

## Rework Instructions for the Module Tester PCB v2

The following rework is necessary for all PCBs fabricated using Gerber file:

Gerber\_PCB\_StearnsMTvrs\_v2\_2022-06-29c.zip

The board silkscreen shows: "file: PCB\_StearnsMTvrs\_v2"

It's recommended to locate all cuts so they are visible after components are installed, if that is possible. This is so the cuts are visible later if there is an issue.

Cuts on the component side (Layer 1):

1. Cut trace between PC1-10 and PC1-12.
2. Cut trace between PC1-12 and PC1-14.
3. Cut trace between PC1-14 and PC1-16.
4. Cut trace between PC1-16 and PC1-18.
5. Cut trace between PC1-20 and PC1-22.

Cuts on the circuit side (Layer 2):

1. Cut trace leading to PC1-20.
2. Cut trace between IC5-11 and IC4-11, close to C6.
3. Cut trace between IC4-11 and IC3-11, close to C5.
4. Cut trace between IC3-11 and IC2-11, close to C4.
5. Cut trace between IC2-11 and IC1-11, close to C3.

Add jumpers on the circuit side (Layer 2):

1. Jumper from anode of D1 to PC1-20. This is a +5V VCC trace so it can be a thicker jumper.
2. Jumper from IC1-11 to PC1-18 (signal CS-IC1-N).
3. Jumper from IC2-11 to PC1-16 (signal CS-IC2-N).
4. Jumper from IC3-11 to PC1-14 (signal CS-IC3-N).
5. Jumper from IC4-11 to PC1-12 (signal CS-IC4-N).

Image showing 5 cuts on the component side (Layer 1)

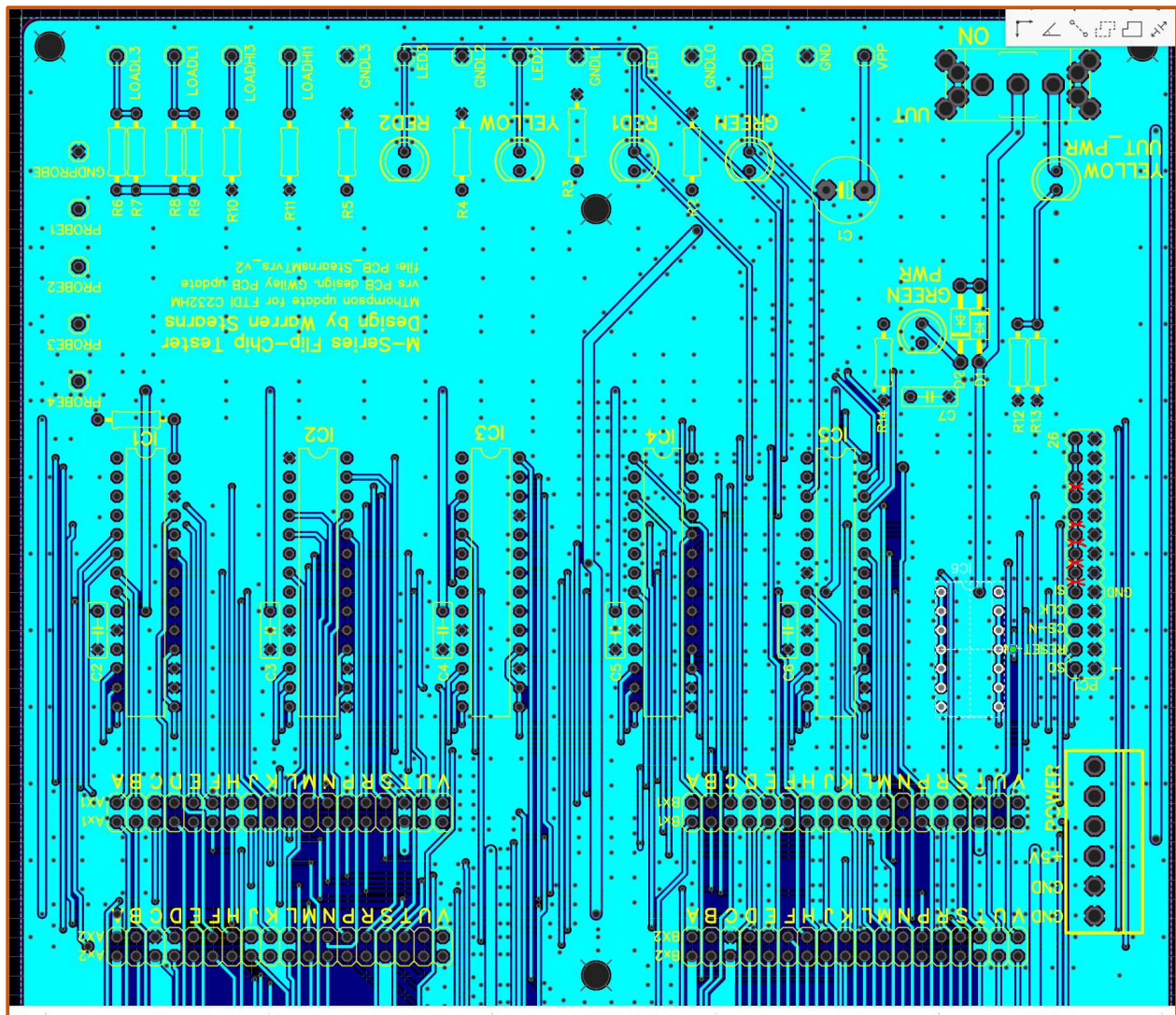


Image showing 5 cuts on the circuit side (Layer 2)

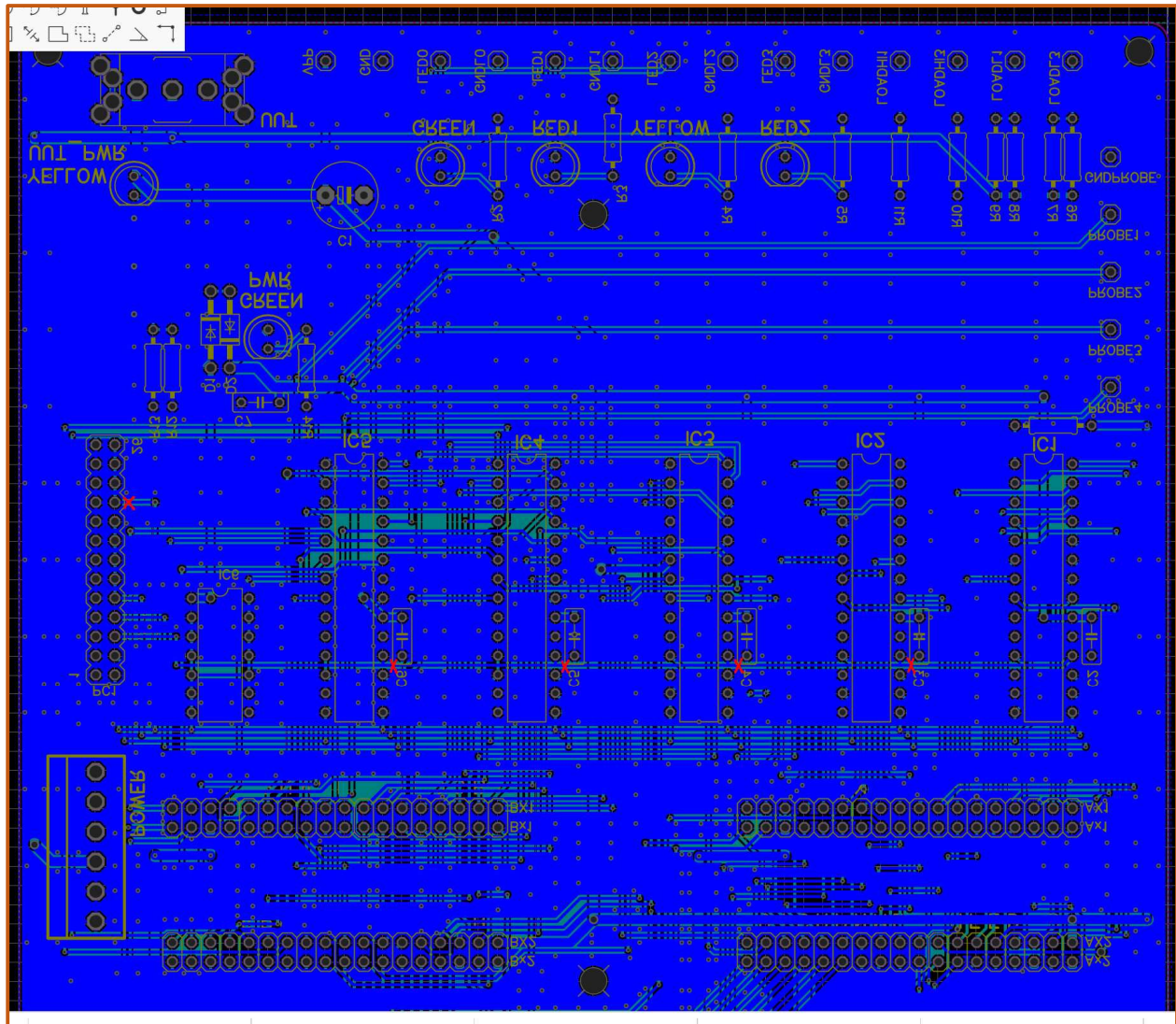




Image showing 5 jumpers on the circuit side (Layer 2)

