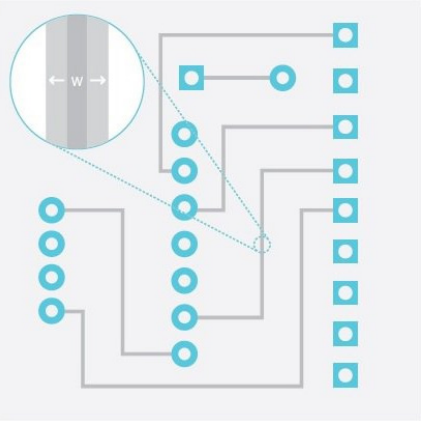
Advanced Assembly
SMT Assembly for Engineers

7 COST-SAVING STEPS FOR YOUR **NEXT DESIGN**

Large or small, every company wants to decrease costs without affecting the quality of their products or services. There is rarely a perfect way to achieve this and PCB design and assembly is no exception. Accurately assembling PCBs is not an easy task; it takes experience, technology and, above all, dedicated people to ensure your projects are done right. When searching for a low-volume assembly shop to populate your boards, it may be tempting to use a low-cost shop to save money. However, doing so may end up costing more in the long term due to reliability concerns and lower throughput, which could result in a damaged reputation. As the famous adage goes—you get what you pay for. With thoughtful planning, you can incorporate the following seven design principles and cost saving steps to bring the overall price down while still using a high quality shop and getting high quality results.

1. DESIGN WITHIN STANDARD SPECIFICATIONS



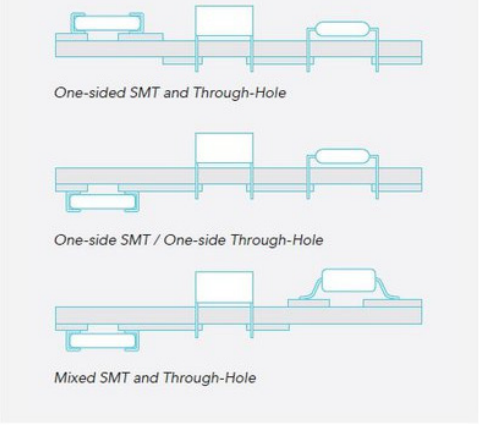
Avoid Design Limitations

By using industry standard sizes and components, the assembly team can use processes that have already been established and thoroughly tested. They will also be able to automate large portions of the process and avoid costly and time-consuming hand placement and soldering. Knowing what the assembly house is capable of is a matter of experience and research. Assembly houses should include their capabilities on their websites or provide a link in brochures which should answer most questions of what they can successfully handle. Any questions that are not answered by these sources can be directed to the company itself. The responsiveness of the company to your questions about their capabilities should be a good indicator of the type of service that you will receive if you become a customer.

When asked for capabilities, assembly houses will provide what they are capable of but not necessarily what is cost-effective. As a general rule, it's always a good idea to not push the design limitations as much as possible. If it is possible to use larger traces, or greater spacing between components, then why not do it? It may or may not make the process easier but if it does not affect the operation of the board, then it only has the potential of helping. Also, when speaking with the assembly house, ask if there is anything specific to their process that may help bring the overall cost down. These suggestions may have a considerable impact on the design of your PCB, so it may be a good idea to start looking into assembly houses towards the beginning of the design of the board once you have a general idea of what you will need.

2. CONSIDER THE COSTS OF MIXED MOUNT TECHNOLOGIES

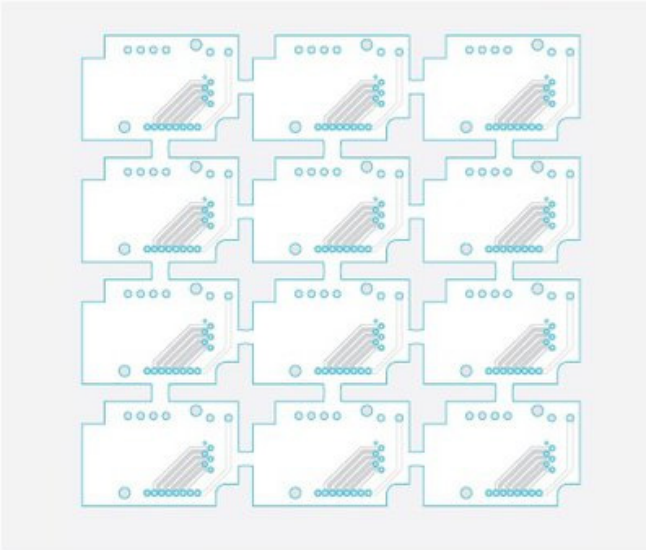
When building a board with solely surface mount technology parts, the boards typically have solder paste applied and are then populated and run through a reflow oven to complete the soldering process. Depending on the disparity of sizes between through-hole and surface-mount components, different soldering iron tips, temperatures, and solders may need to be used. However, when there is a combination, things become more complicated. There are three combinations of mixed technology: single sided SMT and through-hole, double sided with SMT on one side and through-hole on the other, and double-sided with both SMT and through-hole on either side. Each of these scenarios is dealt with differently, but require significantly more steps and have more stringent requirements than a single technology on the board.



Mixed Technology PCBs

For example, even placing SMT and through-hole components on one side requires the additional steps of applying an adhesive to the SMT components, allowing the adhesive to cure, flipping the board and then wave soldering it. As the solder bath temperatures can destroy some SMT components, you will need to verify that the SMT components are wave solderable, making your component selection criteria even more stringent. When there is mixed technology on both sides of the board, this becomes even more complex and frequently requires hand soldering of certain components after the automated process. This is not to say that you cannot mix technologies or utilize both sides of your boards for components, merely that you should weigh the benefits and drawbacks of each and make sure that mixing the technologies is worth the additional cost and assembly time for your application.

3. PANELIZE YOUR BOARDS



Panelized PCBs

The size of panel that is acceptable depends on the assembly house, but this information can save considerable amounts of money. By using the largest panel size possible, you can drop assembly costs by eliminating pick and place downtime.

It is in your best interest to find the largest panel size your assembly house can handle, and put as many iterations of your design on that single panel as possible. It should be noted that the benefits of panelizing boards are only applicable if it is the same design repeated multiple times on a single panel. Including multiple designs on one panel will not provide any cost savings as the panel will, in essence, be a larger, more complex single board.

4. MINIMIZE OVERAGES

If you are producing a low volume of boards, use an assembly house that does not require reels or major parts overages. If you only need 30 boards and only one or two of each component on each board, it does not make sense to buy a reel of literally thousands of parts. At the prototyping stage, components can be changed out quite frequently in the design and you may end up with a great deal of parts that serve no purpose and may have cost considerable amounts of money. Some assembly houses can machine place parts without needing large reels or tubes of parts. While researching the company, find what each is capable of handling—if they require you to purchase large reels for small prototyping runs, look elsewhere.

While you do want to minimize the parts necessary for your build, do not forget to include a small amount of overage in your requirements calculations. Parts, particularly the incredibly small 0402 and 0201 components, can be dropped by the pick-and-place machine or damaged at some point in the process, so your assembly house should have general guidelines on how many extra parts you need to provide. Excess parts after the assembly will be shipped back with the parts.

5. SEEK EXTERNAL REVIEW

Whenever possible, have someone else review your work. When your circuit boards are submitted to a PCB manufacturer or assembly house, trained technicians or engineers will look at the design and verify that it is manufacturable. This is an invaluable service that has saved many designs from basic errors over the years. However, these technicians and engineers are focused on the manufacturability and do not know the purpose of the board. By asking a member of your team to review your work and provide a fresh perspective, you reduce the risk of missing simple yet crucial errors.

Collaboration

6. TAKE ADVANTAGE OF FIRST ARTICLE SERVICES

With prototypes, even after thorough and repeated review, it is still likely that there will be concerns. With final designs, while less likely, it is still possible that there is some error that will be manifest in the end product. Many assembly houses will populate one board and send it out for review, testing, and verification before continuing on the process. If they offer this service, take advantage of it. Hopefully all is well with the board, but if not, it could save a tremendous amount of money to catch the problems before potentially expensive components are attached to boards with errors on them.

7. PROVIDE HELPFUL INFORMATION TO THE MANUFACTURER

If there are any special requirements or areas that you believe will cause concerns, contact the company and discuss it with them. They may be able to provide design advice that eliminates these concerns or maybe let you know that the design is unmanufacturable. Most likely, however, they will pay special attention to where you have indicated and perhaps make minor tweaks to their processes to ensure that it is done correctly. Knowing beforehand can save a lot of time and money on rework that could have been avoided.

There is rarely too much information provided when submitting boards for manufacturing and assembly. If there is information that is recommended but not required, include it when you can. For example, Advanced Assembly does not require fiducials in order to process and successfully assemble circuit boards. However, the fiducials are an additional help that could reduce the difficulties in the assembling process and could improve the final product due to increased accuracy. If it is possible to include fiducials on either the PCB or panel, do so.

CONCLUSION

It is possible to decrease costs without giving up quality. Instead of using cut-rate manufacturers and assembly houses, follow these low-to-no cost recommendations that will save you the money you need to make sure that your designs are produced to the highest quality possible. Never settle for less than the standards that you and your customers expect.

NEXT STEP: FREE CONSULTATION

Receive a free 15-minute, pre-production assembly consultation. Talk with our engineering team for PCB assembly questions such as how design elements affect manufacturability.

To schedule your consultation via phone or email visit: www.aapcb.com/ask-our-engineers/