

Guidelines for PCB design (for test)

There are certain rules when it comes to designing a new PCB, which would be good to implement in a new PCB. Below it will be explained how to design the print board, so the test fixture is as cheap as possible and as reliable as possible.

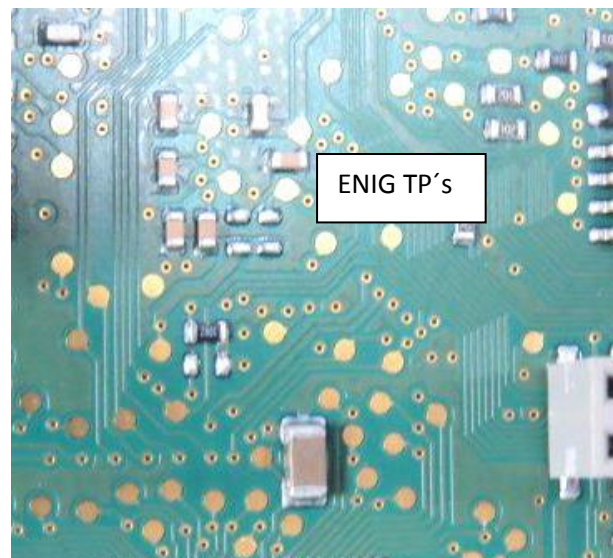


Mechanical Considerations for testpoints:

Ideally, all testpoints (TP) would be tested with the standard POGO 25 (ECT) spring probes (pitch 2,54mm). These low-cost and reliable probes are designed to have a pitch of min. 2,54mm between them, and are the perfect size for both test and manufacturing. If it is not possible to have the luxury of 2,54mm spacing, it is possible to go as low as 1,27mm pitch. Note that as the probe size decreases, the cost increases for both the probes but also the manufacturing of the fixture, but also obtaining high reliability becomes more challenging.

The size of the TP is also crucial for a reliable testfixture. If it is possible, then as big as possible, but the most standard size for a TP is $\varnothing 1,2\text{mm}$. The smallest size of a TP is $\varnothing 0,8\text{mm}$ it should be no less than $\varnothing 0,8\text{mm}$. If it gets smaller it is not possible to guarantee the probe will hit the TP, and it makes the fixture more expensive.

The material of the TP can vary from ENIG (Electroless Nickel Immersion Gold), component pin, and HASL (Hot Air Soldering Leveling). We recommend using the ENIG, because it has the best oxidation resistance. Because the ENIG does not oxidate as much as other TP surfaces, it does not have a hard surface to penetrate as the HASL for example has.



If the PCB has a lot of TP's, it can be a challenge to make sure the printboard does not bend a little when testing. It will not be a problem if it is not ENIG, but we have to take some precautions when designing the fixture.

Test point location.

All TPs should be located on one side of the PCB. That makes sure the test is simple and a lot cheaper than a multiple side connection test fixture.

TP distribution is important, as well. A high density of TPs or clusters should be avoided. TPs should be uniformly distributed throughout the board, making it easier to test with multiple probes and not having to rely on a specific area for all the probes to concentrate on.

If it is possible keep all components higher than 4mm on the opposite side of the TPs.

Good example of design for test.

In the picture you can see a great example of a PCB designed to make the test easier. The blue arrows indicate the component pins from the relay on the opposite side. And the red is the TP's the designer has placed on the board. The PCB designer has made

