Lecture 10 - FAB/DEBUG

How are PCBs Made and Identifying and Solving Circuit Issues

IAP 2026

Outline

- **Fabrication**
- **Assembly**

Fabrication

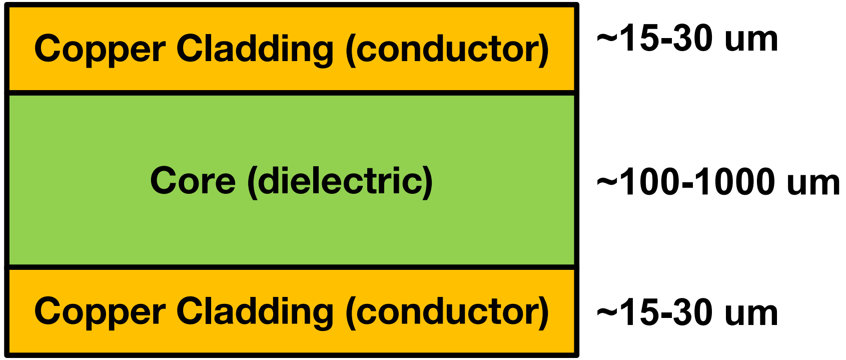
How PCBs are Made

First one starts off with a "core" material → rigid dielectric substrate material

- **FR4, RO4003C, RO4350B, MT40**

These core substrates will be sold with copper attached to (typically) both sides of core material → copper-clad laminate

Copper claddings can either be rolled annealed (RA) or electrodeposited (ED)



The core substrate provides a rigid structure for the PCB.

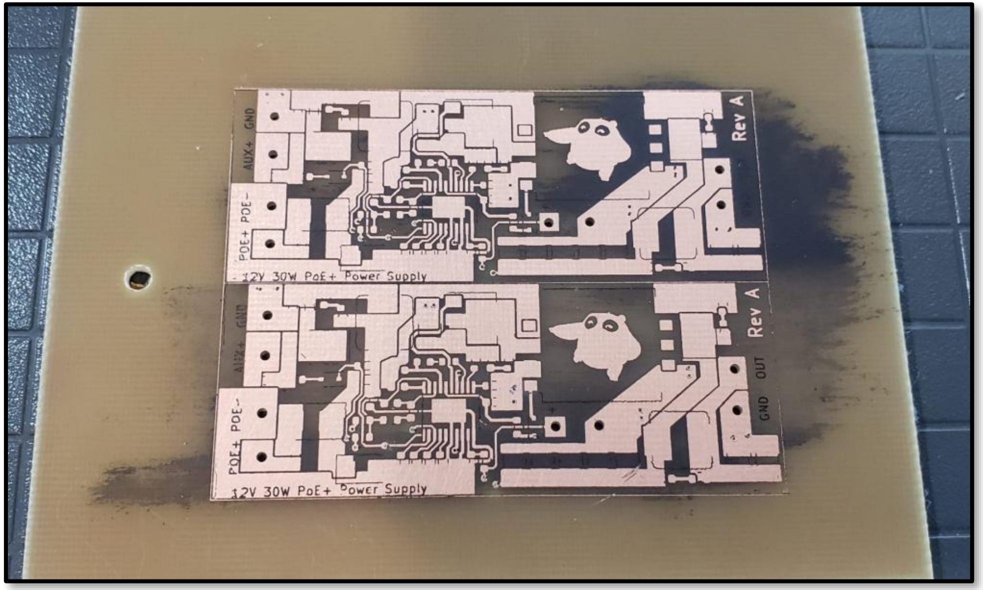
- **For a 2-layer board, this is the whole PCB structure**

Fabrication

How PCBs are Made

Bare Sheet

After Etching



Fabrication

How PCBs are Made

IAP 2026

Rolled Annealed (RA) vs. Electrodeposited (ED) Claddings

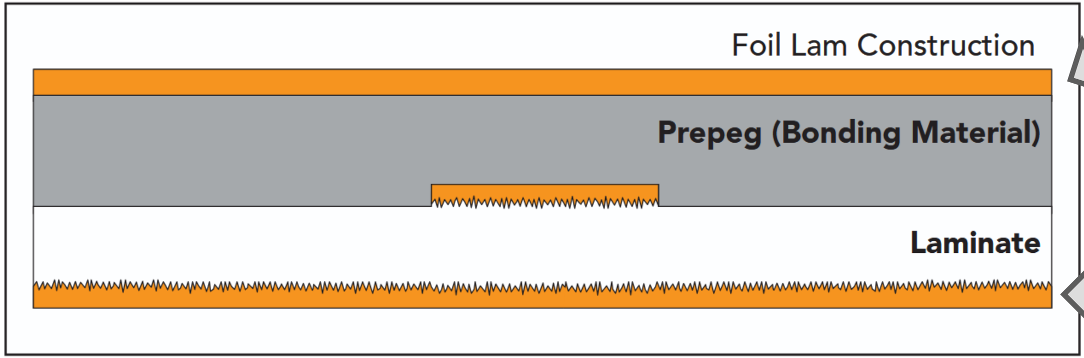


Fig. 2 A depiction of a foil laminated construction with only 2 of 4 copper-substrate interfaces the same in a stripline transmission line configuration.

Rolled foil copper tends to have much smoother interface with dielectric

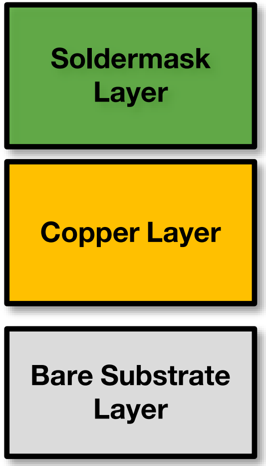
Electrodeposited copper has a rougher surface, but greater mechanical bond

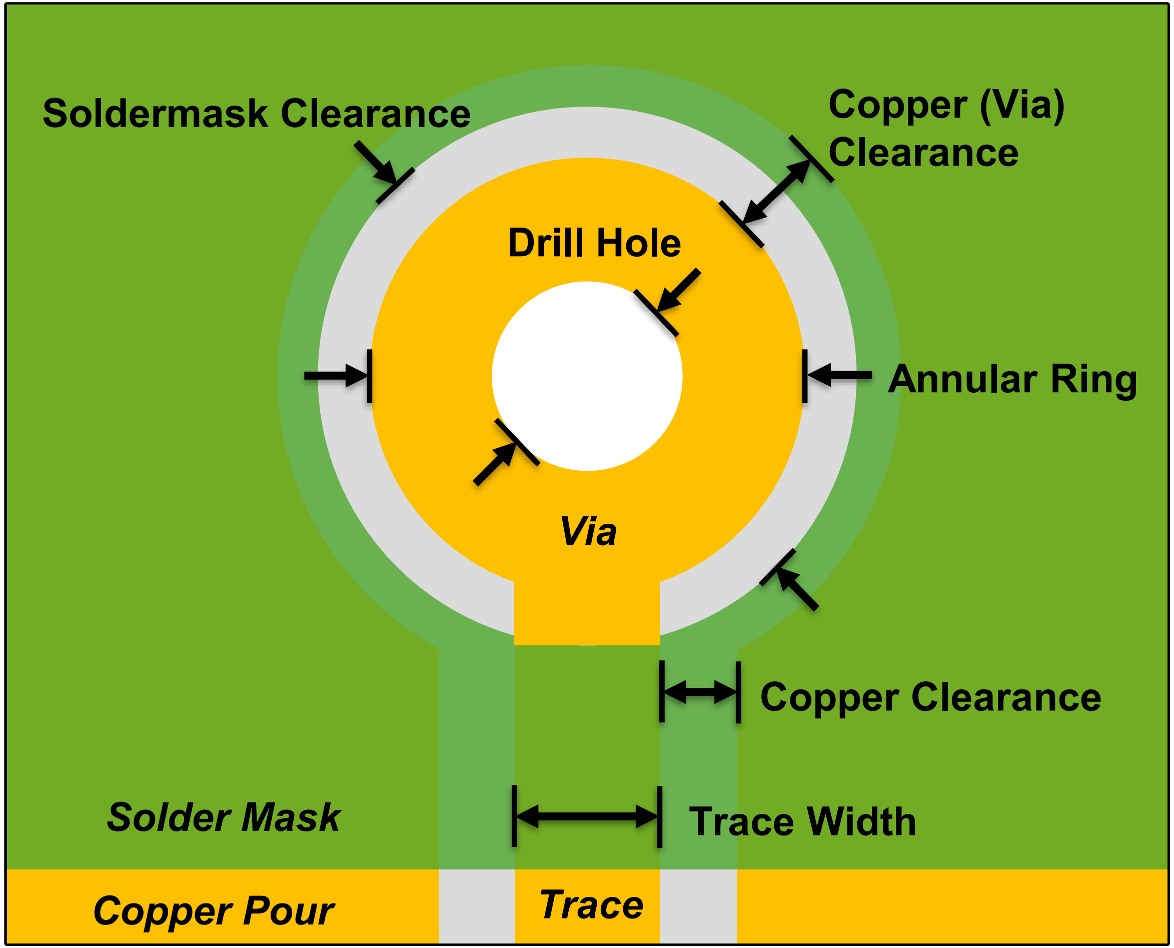
"Fabrication & Design Considerations For RF And High-Speed Digital Multi-layer Boards With Thin Internal Layers." Rogers Corp. Dec. 2024.

Diagram

THE ART AND SCIENCE of PCB DESIGN
IAP 2026

Recall…





@ MIT

Fabrication

How PCBs are Made

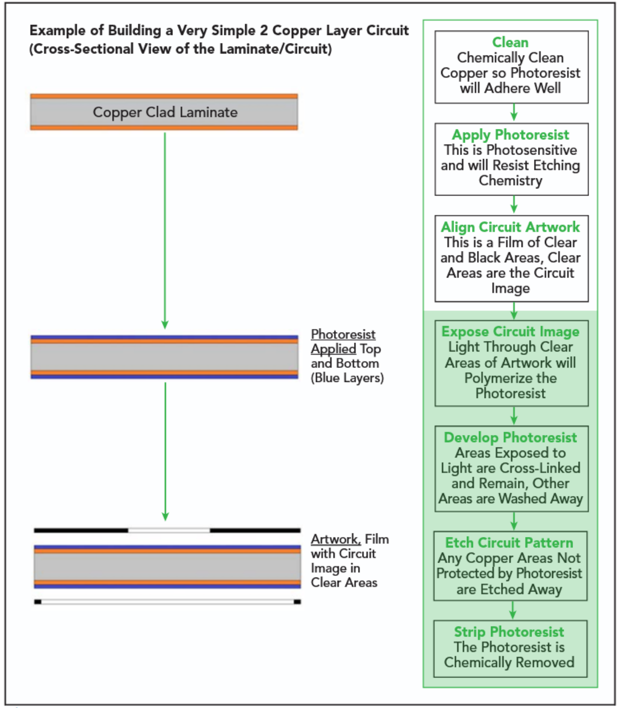


Fig. 1 The start of a simple process flow for building a 2-layer PCB, highlighting the first three steps.

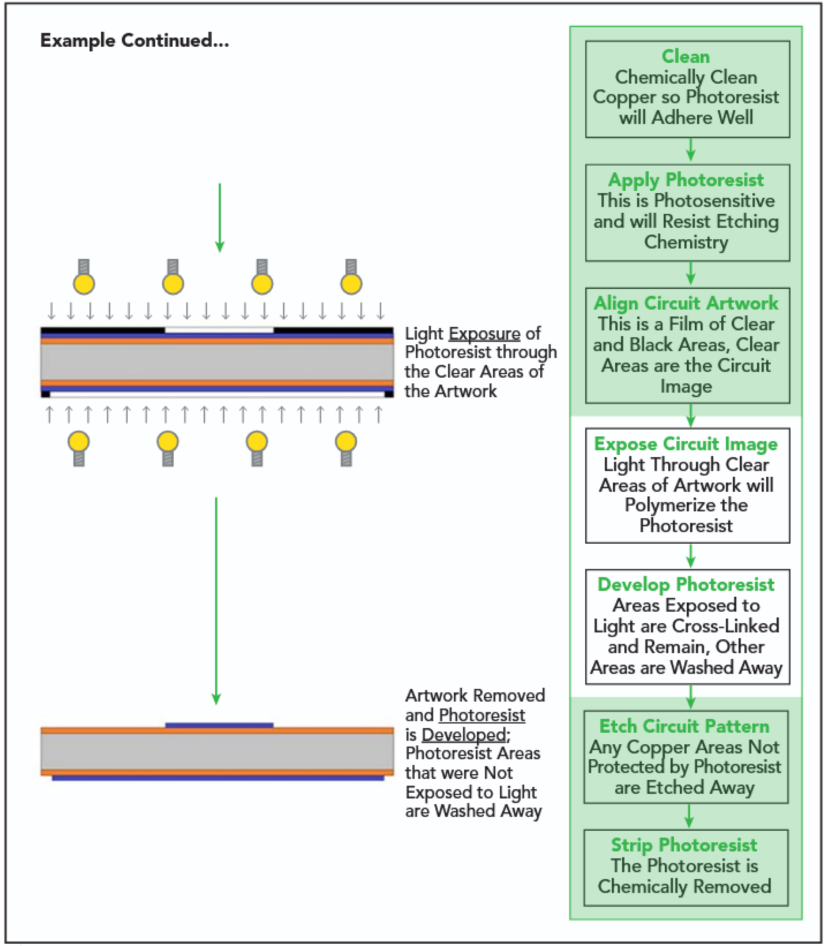


Fig. 2 Continuation of the simple process flow for building a 2-layer PCB.

THE ART AND SCIENCE OF PCB DESIGN

IAP 2026

Note: core material is used for 2-layer boards

J. Coonrod, "*What RF Engineers Need to Know About PCB Fabrication for Optimal Results.*" Rogers Corp. Dec. 2024.

@ MIT

Fabrication

How PCBs are Made

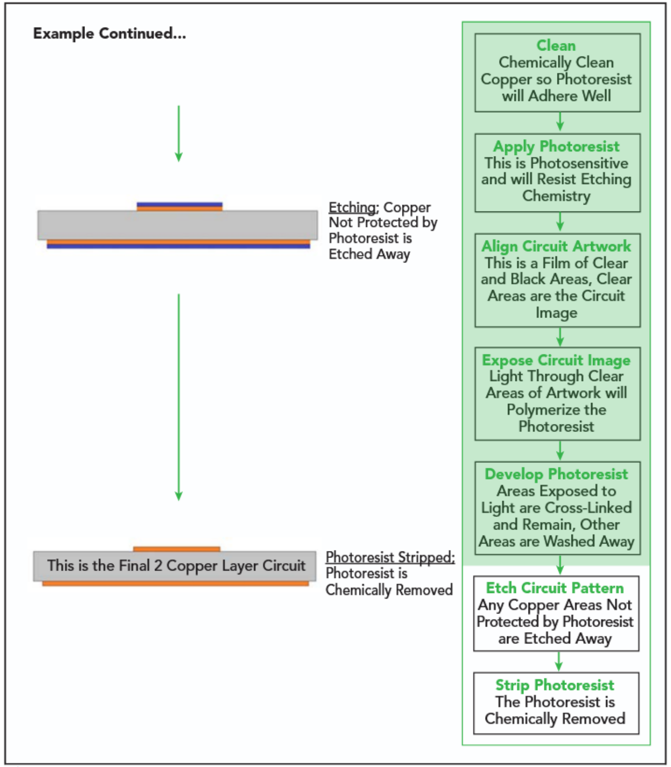
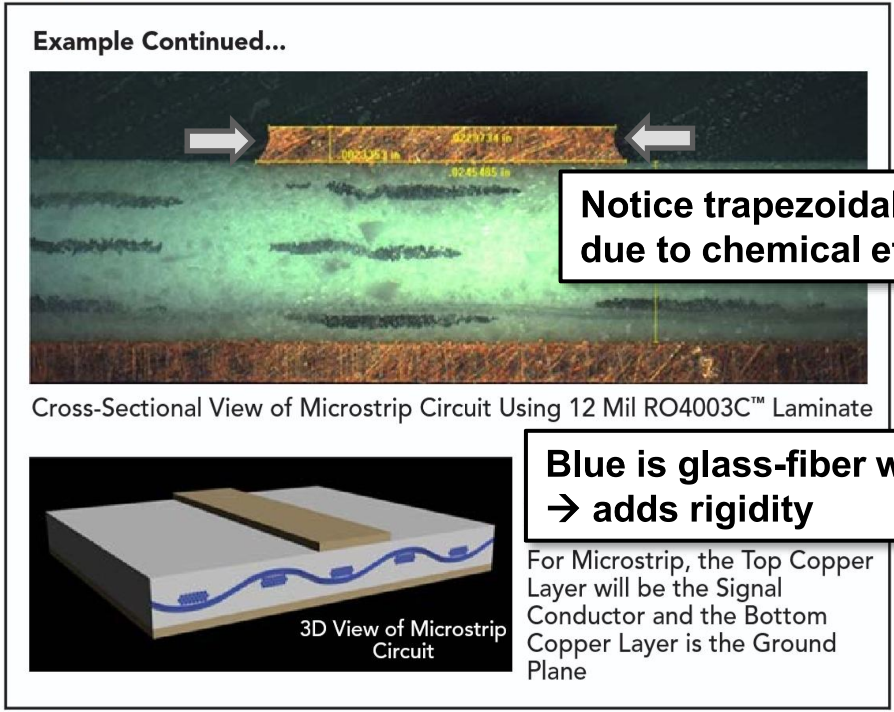


Fig. 3 Continuation of the 2-layer PCB process flow.



Notice trapezoidal shape due to chemical etching

Blue is glass-fiber weave → adds rigidity

Fig. 4 Conclusion of the 2-layer PCB process flow.

J. Coonrod, "*What RF Engineers Need to Know About PCB Fabrication for Optimal Results.*" Rogers Corp. Dec. 2024.

THE ART AND SCIENCE of PCB DESIGN

IAP 2026

@ MIT

Fabrication

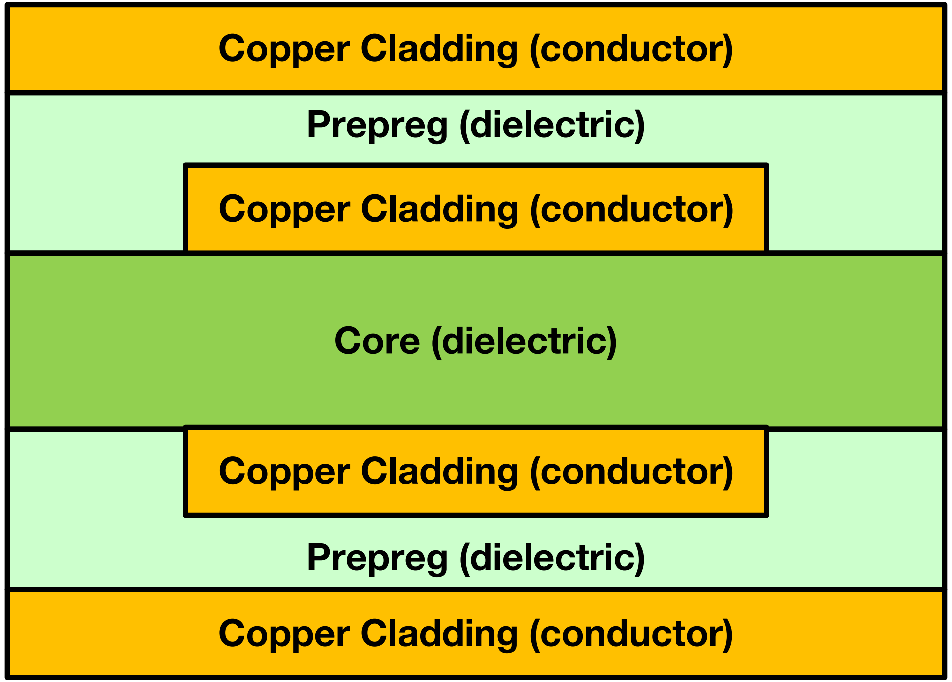
How PCBs are Made

THE ART AND SCIENCE of PCB DESIGN
IAP 2026

Multilayer Boards?
→ Use prepreg to laminate additional copper layers to existing core material

Prepreg is dielectric material consisting of some glass fiber weave encased in resin

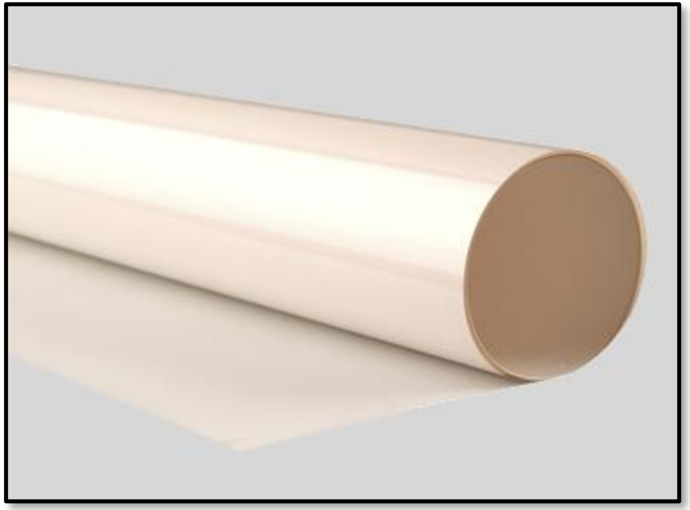
- Prepreg can be laminated onto a core material or other prepregs in a PCB
- Using heat + pressure, resin melts to bind prepreg
- Prepreg will cover the traces on the core it's laminated to.
- On top of the prepreg, we also laminate some more copper



Fabrication

How PCBs are Made

Prepregs can come as films (bondplys)



Fabrication

How PCBs are Made

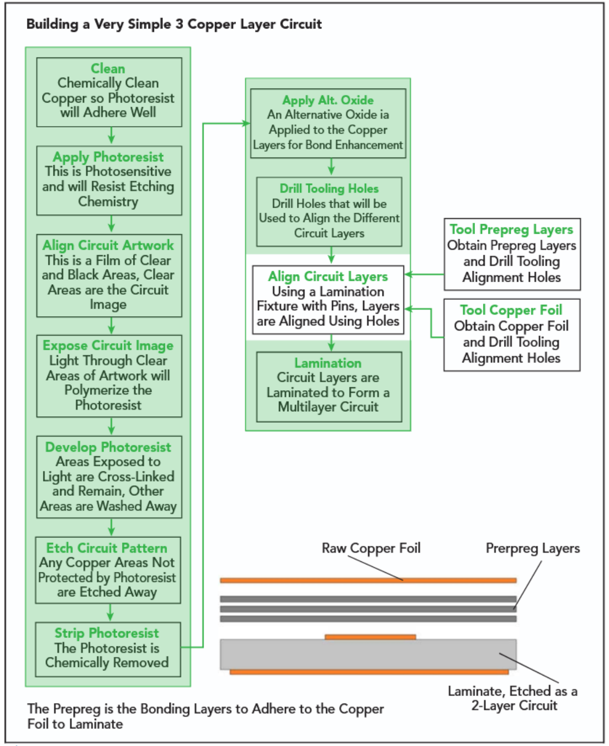


Fig. 5 Example of a 3-layer PCB fabrication process, starting with the 2-layer PCB fabrication method at left.

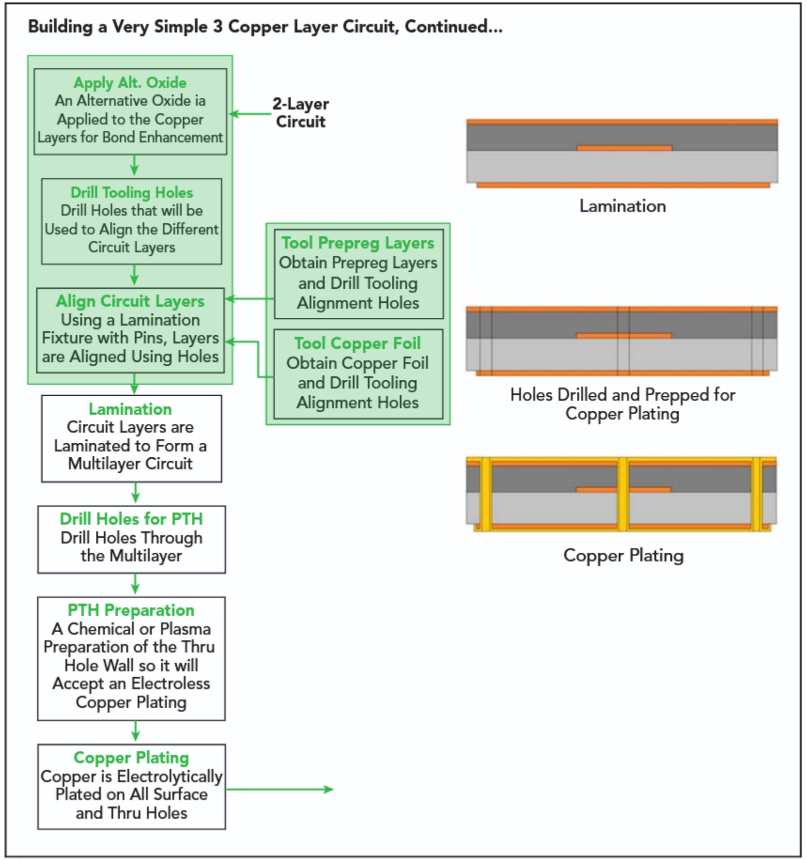


Fig. 6 Continuation of the 3-layer PCB fabrication process.

J. Coonrod, "*What RF Engineers Need to Know About PCB Fabrication for Optimal Results.*" Rogers Corp. Dec. 2024.

THE ART AND SCIENCE of PCB DESIGN
IAP 2026

Fabrication

How PCBs are Made

Prepreg layer (darker grey) wraps around copper trace because of lamination

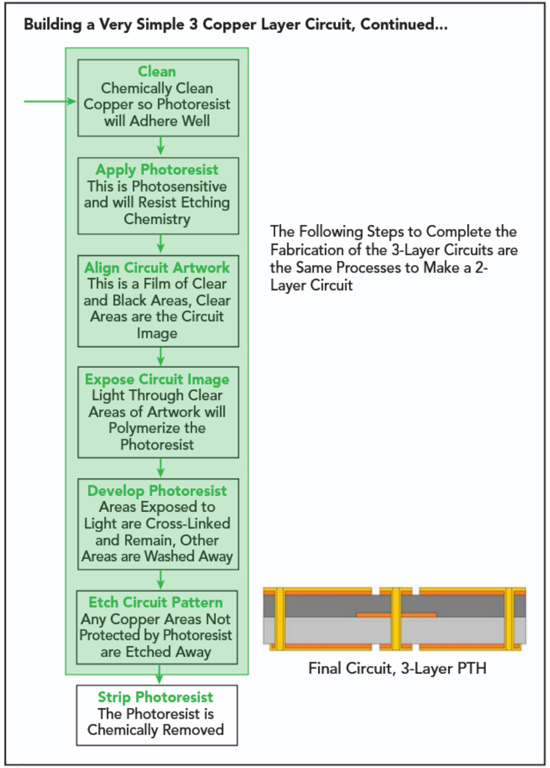


Fig. 7 An example 3-layer PCB with PTH vias.

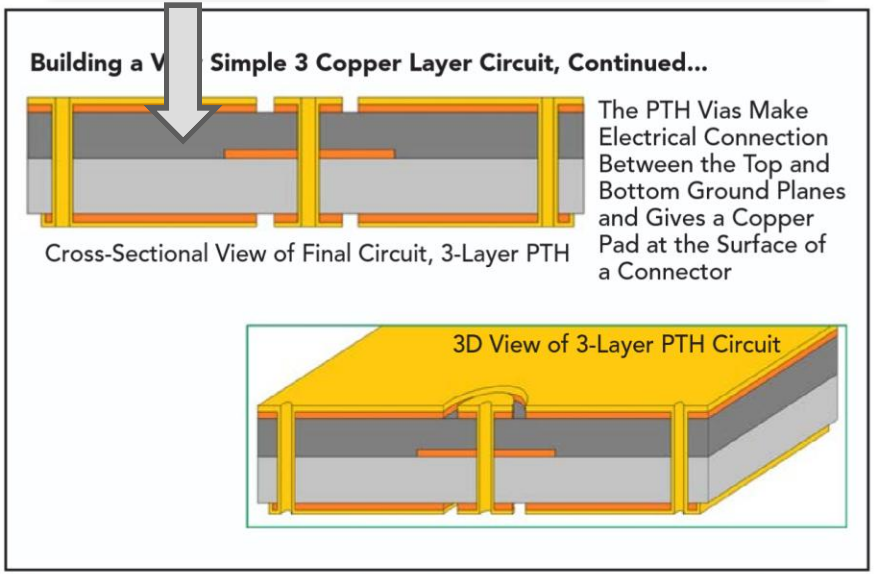


Fig. 8 Final representation of the 3-layer circuit using PTH vias.

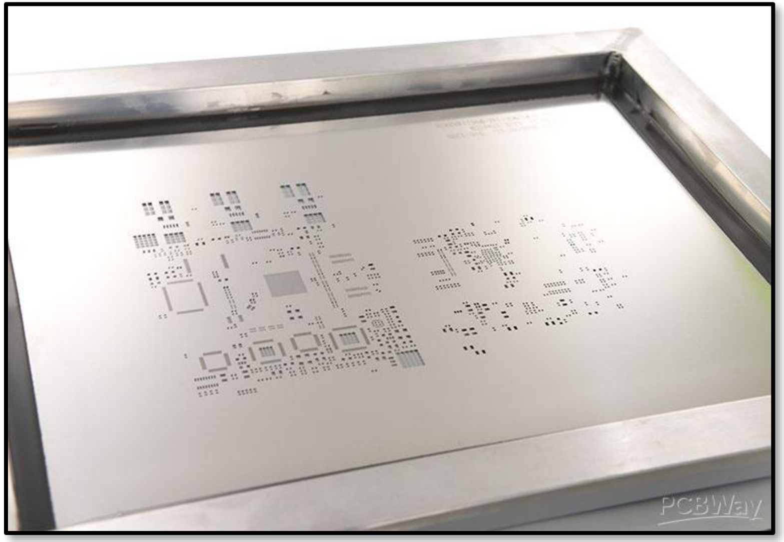
Typically, want prepreg and core materials to be very similar or same to have homogenous dielectric properties

J. Coonrod, "*What RF Engineers Need to Know About PCB Fabrication for Optimal Results.*" Rogers Corp. Dec. 2024.

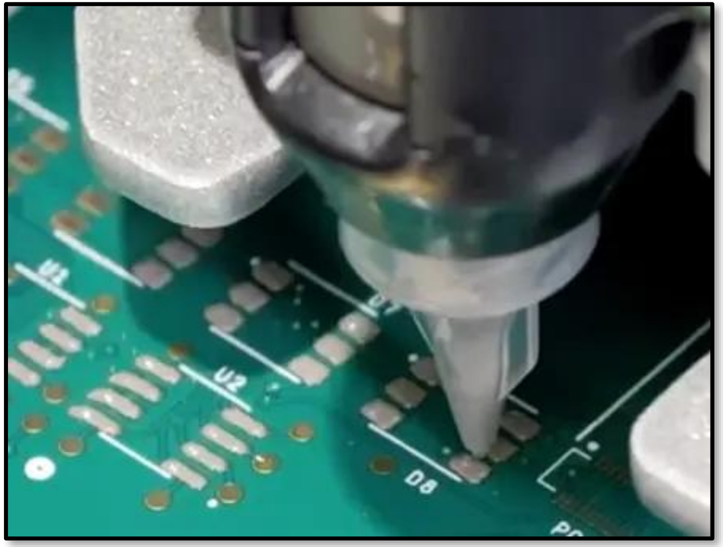
Assembly

Putting Components on PCBs

First we apply solder paste to surface mount pads



Stencil



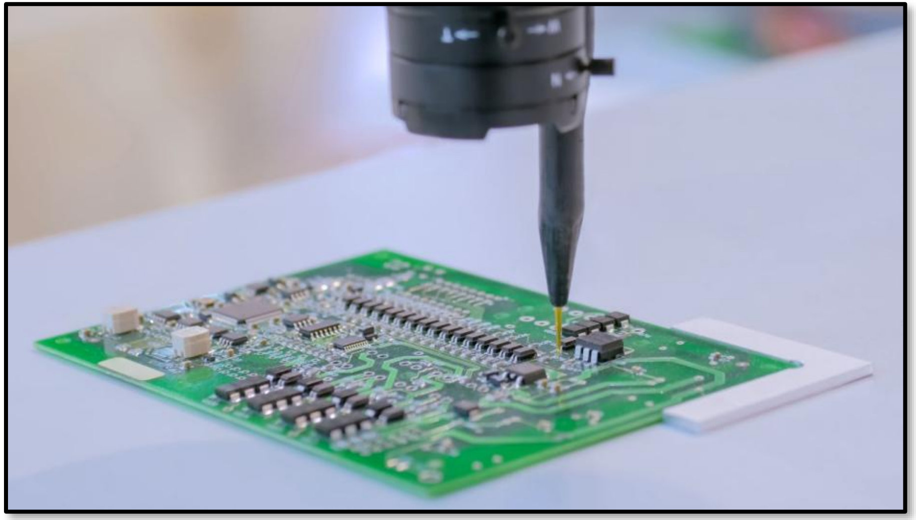
Jet Printing

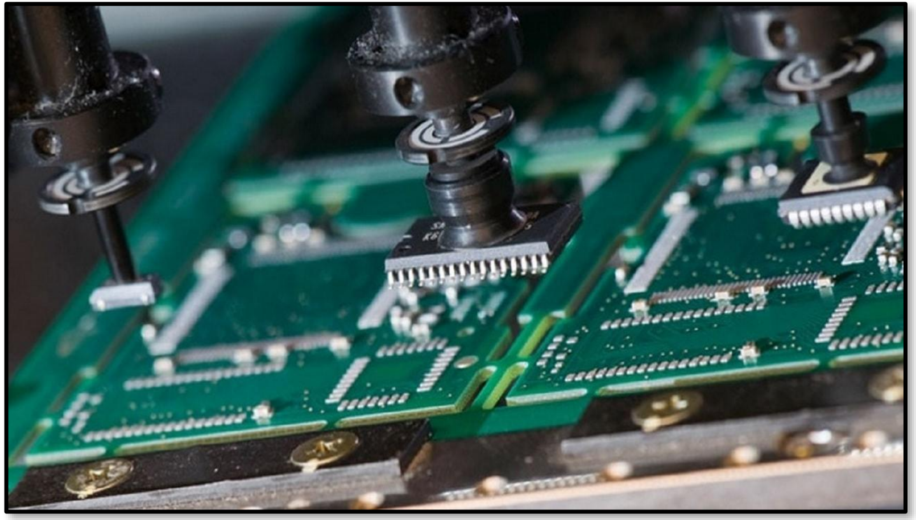
Assembly

Putting Components on PCBs

Next, place surface mount components

Shown are pick-n-place machines → vacuum tip used to hold and move components





Assembly

Putting Components on PCBs

Reflow surface mount parts

Reflow Oven

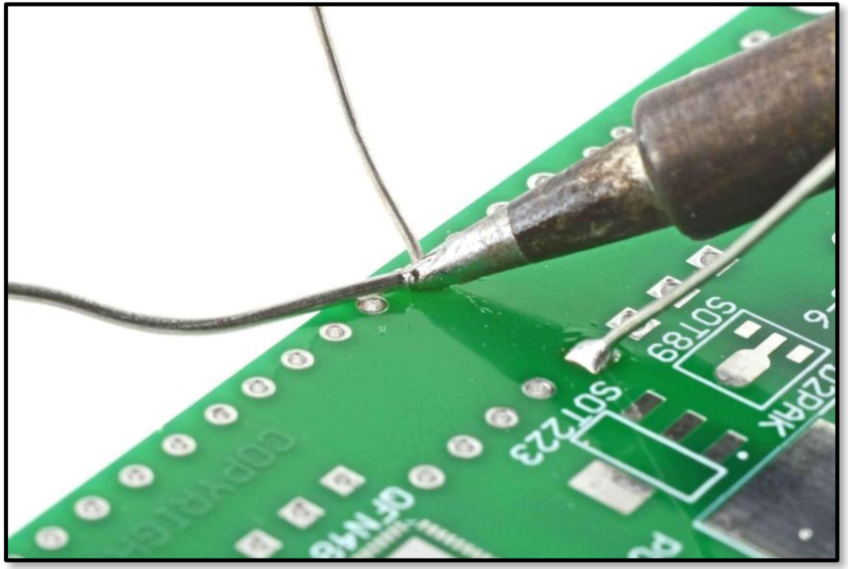
Hot Plate

Hot Air

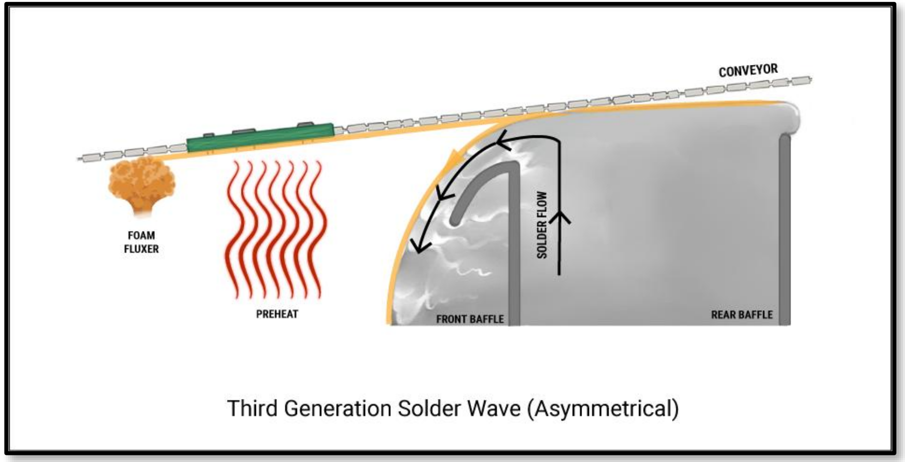
Assembly

Putting Components on PCBs

Solder through-hole components



By hand



Industrial wave soldering

Interesting Fabrications

From a Conference

Biodegradable dielectric substrates?!

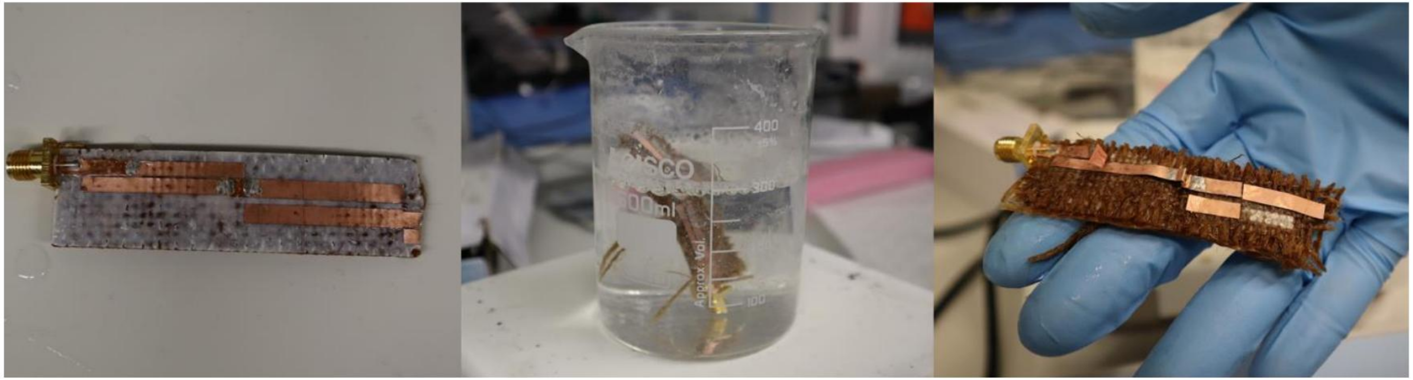


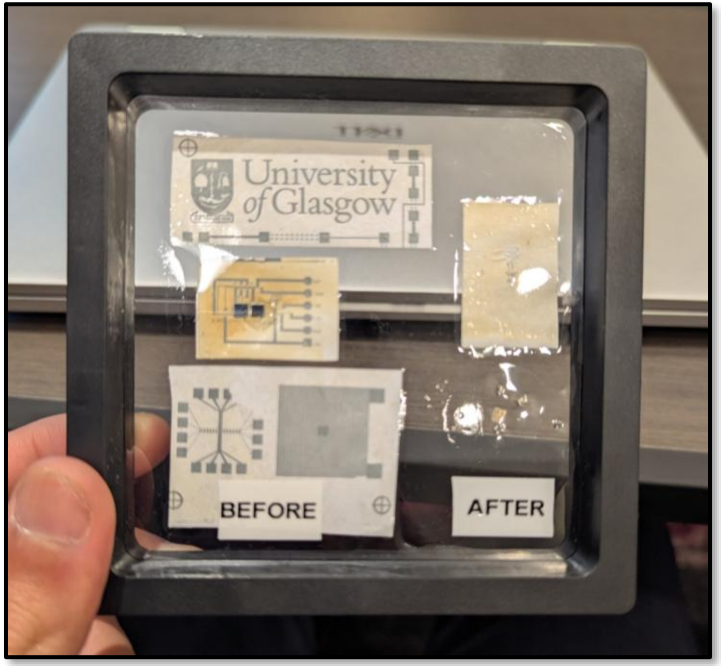
Figure 8. Dissolving the microstrip filter in water and the remaining natural fibres and delaminated copper tracks.

M. Wagih et al. “Biodegradable RF PCBs Towards Sustainable Wireless and IoT” (2025) Documentation. University of Glasgow, DOI: 10.36399/gla.pubs.368649

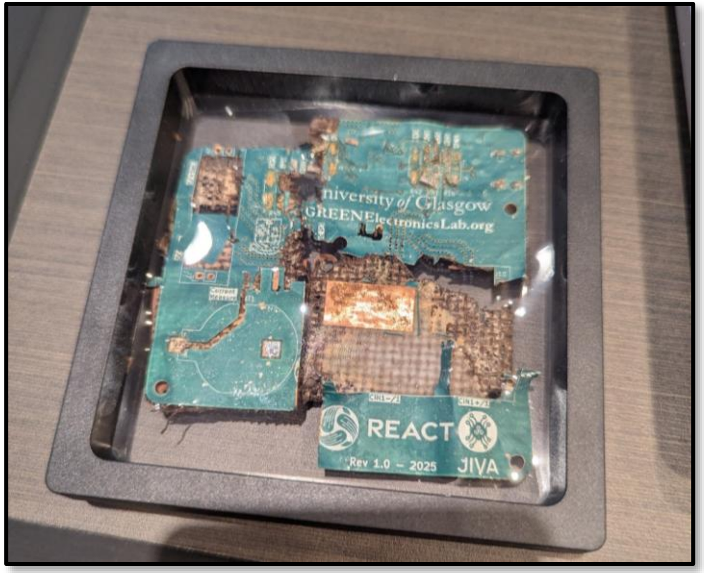
Interesting Fabrications

From a Conference

Dissolvable PCB



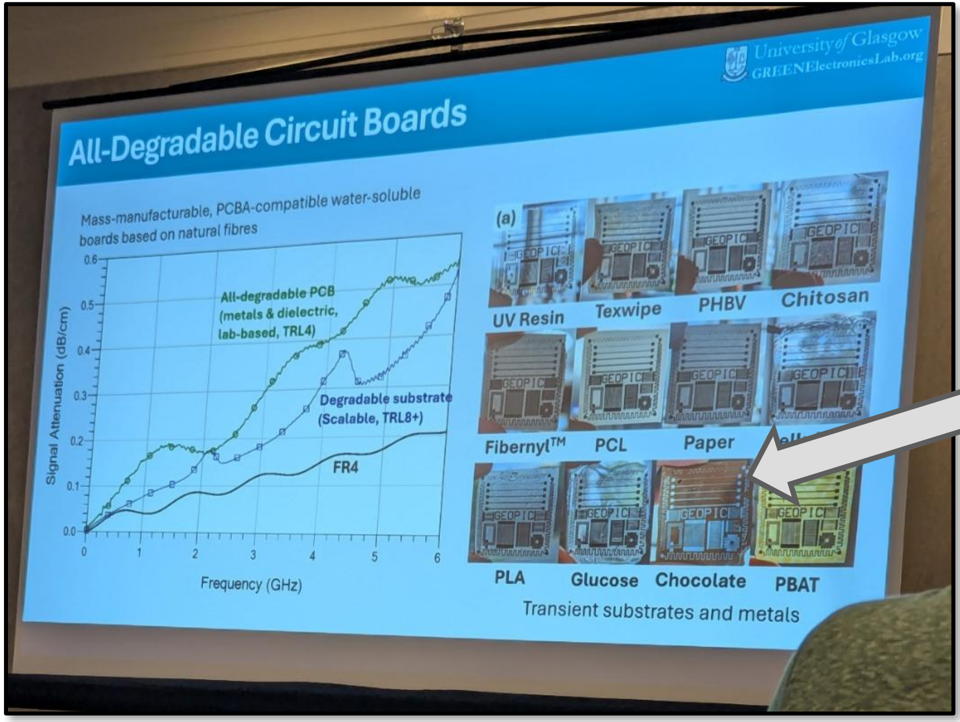
Biodegradable PCB



M. Wagih "*Making Microwaves Green: From Life Cycle Assessments to Chipless Sensing Solutions.*" Distinguished Microwave Lecture. RWW. Jan. 2026

Interesting Fabrications

From a Conference



M. Wagih "*Making Microwaves Green: From Life Cycle Assessments to Chipless Sensing Solutions.*" Distinguished Microwave Lecture. RWW. Jan. 2026

THE ART AND SCIENCE of PCB DESIGN
IAP 2026
@ MIT

Outline

THE ART AND SCIENCE OF PCB DESIGN
IAP 2026

- Introduction
- Tools for Debugging
- Debugging Strategies
- Causes Behind Bugs
- Fixing Issues
- Summary

System Design

Course Project

Identify Problem ✓

Devise Solution ✓

Identify System Parameters ✓

Create/Model System ✓

Design PCB! → Fabrication → Test/Debug

Super Important!

FM radio receiver

Course Project!

THE ART AND SCIENCE of PCB DESIGN

IAP 2026

@ MIT

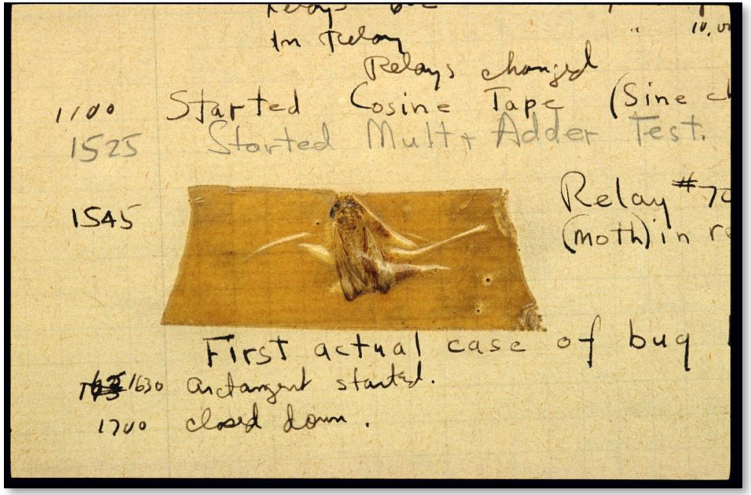
Introduction

What exactly is debugging?

The term was popularized by Admiral Grace Hopper when she pulled a moth out of the Harvard Mark II in the 1940s

Didn't we already spend so much time being meticulous with our design?

- **We're never perfectly sure what's going on**
- **Models are never a fully accurate representation of real-world operation**
- **Tolerances and parasitics can be different in reality than what the datasheet says, or have a greater impact than we expect**
- **We probably didn't model everything**



Introduction

A Bit on Debugging

Debugging is hard...but it's super rewarding!

Through our design process, we have learned a lot about how our circuits are supposed to function, and now we get to apply that knowledge

It's not as scary as it looks

- **Many common errors that are easily fixable**
- **Many tools to help us**

THE ART AND SCIENCE OF PCB DESIGN
IAP 2026

@ MIT

Tools for Debugging

Multimeters

Our Best Friend

Multimeters are easy to use and have a ton of functionality – they should be your first stop

Voltage

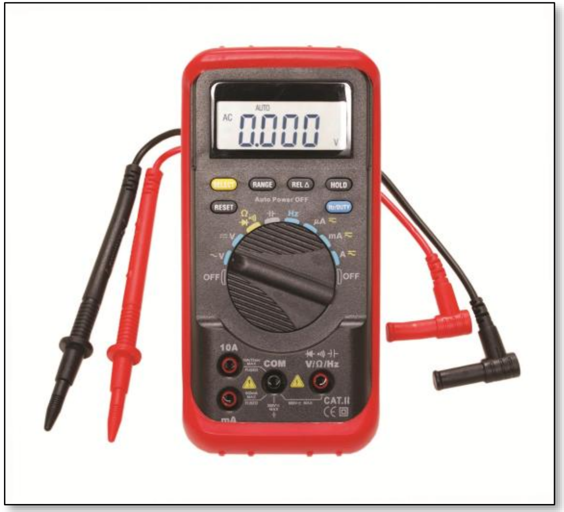
- **Potential difference between the two probes**

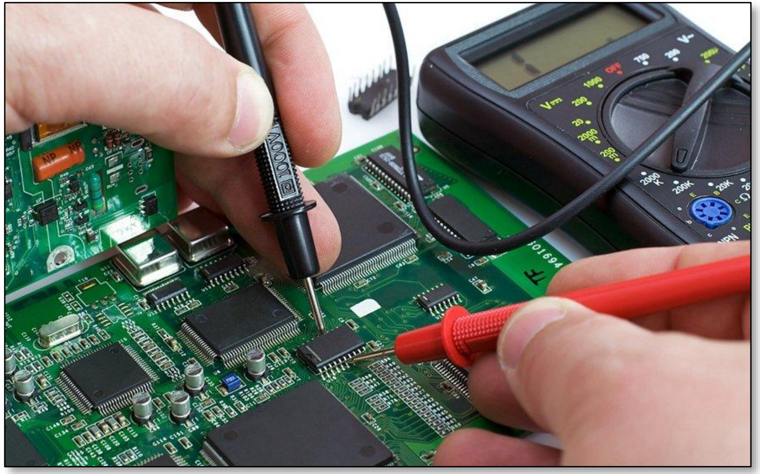
Resistance/Diode/Continuity

- **Useful to check connectivity of traces and make sure we don't have shorts**
- **Make sure that there is nothing powering the circuit when doing this**

Current

- **Not super useful for debugging but important in general**
- **Use cautiously for low currents only**





THE ART AND SCIENCE OF PCB DESIGN
IAP 2026

Oscilloscopes

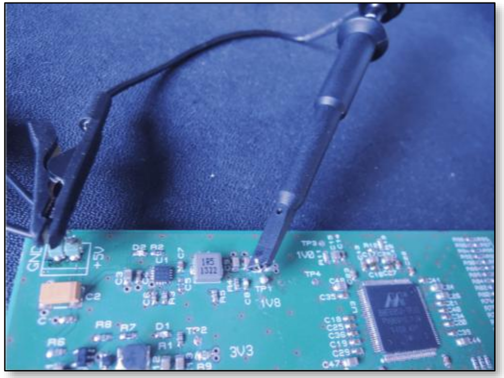
Our Other Best Friend

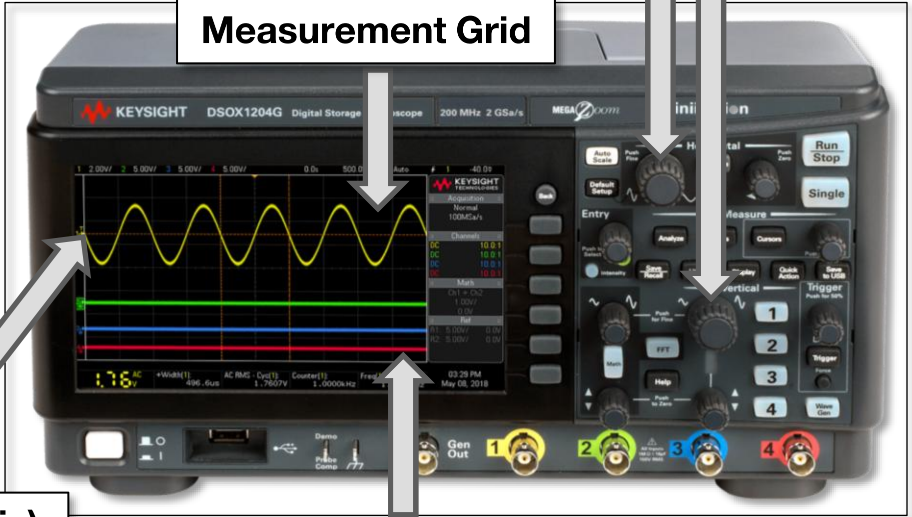
Oscilloscopes help to actually "see" our signals over time whereas multimeters only give one instantaneous value

Thus, we can use scopes for signals that aren't just constant DC values

Scopes show multiple signals at the same time

- **Input vs. output of an IC to make sure it's doing what we expect**
- **Before and after a bypass capacitor to see how well it's filtering our power rails**





THE ART AND SCIENCE OF PCB DESIGN

IAP 2026

@ MIT

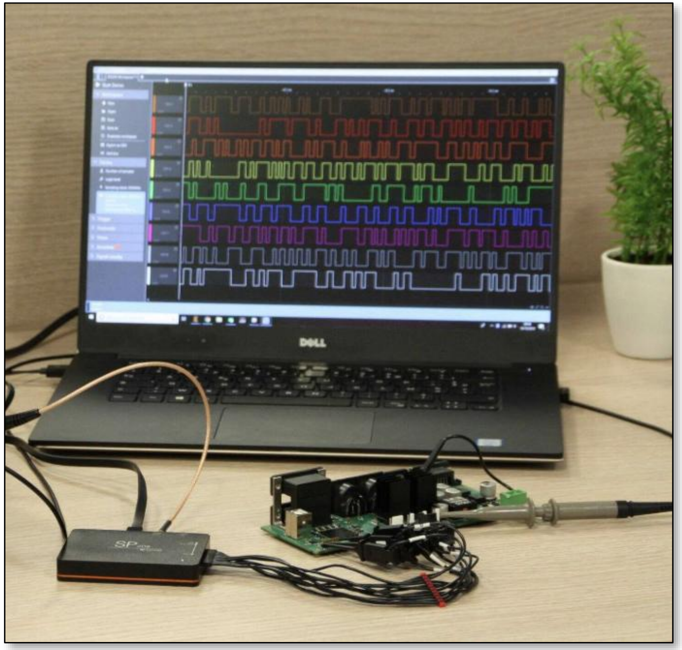
Logic Analyzers

Characterizing Digital Signals

Logic analyzers are basically digital oscilloscopes – they can display several digital signals at the same time

Really good for debugging communication systems and digital timing errors

Shows the values of bits being sent over digital lines



Power Supplies

Injection!

Power supplies allow us to inject a known voltage or current into points on our PCB

Allows us to test the various parts of our circuits without having to power on the entire board and risk damaging any components

We can set current limits and voltage limits to control the amount of power we inject – we won't destroy our board if there is a short

Ex: testing a power regulator

- **To ensure that the USB power is correctly getting stepped down to 3.3V, we can inject 5V into the USB power trace and probe the output of the buck converter**

Debugging Strategies

Continuity Checks

Identifying shorts

1. Ensure that nothing is supplying power to your board (everything is off)
2. Turn the multimeter to continuity mode
    - Represented by propagation waves
    - This will often be combined with the resistance or diode setting
3. Probe across the relevant traces
    - If the multimeter makes a beeping sound or shows a low resistance on the screen, then you have a short
    - If the multimeter is quiet or shows OL or a really high resistance, then your traces are separated

THE ART AND SCIENCE OF PCB DESIGN
IAP 2026

Check Voltages

Probing is super helpful

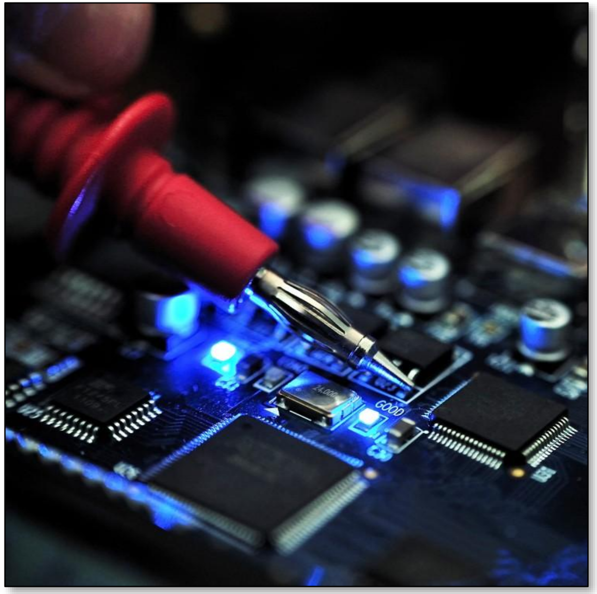
Ensure that all of the power traces are at the correct DC voltages

Ensure that all ICs are getting the correct input power and input signals

Ensure that the output of the ICs are what is expected given their input

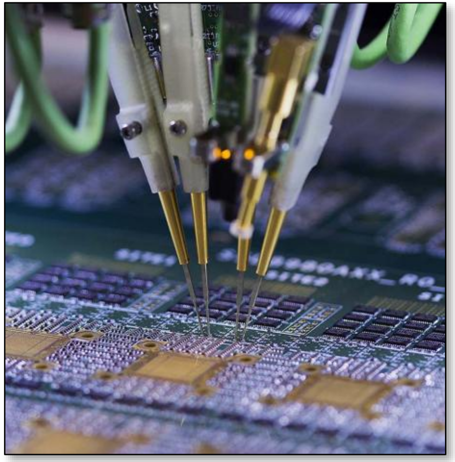
- **A lot of this can be done with a multimeter, but for more complex signals, don't be scared to use an oscilloscope or logic analyzer**

PROBE EVERYTHING!!



Industrial Debugging Techniques

Because they're cool



Flying Probe Machines

- **Consists of several test probes that automatically probe test points on a PCB**
- **Used for earlier-stage prototyping and low-volume production**

In-Circuit Testers

- **Consists of a custom bed-of-nails test fixture that attaches to a PCB's test points**
- **Performs various tests in a given sequence including continuity/shorts, passives measurements, etc.**
- **Used for later-stage testing and high-volume production**

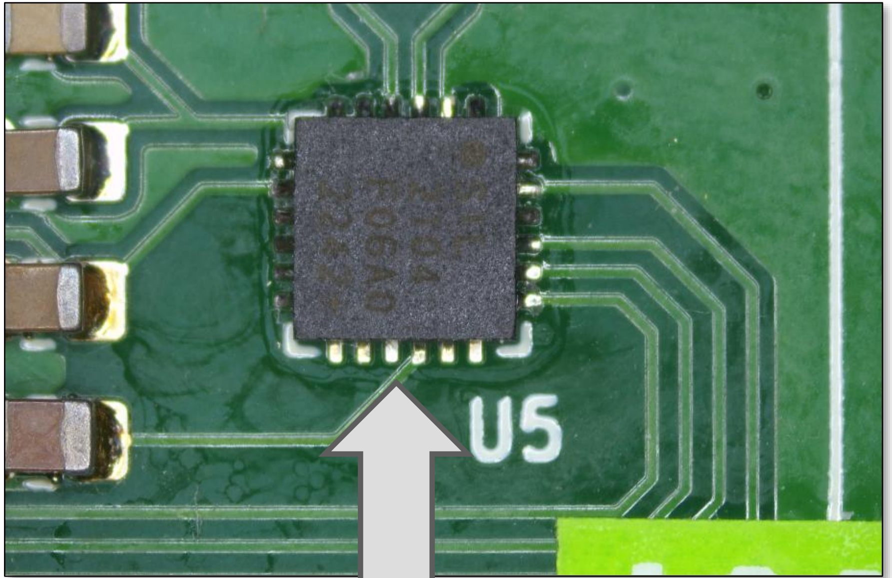
Causes Behind Bugs

Too Little Solder

THE ART AND SCIENCE OF PCB DESIGN

IAP 2026

The bottom row of pins on U5 is not properly soldered, indicated by the gold color of the pads as opposed to the silver color of reflowed solder



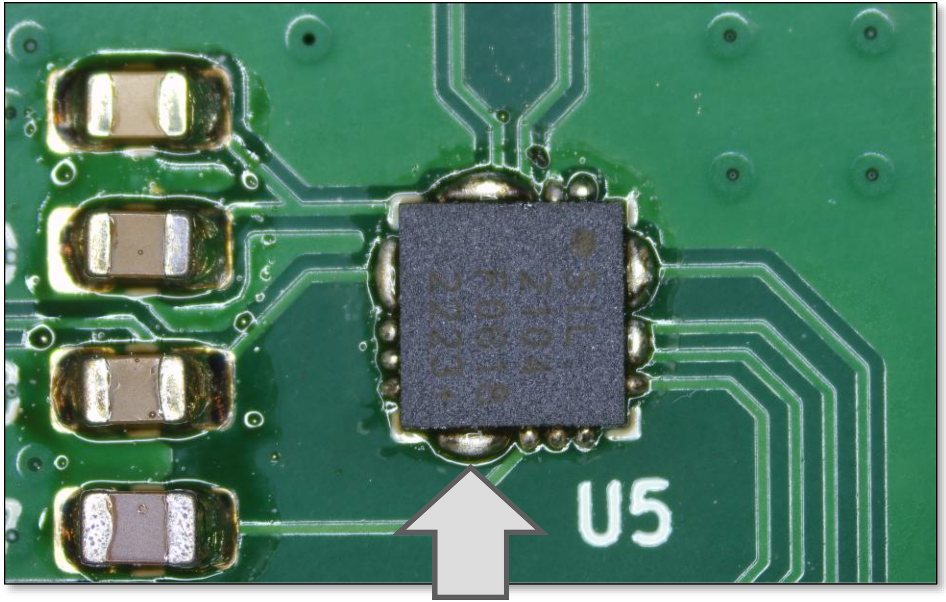
Gold-colored ENIG pads still visible (no enough solder)

@ MIT

Too Much Solder

THE ART AND SCIENCE OF PCB DESIGN
IAP 2026

Excess solder leads to bridges between pads that should be isolated, leading to shorts



Short between multiple pins due to excess solder

@ MIT

Incorrect Components

Sometimes there are sourcing issues or small design errors that lead to a board malfunctioning

Ex: Boost converter circuit

- **Output voltage is determined by the ratio of R1 and R2**
- **In this case, given an R3 of 300k, R4 should be 10.6k to get an output of 36V**
- **If a well-meaning designer or assembler rounds this down to 10k, the output voltage becomes 38V – if other components were only sized for 36V, they could be damaged**

Datasheet Snippet

9.2.2.1 Program Output Voltage

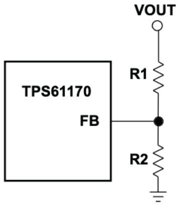


Figure 13. Program Output Voltage

To program the output voltage, select the values of R1 and R2 (see Figure 13) according to Equation 2.

$$Vout = 1.229\ V \times \left(\frac{R1}{R2} + 1\right) \qquad R1 = R2 \times \left(\frac{Vout}{1.229\ V} - 1\right)$$

THE ART AND SCIENCE of PCB DESIGN

IAP 2026

Incorrectly Placed Parts

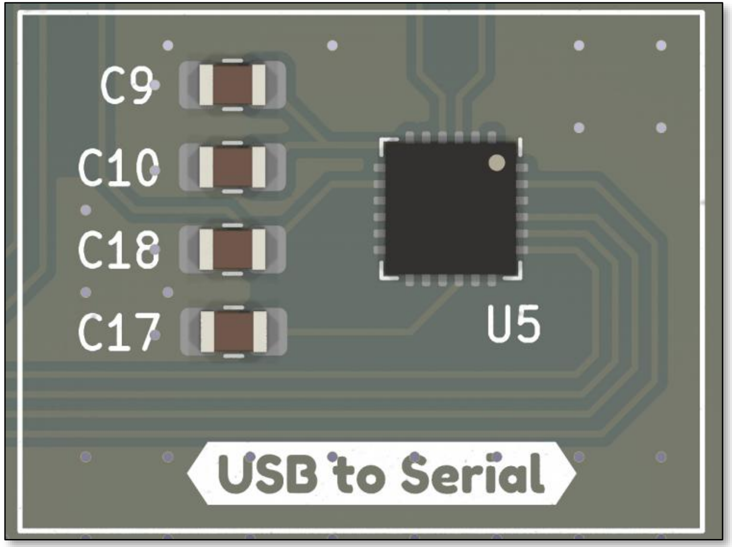
THE ART AND SCIENCE OF PCB DESIGN
IAP 2026

Easy to mess up orientation of non-passive, symmetric components (such as U5 on the right)

- **Square-shaped ICs**
- **Diodes**

Alignment dots are wonderful, but sometimes they're really small on the IC or obscured by the silkscreen on the PCB

We can always double check the orientation of our components by looking at our schematic and layout files and the component's datasheet, and then matching traces and pins



Signal Timings

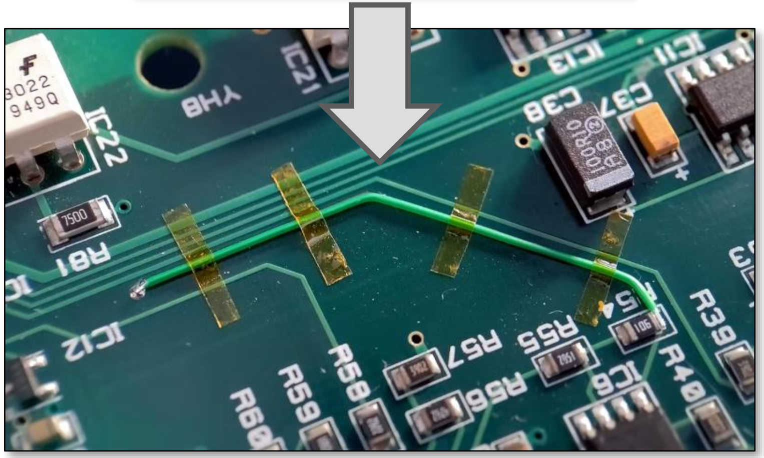
More complex systems with high-speed communication are prone to timing errors

This is oftentimes a result of traces that are improperly sized (either too long or too short) that are attached to RC networks

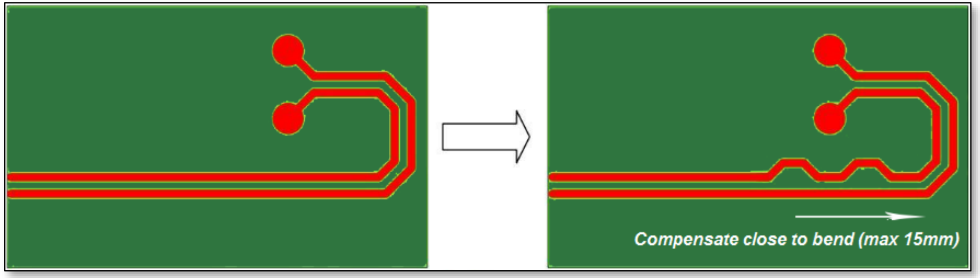
- **If the time constant is too high relative to a signal's travel time, an unwanted delay in data transfer occurs**
- **If the time constant is too low relative to a signal's travel time, needed delay times might not be met**

Differential pair mismatches can also yield timing errors in data transfer, such as with USB

Splice trace and add an external wire to lengthen



https://hackaday.com/2024/07/14/five-ways-to-repair-broken-pcb-traces/



THE ART AND SCIENCE of PCB DESIGN

IAP 2026

@ MIT

Fixing Issues

The Easier Problems

THE ART AND SCIENCE of PCB DESIGN

IAP 2026

Incorrect passive values

- The footprints for passives are generic so it is fairly simple to desolder and resolder a new component with the right value
- Can stack components or solder adjacent components to mimic parallel and series connections for intermediate values

Incorrect IC orientations

- A bit more annoying to fix but can still just be desoldered and resoldered on correctly

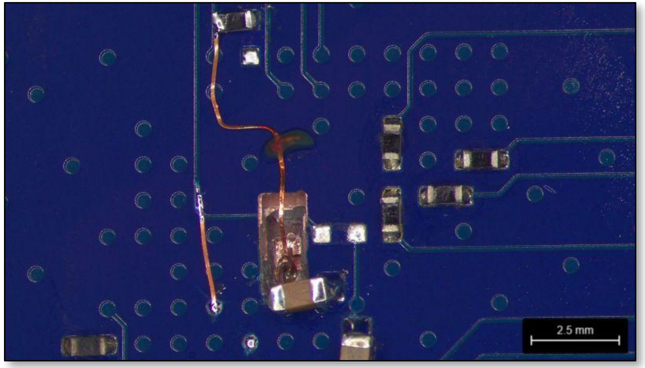
Individual solder joints

- Can usually be fixed with a hand iron, solder wick, flux, and solder

The Harder Problems

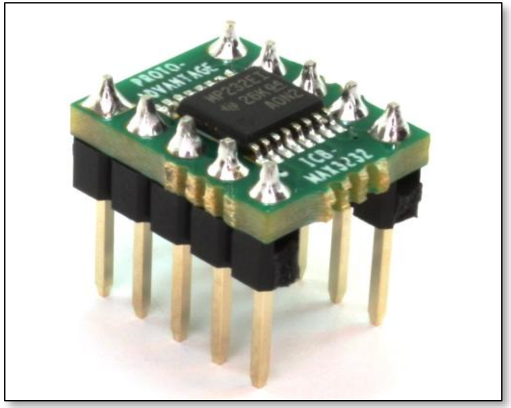
Incorrect Net Connections

- Traces on outer layers can be cut and reconnected using thin magnet wire
- Component connections can be rerouted by soldering wires to the excess space on their pads
- Test points can be utilized for connections



Incorrect Footprints

- Incorrect passive pad sizes can be solved with careful soldering
- If the footprint is missing pins, the remaining pins can be connected using thin magnet wire
- Footprint pad size can be increased by scraping off surrounding soldermask
- IC breakout boards can be used to externally connect to an incorrect footprint via jumper wires



THE ART AND SCIENCE OF PCB DESIGN
IAP 2026

Summary

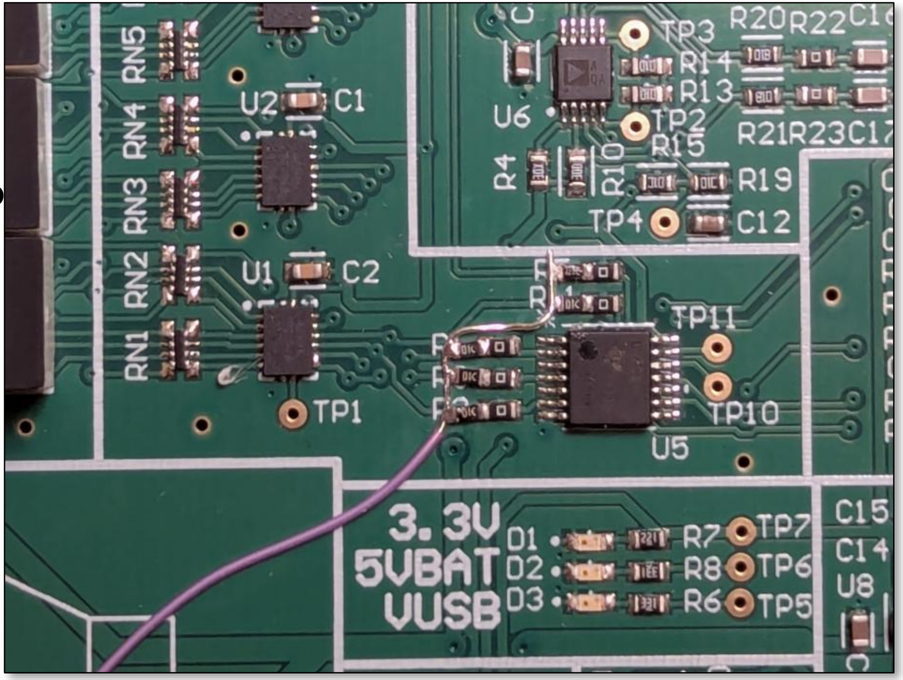
Debugging can be fun!

Debugging can definitely be tedious and frustrating, but it's a really fruitful process!

There are many tools to aid along the way, and to help really understand a board

Some useful resources on debugging:

- Cadence's Take on Debugging Techniques
- Board-Level Repair Guide
- A Really Good Guide to Debugging from a Former 6.204/6.101 TA



Questions

Announcements

- [ ] **Complete Lecture Quiz 10**
- [ ] **Make-up missed lab check-offs during office hours**
- [ ] **Course evaluations (closes Friday)!**