Altium

Manufacturing Checklists and Tools

MANUFACTURING CHECKLISTS AND TOOLS

When it comes to manufacturing, organization is critical. With so many different players involved in the design process (engineers, sourcing agents, manufacturers, etc.), maintaining documentation and utilizing checklists and other tools will keep everyone on the same page and avoid confusion. Additionally, checklists can help ensure that the PCB design is fully complete and ready to send for manufacturing, protecting you from any unexpected surprises..

Join us as we discuss a variety of topics to help you with Manufacturing Checklists and Tools, including:

- Don't Settle for Better Late Than Never: Packing and Shipping Your PCBs
- Optimizing Your BOM for Rapid Prototype Manufacturing
- Remove the Chance of PCB Manufacturing Delays by Using a Final Deliverable Checklist

DON'T SETTLE FOR BETTER LATE THAN NEVER: PACKING AND SHIPPING YOUR PCBS

I mailed my last Christmas present yesterday, a few weeks after the holidays are over. I'm only a little embarrassed; I like to do a very precise job with wrapping even if I'm not terribly concerned with punctuality. The package was for a friend overseas who had a craving for her favorite American candies. Let me tell you: a beautiful wrapping job on a bag of fun-size Butterfingers is no mean feat.

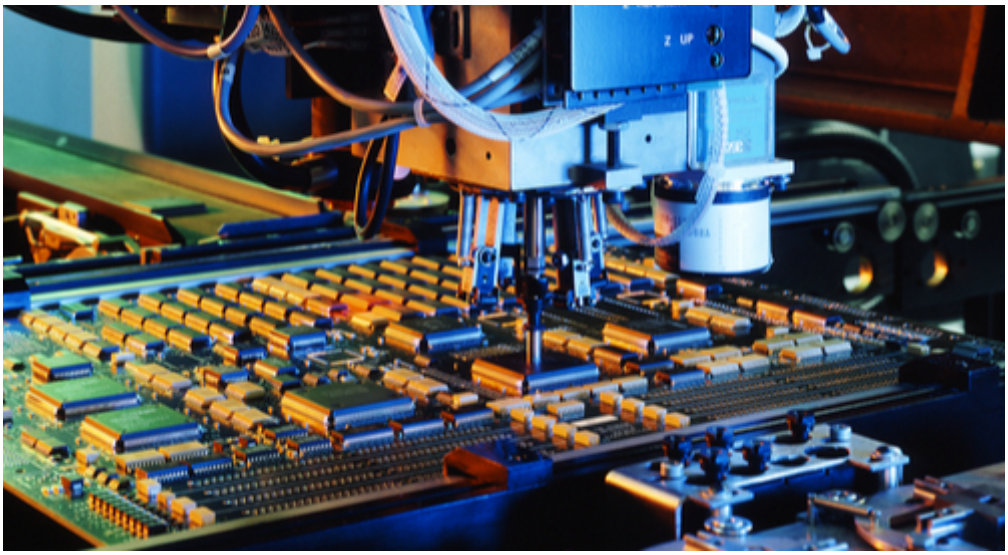
Wrapping and packaging are more than ornamental when you're packing up a PCB. Before you get anything produced by a new manufacturer, you should ask about packaging. You want to be sure that your board is adequately protected. If a Butterfinger is damaged in shipping, you still get delicious crumbs, but a PCB can't always be salvaged.

WHY DOES THE PACKAGING MATTER?

If you don't have an obsessive need for square corners and symmetric tape, then fussing over your PCB packaging probably seems like a waste of time. As long as the boards don't get crushed in shipping, everything is fine, right? PCB packaging is almost as specific as the PCB function. You designed your PCB to protect it from its expected operating environment and you need to wrap your PCB up to protect it from hazards of shipping and storage.

Shipping hazards range from the obvious dropping and crushing, but other mechanical damage can also occur. In extreme heat or cold, boards can warp, solder can creep and crack. You might have someone handle the packages who, like me, shakes boxes very aggressively to find out what's inside.

Less obvious damage sources are the same things you protect PCBs from: moisture and electrostatic shock. It's possible to get any, all, or none of the types of damage to your boards during shipping. It just depends on how well they're protected and how aggressively they're shaken.



Try not to shake your manufacturing machinery.

WHAT TYPES OF PACKAGING ARE THERE?

Packaging varies based on what you need to protect against. The protection is not always obvious just from looking at the wrapping, so it's helpful to have a little background:

Tissue paper and bubble wrap: Unless you're shipping bare PCBs (which just have a rubber band holding them together), this is probably the most basic packaging you'll see. This is fine for very simple, robust boards, but if you have more sensitive components, you probably want to step it up.

Pink poly: Pink poly is a type of plastic wrap that provides ESD protection. It comes as plastic wrap, bags, and bubble wrap. In addition to the cushion of a pink poly bubble wrap, it protects against shocks that occur when the box full of boards is packed and unpacked. You should be aware that the material is not compatible with certain plastics, especially polycarbonate, so select your materials accordingly.

ESD Bags: Many ESD sensitive components and some larger hardware will get shipped in ESD bags. They are shiny, silver plastic, and

often have a Ziploc style seal. You can get these in various sizes, and have your ESD sensitive boards packed into individual bags to keep them safe from static shocks during transit

Moisture barrier: If your PCBs are moisture sensitive (most are, at some level), or will be shipped by ocean freight, or through a humid climate, look at moisture barrier packaging. Your PCBs will be individually sealed, probably with a little packet of desiccant, to keep out moisture in the air, regardless of where the boards travel en route to you. Moisture barrier packaging is usually puncture resistant, too, so it gives a little extra protection to your boards.

Vacuum sealed: For the hardcore moisture sensitive applications, you can have all the air sucked out of the packaging around your PCB. I think it looks like a neater version shrink wrap, and of course, I prefer everything neat and tidy. You can also get vacuum sealed ESD packaging if you need protection on multiple fronts.

Vacuum sealing is used for food, electronics, and anything else that needs to be protected from air and moisture during shipping.

WHAT DO I NEED FOR MY PCB?

Choose a packaging that's going to provide adequate protection for your board. After designing and manufacturing, you don't want the boards to get damaged in transit. If you are receiving multiple boards in a single shipment then have them packaged based on their custom requirements. I wrap a drone very differently than a bag of candy bars, but I want them both to arrive unscathed.

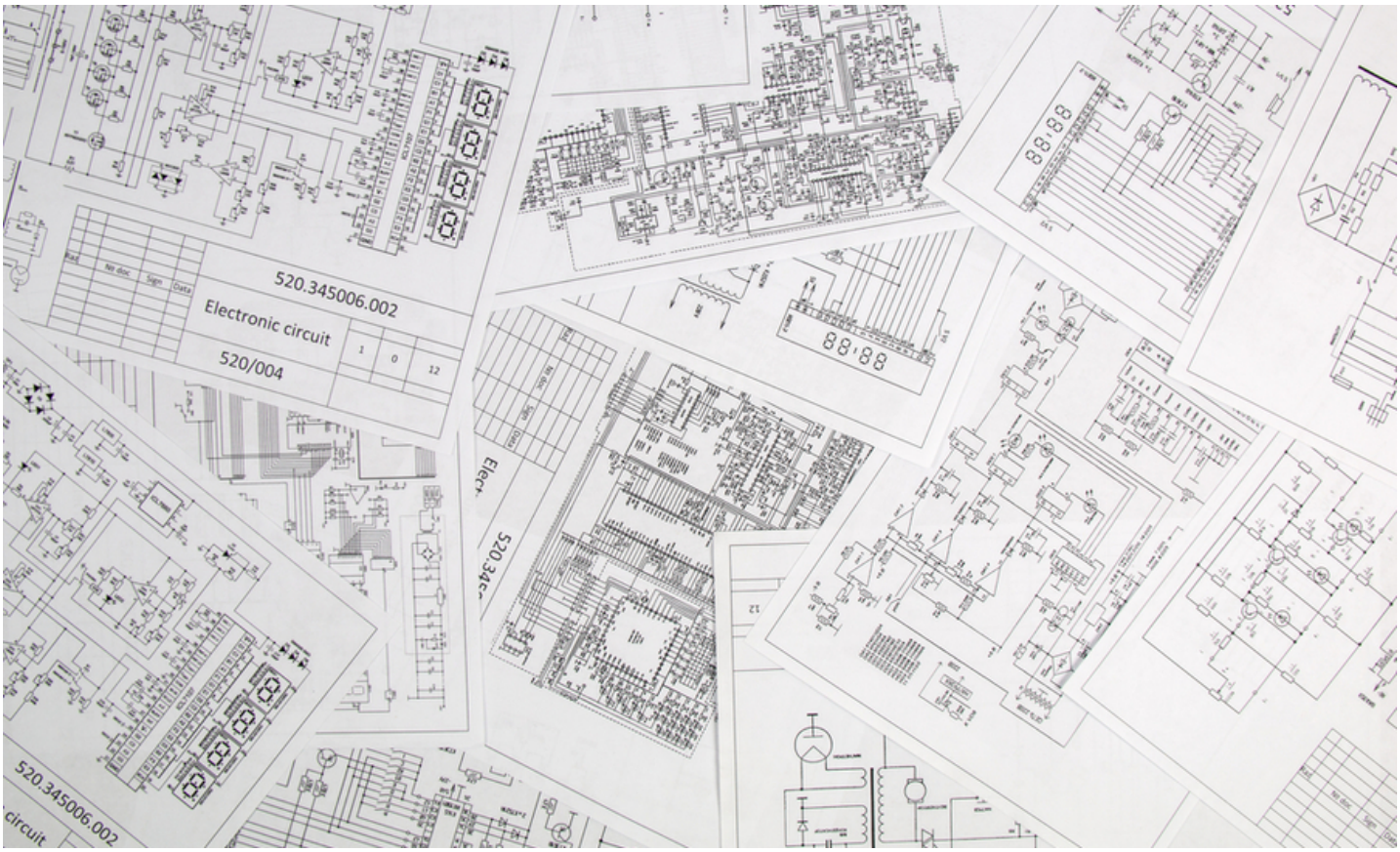
When you're getting PCBs manufactured ask what the standard packaging is. Find out about pricing for other options. If pink poly bubble wrap is enough, and they usually vacuum seal, you might be able to get a discount. If you have concerns, ask to get a sample of the packing material before they start manufacturing anything for you. If you are on-site for an audit, watch their packing methods, and take a good look at the materials they are using.

MANUFACTURING CHECKLISTS AND TOOLS

While not as fun as picking wrapping paper, making careful decisions on packing will keep all your hard work from going to waste right at the end. On the other hand, a good decision to make right at the beginning is which PCB design software to use. With easy and smart manufacturing outputs, intuitive drawing processes, and strong templates to work off of, Altium's CircuitStudio is a great option.

If you're looking to know more about your PCB manufacturing process, and how your design software can make it easier, consider talking to an expert at Altium today.

OPTIMIZING YOUR BOM FOR RAPID PROTOTYPE MANUFACTURING



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

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

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

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

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

BE THOROUGH, VERY THOROUGH

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

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

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

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

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

LABELING VARIABILITY

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

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

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

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

TRACKING CHANGES IN YOUR BOM

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

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

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

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

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

REMOVE THE CHANCE OF PCB MANUFACTURING DELAYS BY USING A FINAL DELIVERABLE CHECKLIST

Several years ago I had knee surgery. Something had happened to the membrane that provided protection in the joint, and my knee was making all kinds of noise that a healthy knee shouldn't be making. Before going into the surgery the doctor said that I should take a marker and carefully notate which knee required the surgery. Considering the circumstances, I hope he wasn't pulling my leg before cutting into it.

Using a checklist is always a good thing in life and in my mind anyway, having a checklist to make sure that the correct knee was operated on was extremely important. I keep checklists to make sure that my bills are paid on time, we get everything that we need from the grocery store, and that I've packed everything I need before I leave on vacation. But on a more day-to-day basis, I use checklists when I'm designing PCBs.

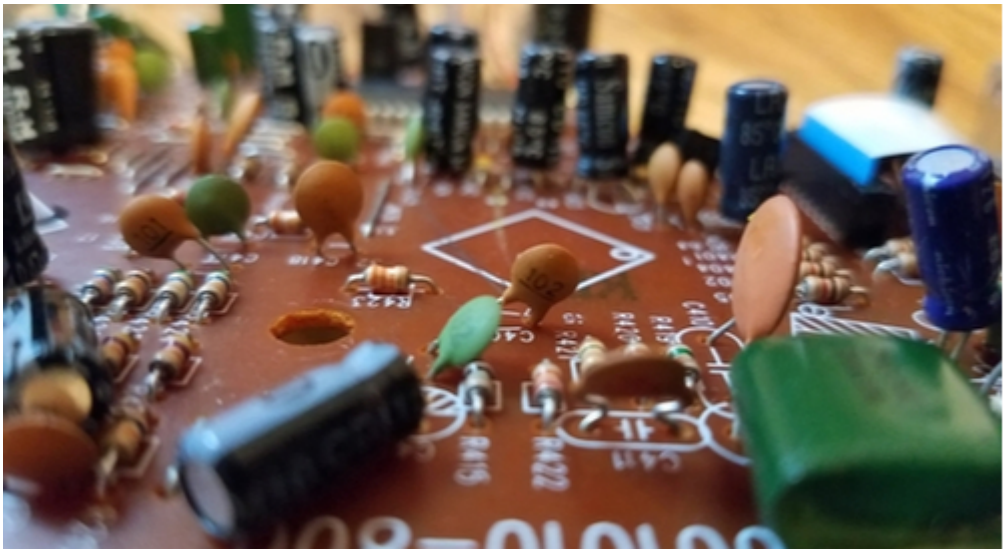
A pre-design checklist will make sure that you have everything that you need to start a design, while a design review checklist will make sure that all aspects of the design have been looked at. Perhaps the most important checklist of all though is the final deliverables checklist. If you aren't using a checklist for your final deliverables yet, here are some of the things that I check for that

might help you too.

YOUR DELIVERABLE CHECKLIST IS INCOMPLETE WITHOUT A FINAL DESIGN REVIEW

The first thing that I like to confirm in my final checklist is that the design review, or to be more precise, the final design review has been completed. Sometimes important steps like a final review get bypassed, especially when the design is either small or a simple re-spin. Yet you know that according to Murphy's law, those are the designs where something will go wrong.

With the final design review, you'll be able to rest assured that you've gotten your necessary approvals to move forward in the process. And, later on, if you are worried about having skipped a step, you'll be able to rest easy. Do yourself a favor, and put that final design review at the top of your list. Make sure that all of the requested changes have been made and that everyone with a stake in the design has signed off on its completion.



Without a checklist, updating part numbers and other PCB markings can be forgotten

KEEP THOSE PCB MARKINGS FRESH AND CLEAN

In addition to the output files that go to the manufacturer, there are many things that should also be included in your final deliverables checklist. I've seen plenty of designs come back from the manufacturer because these items were ultimately incorrect and had not been checked. In the interest therefore of being completely thorough, here are some of those things that I include in my final checklist:

- PCB part number markings: There may be different part numbers on a design like raw board numbers, assembled board numbers, etc. These numbers may be in etch or on the silkscreen or both. In a redesign updating these part numbers is often forgotten as it is usually a simple change of the last number from a -001 to a -002.
- PCB board names and revision markings: Like the part number markings, the board name and the revision may be changing, especially in a redesign. You should check these again as well.
- Miscellaneous markings: There will also be plenty of other markings on a PCB like copyright information and certification numbers. These all need to be looked at.
- Film title blocks: It is very common to catch all the changes in numbers, names, revs, and dates on the physical board, but then completely forget to make the same changes in the title blocks. Make sure to check those title blocks as well.

A final deliverable checklist will protect you against missing output files

MANUFACTURING DOCUMENTATION, THE LAST PIECE OF THE FINAL DELIVERABLE CHECKLIST

The last portion of your final deliverables checklist will be the actual manufacturing output files themselves. Not only should the different output files be listed, but also the media that they will be delivered in. Most manufacturers will require electronic media for the output files, but some may request paper copies as well. Here are some of the output files that you should consider for your checklist:

- Gerber files
- Aperture lists and Gerber configuration files
- NC drill files

- Manufacturing drawings (fabrication, assembly, panels, etc.)
- Bill of materials
- Pick and place

There are many other files that may be specific to your manufacturing process that I haven't listed here. These could include special build instructions or read-me files. Whatever your specific file needs are, make sure to include them in your checklist as well.

A checklist like this will help to ensure that your PCB design has been fully completed and that you have everything that you need to send it out for manufacturing. Protect yourself from unexpected surprises by following a checklist.

For your design needs that come throughout your design process, using Altium Designer's smart PCB design software will ensure your designs are complete and ready for manufacturing in a timely fashion. Altium Designer 18 comes with utilities that will output all of your files together in one master job file helping you to cross these items off your checklist all at once. With help like this, your board will be manufactured correctly and on time from beginning to end.

To find out more about how you can achieve your DFM compliant PCB design needs, consider talking to an expert at Altium.

ADDITIONAL RESOURCES

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

- Don't Settle for Better Late Than Never: Packing and Shipping Your PCBs
- Optimizing Your BOM for Rapid Prototype Manufacturing
- Remove the Chance of PCB Manufacturing Delays by Using a Final Deliverable Checklist