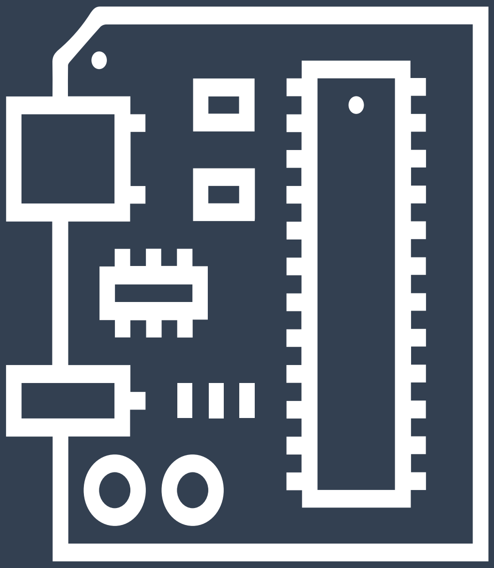
DFT
TESTING THE LIMITS



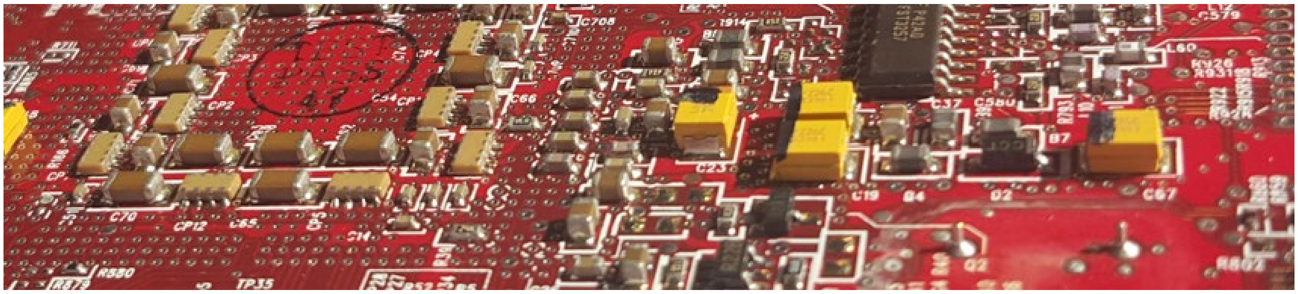
EMA Design Automation®

DESIGN FOR TEST:
WHY DOES IT MATTER?

To a design engineer, it is no secret that designing a PCB takes many steps, iterations, and a lot of time and hard work before it can come to fruition.

Within that process, ensuring your PCB designs are optimized for test can often take a backseat to higher priorities. Yet failing to implement proper Design for Test (DFT) accommodations within the early stages of your design cycle can lead to future consequences that may derail your project entirely.

A modern PCB has thousands of parts and components that must work together before being sent off to final manufacturing. As such, preparing your PCB design for test during the schematic and layout phase will ensure you are prepared for adequate circuit examination and debug capabilities when the time comes. In the manufacturing world, all processes and materials are tested for performance every step of the way, and PCB material does not move onto the next manufacturing step without it.



Checks, tests, and verification of design constraints should be done methodically before progressing forward with each major design step.

Within this ebook, we will discuss common testing tips, challenges, and misconceptions, as well as what should be tested within your design. Additionally, you will learn the overarching five W's of DFT.

THE WHO
IDENTIFYING YOUR COHORTS

Asking the right questions, identifying and engaging with your fellow stakeholders, and implementing checking are all ways to ensure success.

Each of your contacts will have different testing and verification requirements, so it is imperative that you keep an open dialogue with them to address major issues as they come up. For starters, your fabrication suppliers will be invaluable in testing for continuity and shorts on the bare circuitry. Electronic engineers will also have requirements for probing and debugging the assembled prototype circuits. From there, EMS test engineers will have requirements for testing overall quality in volume manufacturing.

Each of your fellow stakeholders has an important role:

Your ***fabrication suppliers*** will need to receive manufacturing data and use it to process the bare PCBs using Flying Probe test equipment. These probes use tangential test points on the bare board to find shorts and verify continuity between traces and inner layers before use. This is a great way to mitigate large errors before going too far in the manufacturing process.

Electronic engineers are the most familiar with the design, as they are the ones who create often-complex front-end criteria to its function. As they work, EEs can often foresee the need to add test points onto a PCB for measurement and verification of circuit function. These test points are generally added at the schematic level for functional testing near a specific component or circuit that may need debugging or fixing later in the cycle. The addition of this level of testing is sometimes enough to allow for manual testing of the assembly, ultimately saving time and money by replacing the need for more expensive in-circuit test strategies.

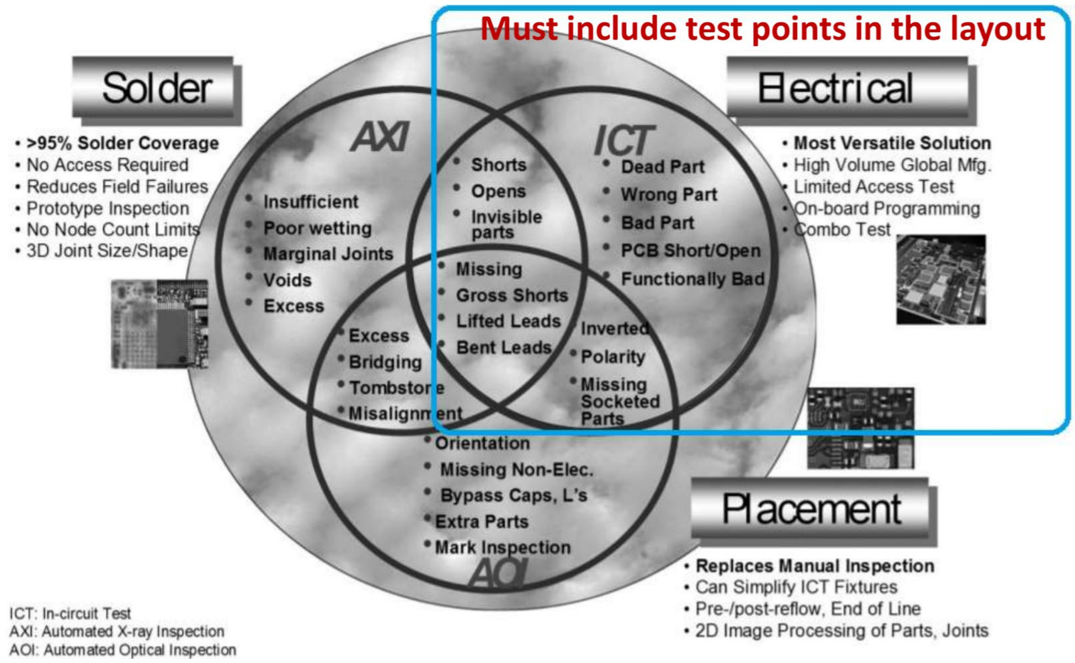
Lastly, ***EMS test engineers*** work for the electronic suppliers to implement as much testing as possible to collect data about the design as it runs through its processes. This step is a quality check to ensure that all assemblies coming off the line work as directed. The test engineers work with design engineers to understand circuit functions and expectations from the design. They also work with manufacturing engineers to implement testing for manufacturing defects that may occur on the PCB. This is especially important, given they have eyes into something an electrical engineer may not, and are the last line of defense before the PCB is put out into the world. EMS test engineers are the ones who help develop the in-circuit test fixture and routine software, so they require access to all nets on the PCB, preferably from a single side.

THE WHAT

UNDERSTANDING PROCESS MODES

There are three common process modes to testing the PCBA:

1) Automated Optical Inspection (AOI)
2) Automated X-Ray Inspection (ASI)
3) In-Circuit Testing (ICT)



This diagram from the SMTA/TMAG TP-101E guidelines provides a good illustration of the various failure modes each process tests for, and how these processes overlap in the data they gather.

The test modes work together to create an overall comprehensive picture of your design and its inner and outer workings. Conversely, you can use this diagram to see everything that can easily go wrong throughout design and manufacturing. Any one of these points can spiral into a massive delay if they are not properly accounted for early enough in the process.

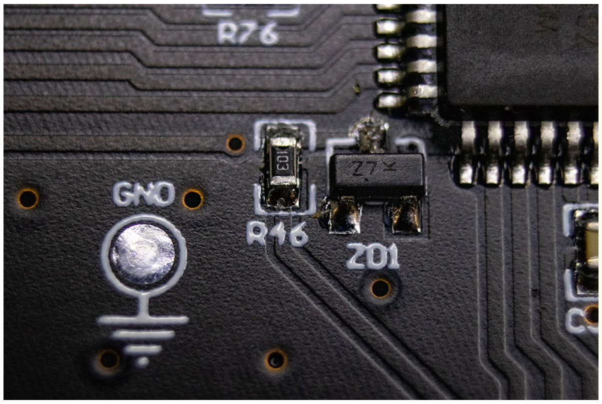
It is good to keep in mind that AOI and AXI are mostly mechanical inspection checks by nature, while ICT is an electrical test to check for electrical and mechanical quality. Therefore, test points are extremely important in the layout stage, as they are the key to ensuring a symbiotic relationship with the rest of the project production.

TEST POINTS

THE BACKBONE OF DFT

Let’s bring it back to basics for a moment:

A *test point* is a small, metal terminal used as a connection point to test the circuitry on a PCBA. Their sizes are dependent on many factors, including accuracy of tooling, size of board, the manufacturing process, and probe shape and tolerance.



Test points are utilized with a *test probe*, which are spring-loaded pins that mount into an ICT fixture.

The mounting makes an interface connection between the PCB and the fixture, allowing for data to be gathered and any mistakes to be found. The probe head and test point combo can come in many different shapes, sizes, and lengths to best suit the available real estate in which it’s located. The many types of probe heads are also designed to interface to test points other than pads; they can also be connectors or other PCBA features that were tagged or identified in the layout. It may help to do some legwork and research which type of test point and probe will best work for each area you need to reach for testing on the board. Your later self will thank you for the work done upfront in this matter.

THE WHEN
STAGES TO IMPLEMENT DFT

Though it seems obvious, it cannot be overstated that test points and overall testing foresight should be considered right from conception.

The PCB conceptual stage is the best time to address initial DFT concerns because they will be fresh in mind as you start designing the layout. Here, it is the PCB designer's job to be vocal and ask the important questions regarding the logistics of testing- and follow up on them! Oftentimes, when reviewing why a project was delayed or failed it was found these questions were overlooked, waved away, or not properly addressed when they should have been.

Some good questions to ask:
+ What is the risk of failure?
+ What is the class of the product?
+ Does your device require testing in multiple stages?
+ Will the design go to volume production and an EMS provider?

...and any other major questions that may affect the test point layout. These questions may get you started, but they cannot be determined in a vacuum. The PCB designer must seek stakeholder council in determining needs for your board well before design has commenced.

That said, take note that not all your stakeholder's needs will be met in this initial meeting.

Rather, this conceptual stage is meant to form the rough skeleton of what your design can look like, and that structure is likely to change and warp throughout the process as different problems, requirements, and concerns arise and are addressed. The ever-changing nature of a design turnaround often ends up looking much different than your initial sketches, while still maintaining the core structure you hashed out in concept. Therefore, it is highly important for DFT updates and check-ins to continually be implemented at regular intervals throughout the mission. This way, no stone is unturned or missed when the final project goes to manufacture, and perhaps even more importantly, your design can make it through the process with first pass success. Especially for larger-volume outputs, the reduction of costs and time makes your initial team approach planning a true investment in the ultimate success of the project.

THE WHERE
LOCATION, LOCATION, LOCATION

L*ocation and size matters* when it comes to DFT placement.

Failure to implement guidelines with all the stakeholders in mind will result in difficult challenges down the line. As such, the design engineer must become familiar with all guidelines necessary for testability on the board, and then work with your stakeholders to determine where and how to best fulfill them. The SMTA/TMAG TP-101E guidelines are available to view from the SMTA organization to help you on this journey. Be sure to confirm them with your stakeholders to ensure everyone is on the same page.

Test points can be located anywhere *only if* they are accessible after assembly and wont interfere with mechanical parts or objects.

ICT test points have basic locational parameters to ensure they are cost effective and work as directed. Therefore, proper layout research must be done before placement is complete. Generally, the more complicated the test fixture needs to be, the more expensive the process becomes. Start with bottom-side test access and avoid probing both sides if possible. The test point placing should be pre-meditated, as having a strategy will keep things orderly during the automated run.

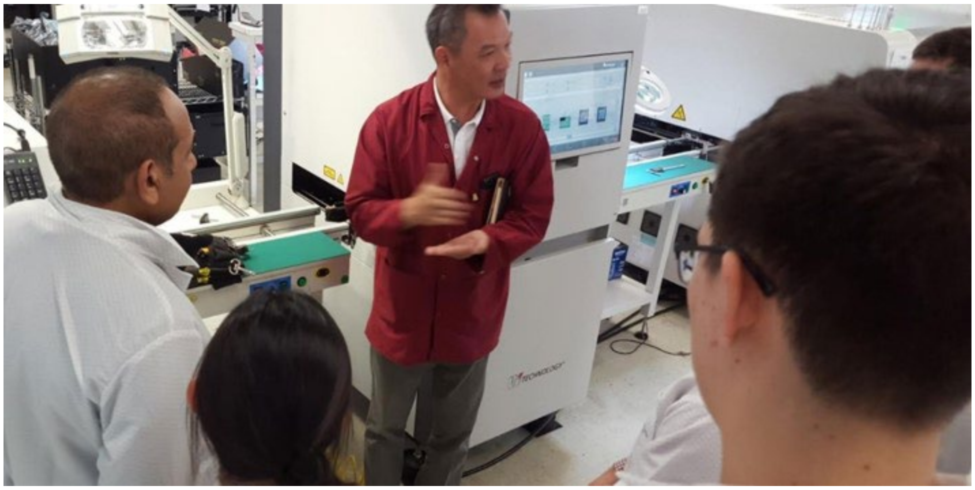
Designers will also need to account for test point sizing. With a significant upfront cost to build the fixture, the possibility for a test probe miss needs to be negligible to prevent a respin. One way is to select a large enough pad diameter to accommodate the probes without compromising the PCB topology. An .035 top-side or .030 bottom-side pad diameter is best, as testability can be a double-edged sword in the balance between providing useful insight into your design and staying out of the way of the working mechanics of the product. Stubs may need to be removed to provide room for a test point, but forethought into how this may affect overall performance must be considered before doing so. A good rule of thumb is to use a pad slightly larger than the minimum requirement, and to render the shape into a square to increase its area. Again, the SMTA/TMAG TP-101E guidelines have charts to help guide sizing choices, including specs for pad positioning and minimum placement distances.

When you are ready for testing, keeping your probe-to-board edge distance over .125 will ensure components do not interfere with the vacuum seal and seating of the probe. This will also provide a longer life for the gasket and is generally good for conveyorized design.

THE WHY
GOOD DFT YIELDS GOOD RESULTS

To quote a cliché: *teamwork makes the dream work.*

And this concept is never more apparent than when it comes to DFT in a PCB design process. It is a good thing to step back from the computer and ask yourself: when was the last time you spoke to or visited your EMS provider? Visiting a physical supplier (either live or virtually) and taking a tour will ground you in the myriad of processes at work beyond your computer screen. Many EMS providers are very welcoming into their world, and it can be an eye-opening experience to observe their test processes at work. Taking this opportunity can help you understand why these testing steps are so important to the overall mission of PCBA production.



As you tour their facilities, speak up and ask them questions, such as their greatest challenges or things learned in past projects.

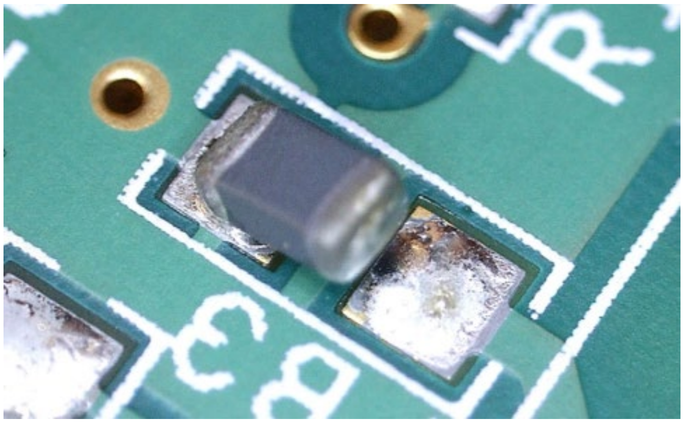
Experiencing their point of view will give a greater insight into why certain ways of doing something may work, while others will not. If you have a design running through them now, ask the directly what challenges they are facing with it and how these issues can be mitigated in the future.

The overall objective to DFT is to simply detect failures and fix them. Your EMS provider has likely "seen it all", and by opening your eyes and mind to past mistakes and lessons learned, you can gain insight into how to best move forward in your next project.

THE HOW
LEVERAGING PCB LAYOUT TOOLS FOR DFT

An experienced electronics engineer knows that when things go wrong in a PCBA, they can go very wrong.

Electronic failures such as skewed chip components, tombstoning, and short circuits are all reasons why in-circuit testing can be critical to a successful end-product. By tracking these failures, you can gather compelling data to justify an appropriate fix to consistent problems. In some cases, this could mean a redesign to your PCBA.



But what if you caught these issues early in the process?

Currently, PCBA testing is generally thought of as a necessary evil. However, in the world of new product introduction, it's a fact that incorporating in-circuit test goes together with high volume production. Utilizing DFT practices throughout the design process can ensure that the needs of the customer are met, product health is monitored, and data can be collected to improve your future design projects.

With the DesignTrue DFM tool built into OrCAD Professional and Allegro PCB Designer, you can set up and incorporate DFT for in-circuit test from the get-go. Our software allows for designation of pad stacks as test points, automatic test point placement, and real-time DFT reporting on any red flags or issues that may occur as you design. Ensure the highest quality to your design using ideal DFT practices with DesignTrue DFM today.

More info on DesignTrue DFM, or contact us at info@ema-eda.com.

EMA Design Automation®