using real-time pricing, there are great tools for figuring out where components are used. These features make it much easier to manage the necessary information for complex PCB assemblies. So make life easier. Be more popular at parties, and more organized at work. Give Altium a try!

Have a question about obsolescence? Contact an expert at Altium.

Altium.

HOW PCB BOM MANAGEMENT TOOLS FIT INTO A PAPERLESS MANUFACTURING BOM SYSTEM

My wife is an avid reader, and we have a large selection of her favorite books filling the bookshelves in our family room. Lately though, she has been purchasing more and more E-books to read on her tablet. E-books are easier for her to store and transport, and she now prefers that format over the traditional printed book.

This transition from paper to electronic format has been going on for a while now, and E-books are just a small portion of this change. Printed newspapers and magazines have diminished in favor of online delivery, and similar changes are being made in the business world. More and more companies are reaping the benefits of electronic documentation and this includes the design and manufacturing of printed circuit boards.

The traditional paper documentation processes on the PCB manufacturing floor are being replaced by paperless systems. In order to function though, a paperless system has to be driven by electronic documentation. PCB BOM management tools can provide an online bill of materials and other electronic documentation to drive the paperless manufacturing processes. Once in place, your company will see new benefits and improvements from using a paperless system.

Altium.

WHAT IS PAPERLESS MANUFACTURING?

The manufacturing process of a PCB requires a lot of documentation. Parts lists, build instructions, and safety procedures all need to be documented for use on the manufacturing floor. A paperless manufacturing process removes the paper documentation that is traditionally used and replaces it with electronically distributed documentation.

The distribution of electronic documentation to devices on the manufacturing floor can be classified in three categories:

1. Mobile devices for individual contributors.

2. Desktop workstations for data management.

3. Large wall-mounted displays for group announcements and information.

Adding electronic enhancements to a traditional paper documentation system will provide some benefit, but it isn't the total solution. To be truly paperless, all aspects of the manufacturing documentation process need to be designed from the ground up to work together as one system. The completed system will need to connect with and support each device as well as interface with other company systems.

Paperless processes help with document management

HOW PCB BOM MANAGEMENT TOOLS CAN HELP A PAPERLESS

Altium.

MANUFACTURING PROCESS

As we've discussed, a paperless manufacturing process requires electronic documentation instead of paper-based forms and spreadsheets. The traditionally created PCB bill of materials is a printed copy of a spreadsheet, which is unacceptable in a paperless system. This bill of materials may even have to be manually re-entered depending on spreadsheet format compatibility. The ideal solution for a paperless manufacturing BOM is for the data to be available online.

PCB BOM management tools provide online data that can readily be used in the paperless manufacturing system. These tools provide the ability to search for parts, make part requests, track inventory, and ultimately generate a bill of materials. All of this is done in real-time from within the PCB BOM management system. With design, purchasing, and manufacturing all interacting together in the same system, interdepartmental communication is improved. This reduces the chance for human data error, and speeds up the BOM generation process.

On the manufacturing floor, the electronic bill of materials data from the BOM management tools can go directly into the paperless manufacturing system. Instead of having to scan BOM data or recreate it from paper documentation, it is instantly available to the paperless manufacturing process.

Using a paperless system can benefit your company

THE BENEFITS OF A PAPERLESS MANUFACTURING BOM TO YOUR COMPANY

A paperless manufacturing process can benefit your company in several areas:

Paper costs money: The manufacturing process can create thousands and thousands of documents. The cost of paper for each of these documents over time can be substantial.

Altium.

BILL OF MATERIALS MANAGEMENT GUIDE

Distributing paper costs time: Paper has to be manually distributed, and that takes time. Each delay due to late or lost documents can contribute to schedule slip.

Managing document files costs labor: All of those thousands and thousands of documents that you will collect over time will need to be cataloged and stored. This is a very labor-intensive activity that is magnified when documents need to be updated for engineering change orders or scanned for electronic storage.

Paper documentation errors cost profit: Ultimately all of these different costs will end up costing your company profit.

A paperless manufacturing process will reduce or eliminate these costs. Paper will not have to be purchased, time will not be spent in distributing paper documents, and labor will not be needed for document management. In addition, a paperless manufacturing process will give your manufacturing personnel immediate real-time access to important data that is critical to their success.

Just as the transition from paperbacks to E-books has helped at home, your manufacturing processes can also benefit by switching to electronic documentation. Not only will your company profit from eliminating paper documentation, but you will also help to reduce the overall dependence on wood and paper resources. The key is to adopt systems and processes such as PCB BOM management tools that will work with a paperless manufacturing process.

Does the creation of electronic documentation with BOM management tools for a paperless manufacturing process sound like a helpful solution to you? Then Altium's BOM tools might be the answer that you are looking for. If so, find out more information by talking to an expert at Altium.

Altıum.

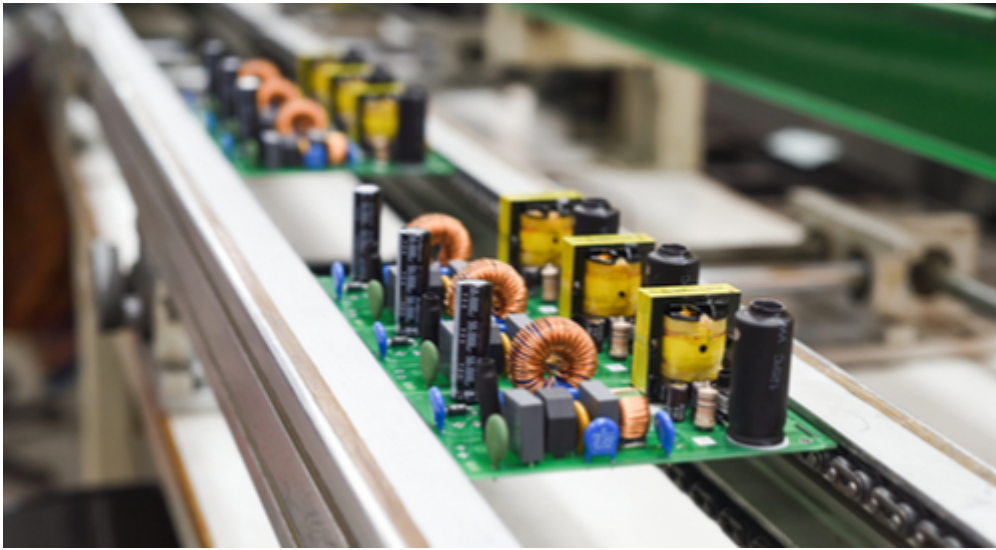## HOW ONLINE BOM MANAGEMENT HELPS BUILD TEAMWORK

Last week one of my co-workers, a brilliant guy named Phil, quit his job for a position in another company. Phil was an amazing designer, and losing him was a tremendous blow to our department and the company. The tragic thing though is that Phil didn't leave for a better job, he left because he felt that he no longer added any value.

It is amazing at how many truly exemplary employees end up quitting because of workplace frustrations and feeling unappreciated. How is it that people who are so talented end up feeling that they no longer have any relevance or are part of the team? One of the causes is communication problems that cause delays and re-work. When people put their heart and soul into what they are doing, and that work ends up in the dumpster, it can motivate them to start looking elsewhere.

Take my company for instance, who design and manufacture printed circuit boards. There is a design side and a manufacturing side of the company. What ties the two sides together is the documentation process. Key items, like the bill of materials (BOMs), are essential for creating a final product. When the documentation process is broken, errors can creep in. This causes schedule slips, re-designs, and entire projects being scrapped.

Let's take a look at how these kinds of problems can negatively impact the self-esteem of both engineering and manufacturing personnel. Then we'll look at some ways to improve communications in order to build teamwork within the company. Hopefully this will help you to avoid the same pitfalls that caused us to lose a key person.

BILL OF MATERIALS MANAGEMENT GUIDE

Don't let communication problems slow down the assembly of your PCB's

THE IMPACT ON ENGINEERING

The engineering department usually has people with a creative flair in their personality. Their goal is to create the best product possible so that it works the way that it is expected to. These people thrive in an environment that allows them to achieve that goal.

However, when that environment is disrupted by broken processes or bad communication, the creative process is interrupted. Take for instance a part that is no longer available even though it is still listed as a viable part. A problem like this could cause an engineer to partially or totally re-design what they are working on in order to replace the part. Engineers aren't just working on what's right in front of them, they are actually working several steps ahead. With a re-design, all of that could be lost. Additionally, the work that has already been done may have to be scrapped. A situation like this kills motivation and leaves engineers feeling like their skills could be better used elsewhere.

THE IMPACT ON MANUFACTURING

The manufacturing department is full of people who are motivated by success. They have a healthy "can do" attitude, and they always want to get the job done better and faster. These people are at their best when the assembly line is up and running well.

When the assembly line goes down though, these folks tend to take it hard. Take for instance an incorrect PCB BOM. A misidentified part can cause the assembly line to be delayed or even stopped, which will keep these people from hitting their manufacturing goals. Unattainable goals often create job frustration, which can lead people to look for job success elsewhere.

Altıum

Improve communication with online tools

IMPROVE COMMUNICATION WITH ONLINE BOM MANAGEMENT FOR PCB DESIGN

As we have seen, broken processes and bad communication can erode teamwork and cause people to lose their sense of self-worth. This can lead to dissatisfied employees and a greater than expected turnover rate. However, this trend can be changed by fixing the processes and improving communications.

The core of the communication between engineering and manufacturing is in the documentation between the two departments. There is the bill of materials, inventory lists, and part requests that are all part of the process of designing and manufacturing a printed circuit board. Online BOM management tools can help improve the documentation process in the following ways:

Part inventories: With part inventories listed and maintained in the BOM management tools, engineering can have immediate access to them. These parts will be up-to-date eliminating problems due to incorrectly listed parts.

Part requests: Engineering will be able to file part requests online through the BOM management tools. With immediate access to current part information, they will know which parts are available for ordering instead of having to wait for approval.

Bill of materials: Manufacturing will have confidence that the BOMs that they receive will be accurate and ready for the assembly line.

Online access and distribution: All personnel throughout the company will have the same online access to the BOM management tools. This will ensure open communications and timely distribution of all required documentation.

Online BOM management tools can solve many of your company's process and communication problems. With everyone having immediate access to the online information, the opportunity for miscommunication is greatly reduced. Engineering won't have their

creative processes shut down because of faulty information, and manufacturing won't be derailed in their drive to success.

With improved communication, everyone will have more ownership of the processes and a greater sense of being part of the team. Teamwork, in turn, will help people to have a better sense of self-worth, which will ultimately help your company to retain key personnel.

Does the use of online BOM management tools to help your company improve communications and processes sound like it would be helpful? If so, then Altium's BOM tools might be the answer that you are looking for. Find out more information by talking to an expert at Altium.

Altium.

HOW MULTI-LEVEL BOMS CAN ORGANIZE YOUR NEXT SYSTEM DESIGN

If only packing for vacation were as simple as it sounds: figure out what clothes you need, throw them in a suitcase, and you're ready to go—wrong! Here's the more realistic vacation planning scenario: pack your clothes, pack your children's clothes, and then pack all the other personal necessities such as diapers, games, etc. Next you need to pack the food followed by whatever gear your particular vacation requires including tents, fishing equipment, etc. Finally you are ready to pack it all into the car, and hope all those years playing Tetris pays off with it all fitting in the car. Oh, and don't forget the dog.

The only way that I have found to successfully arrive at our destination without forgetting anything important is to be organized with packing lists. That means everyone has their own list to manage, and I have the master list to bring it all together. If this structure sounds familiar, don't be surprised—it is exactly how a system level multi-level bill of materials (BOM) for printed circuit boards works.

If you haven't worked with a multi-level BOM before, let me take you through some of the highlights. We'll talk about how a multi-level BOM is different than a regular flat BOM and some of the challenges that go along with it. Then we'll look at ways to help you manage and work with a multi-level BOM and how it can ultimately benefit you.

Altium.

WHAT IS A MULTI-LEVEL BOM?

A PCB BOM lists the components needed to assemble a circuit board as individual line items. The components are usually listed in the BOM by their company part numbers, followed by quantities, reference designators, and general descriptions. There may also be additional information listed such as the component tolerances and values, or the physical footprint used by the component on the board.

A multi-level BOM, which is also known as an indented BOM, works the same way except that it is at the higher system level instead of the lower individual PCB level. In a multi-level BOM each PCB used in the system is listed as a sub-assembly, and each PCB sub-assembly contains all of the regularly listed PCB BOM contents. The concept is similar to a hierarchical schematic where you can see all of the functional areas of the design represented by block symbols on the top sheets.

A multi-level BOM represents the entire system instead of only one PCB

PROBLEMS MANAGING A TRADITIONAL MULTI-LEVEL BOM

Multi-level BOMs have been in use for a long time and are typically managed manually. As an example, let's take a simple master design that has a controller board and a power supply board within it. You would extract the individual PCB BOMs from both CAD databases and then combine those BOM's into one master spreadsheet for your multi-level BOM. Since this is a manual editing process, there are some potential risks:

- **Manual editing can lead to input errors:** No matter how careful you are, there is the possibility of introducing errors into the master multi-level BOM. You may make a selection error when cutting and pasting, or a typing error when inputting the data.

Altium.

BILL OF MATERIALS MANAGEMENT GUIDE

- **The two sub-assembly BOMs are not connected:** When you create the two separate BOMs, there isn't any electronic connection between the two designs. This forces human verification to make sure that both BOMs and their designs are using common components to lower manufacturing costs.

- **Parts may not get updated across both sub-assemblies:** Without an electronic connection between the two designs, parts that are updated in one may not be flagged for updates in the other.

A multi-level BOM will contain all the separate BOMs in the system

BOM TOOLS CAN HELP WITH MULTI-LEVEL BOMS

The good news is that today's bill of material management tools can help you with the creation and management of multi-level BOMs. With online BOM tools you will be able to add individual PCB BOMs as sub-assemblies into a master multi-level BOM. This will give you the following benefits:

- The multi-level BOM is created by the online tools: When you create a multi-level BOM, the work will be done by the BOM management system. You will no longer have to worry about potential errors created by manual editing.

- The separate BOMs are connected electronically: Once you have added the separate BOMs as sub-assemblies, they are now connected together in the master multi-level BOM. This will give you the ability to compare components through the system to verify commonality of designs.

- Online BOM tool functionality works across all sub-assemblies: With functionality such as part life-cycle and purchasing built into the online BOM tools, you will be able to manage the parts across all sub-assemblies. For example, when a part is flagged

Altium.

for needing an update, it will be flagged in all sub-assemblies.

- Sub-assemblies can be pulled out for manufacturing: If required, sub-assemblies can be pulled out of the main multi-level BOM as an individual BOM and sent to a different manufacture.

- Sub-assemblies can be modified individually: Sub-assemblies can be changed out with other sub-assemblies for updates or corrections.

What used to take a lot of time and careful checking can now be done online by simply pointing and clicking to add separate BOMs into one multi-level BOM. Fortunately, packing your multi-level BOM using BOM tools doesn't have to be as frustrating as your vacation packing. The creation of a multi-level BOM with BOM management tools can help your BOM creation process by speeding it up—and making it less vulnerable to error.

Don't suffer through packing your multi-level like the old Wagon Queen Family Truckster for vacation. Use Altium's BOM tools, and find out more information by talking to an expert at Altium.

ADDITIONAL RESOURCES

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

- What is a BOM in PCB Design?
- PCB Design and BOM Management: How to Ensure Profitability in Low Volume Manufacturing
- BOM Management Tools and Real Time Part Access: How It Will Help Your PCB Design for Manufacturability
- How PCB BOM Management Can Reduce Delays in Manufacturing
- How PCB BOM Management Tools Facilitate Efficient Communications Between Departments
- PCB BOM Creation as a Living Document
- Obsolescence Information Management for PCB BOM Maintenance
- How PCB BOM Management Tools Fit into a Paperless Manufacturing BOM System
- How Online BOM Management Can Get Help Build Teamwork
- How Multi-Level BOMs Can Organize Your Next System Design