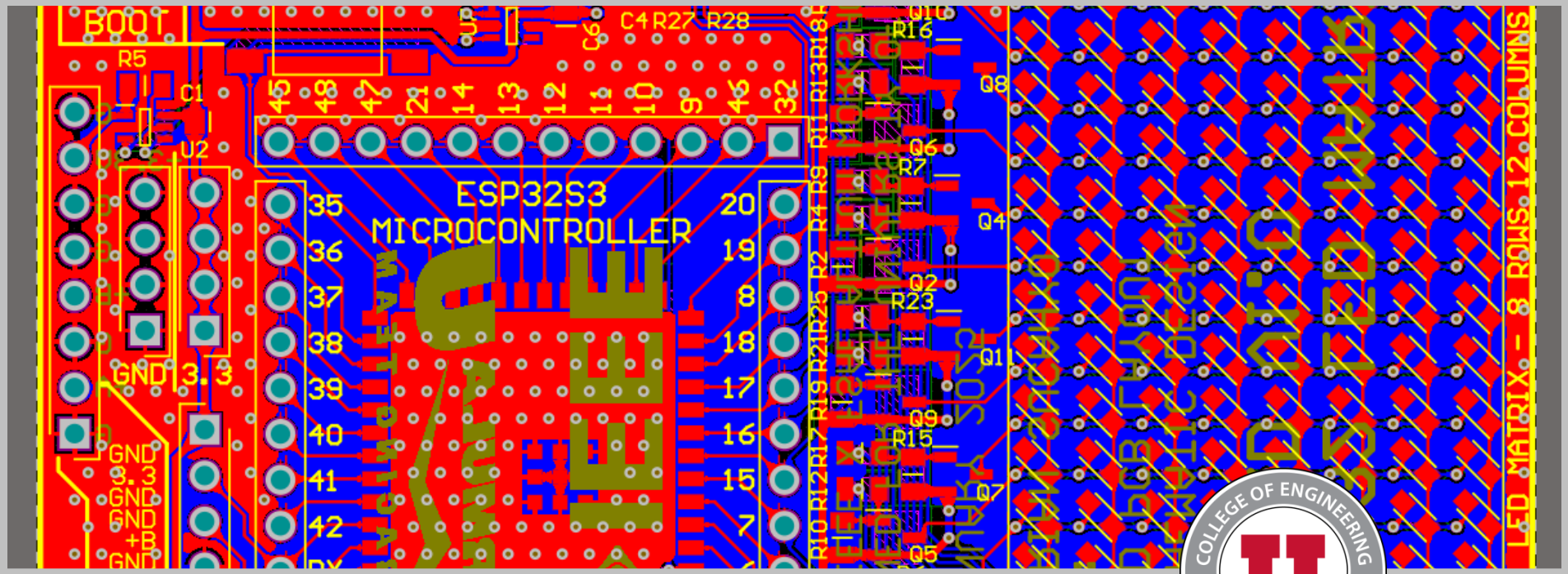
IEEE X FSAE PCB Design Workshop: (Week 04)

Intermediate Printed Circuit Board Design (LED Matrix)



Hosted By: Adrian Sucahyo and Nick Howard

COLLEGE OF ENGINEERING
THE UNIVERSITY OF UTAH

Announcements

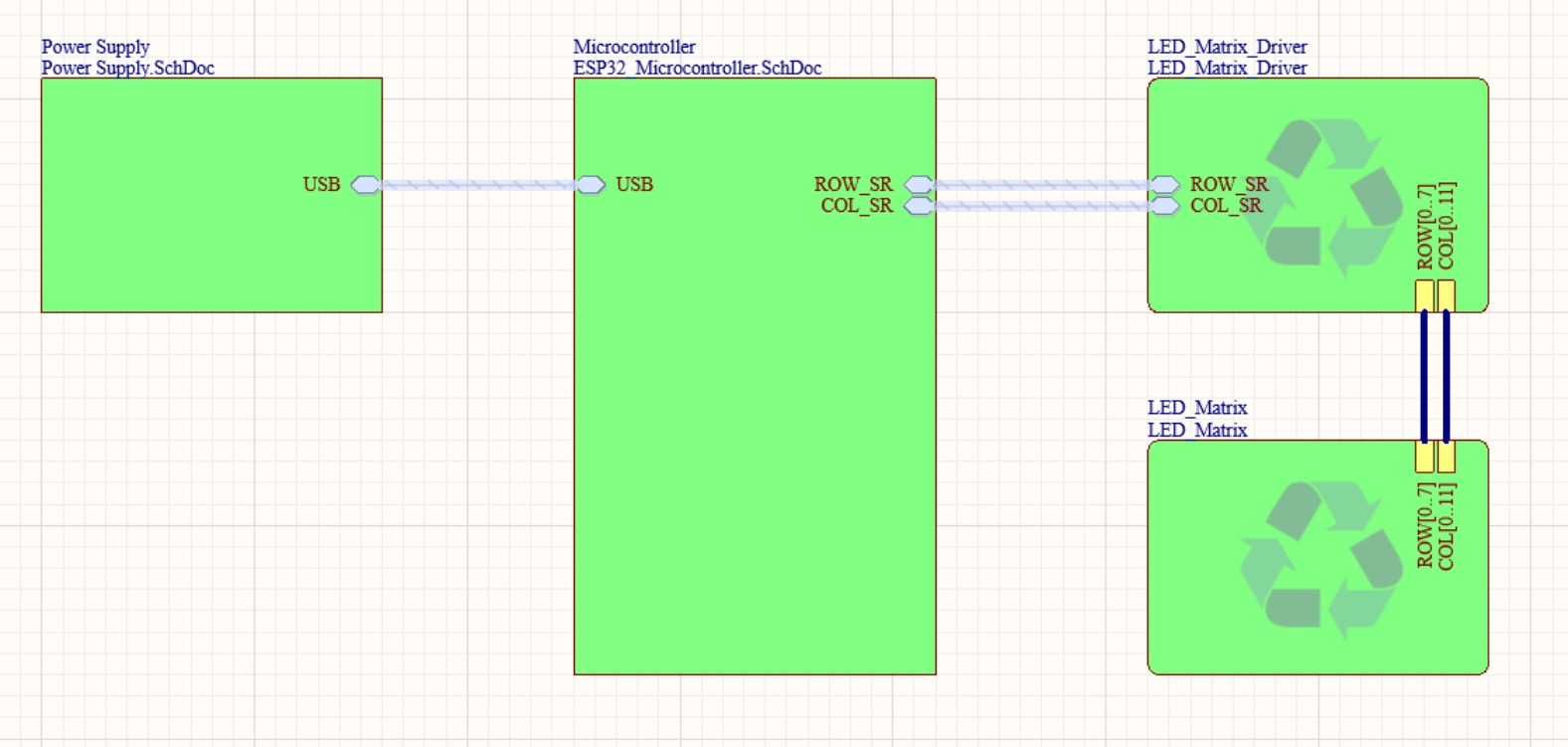
PCB Design Submissions

- Soft Deadline: March 1st
- Hard Deadline: March 4th

Next Week

- Food, Work Time, and Design Reviews
- Any other topics of interest?
  - Will send out a survey

Project Review



- Hierarchal Design of the LED Matrix
- Introduction to ESP32 Microcontrollers

Microcontroller Review

- LED Matrix based on ESP32S3
  - Simple to design around
  - Beginner friendly development env.
    - (Arduino or esp-idf)
  - More advanced that ATMEGA328p
  - Many additional features
    - WiFi
    - Bluetooth
    - USB OTG

ESP32S3 Module

- We don’t want to focus on designing an ESP32S3 microcontroller circuit from scratch.
  - Many extra components
  - Sensitive data lines
  - Someone has already designed if for us
- Use the ESP32S3 Module!

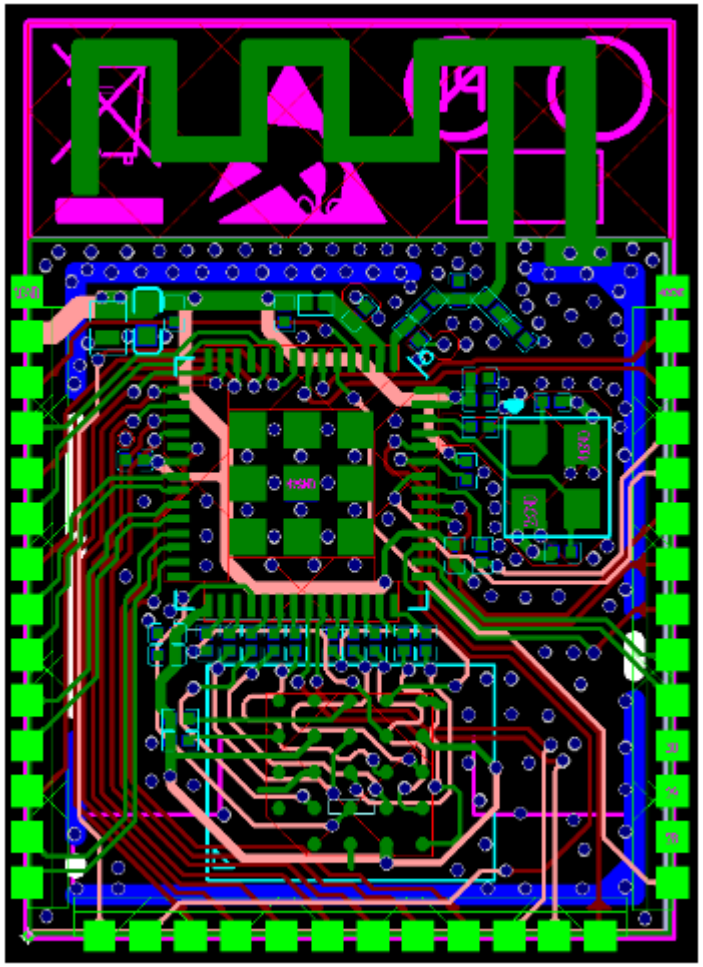
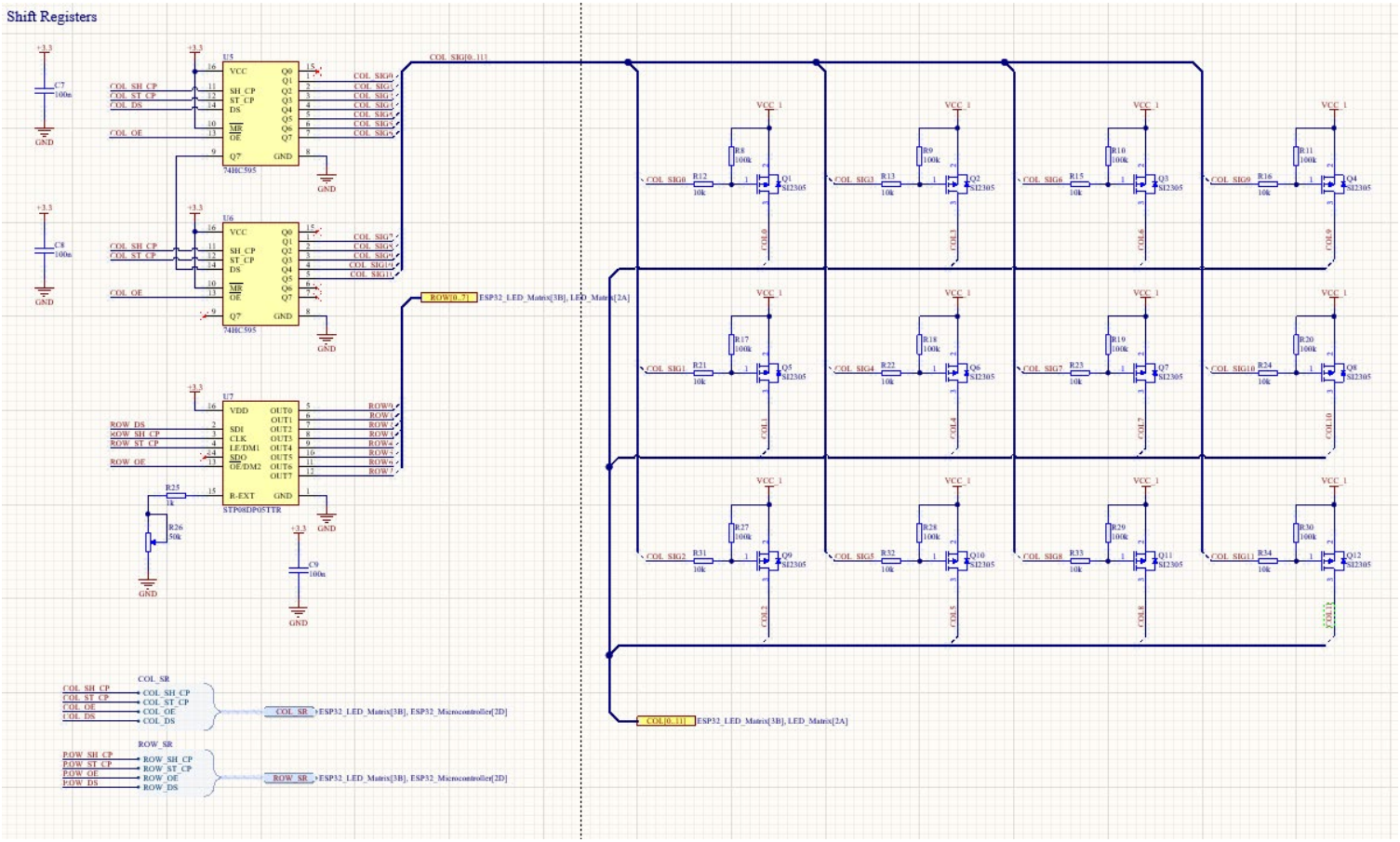


Fig. 1: ESP32-S3 Reference PCB Layout

LED Matrix Driver Review



- Driver provides the current to power all LEDs
- Uses shift registers to minimize input on GPIO

Layout and Placement

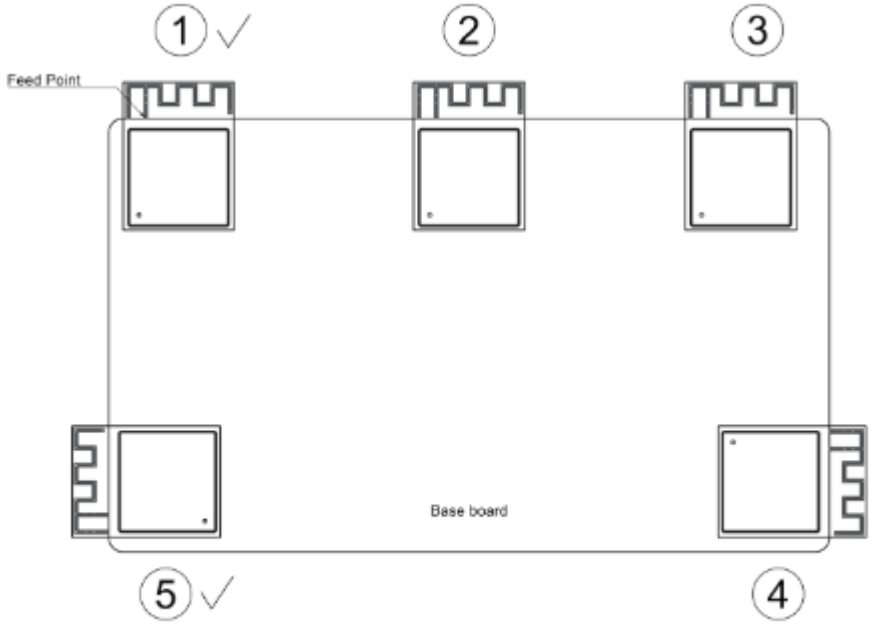


Fig. 3: Placement of ESP32-S3 Modules on Base Board (antenna feed point on the left)

- The ESP32S3 has an antenna that needs to be placed correctly to function.
- The antenna portion of the module hanging.

Layout and Placement

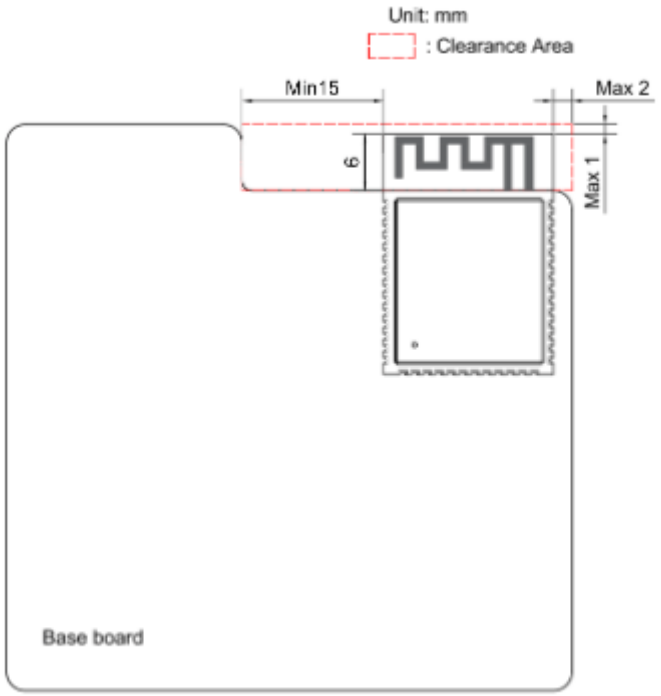
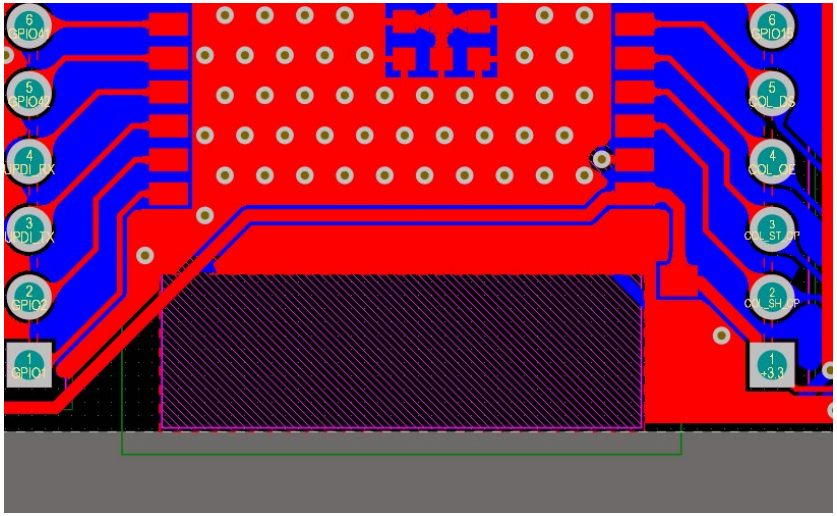
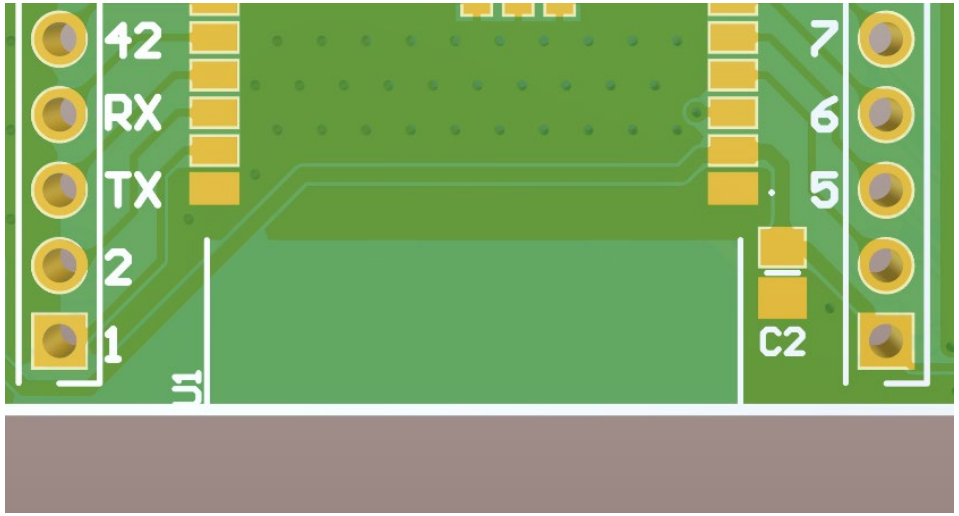


Fig. 4: Keepout Zone for ESP32-S3 Module's Antenna on the Base Board

- A board cutout is also appropriate for the antenna.
- Note: We will not be using this method (antenna is exposed and could be broken off)

Keepout Regions



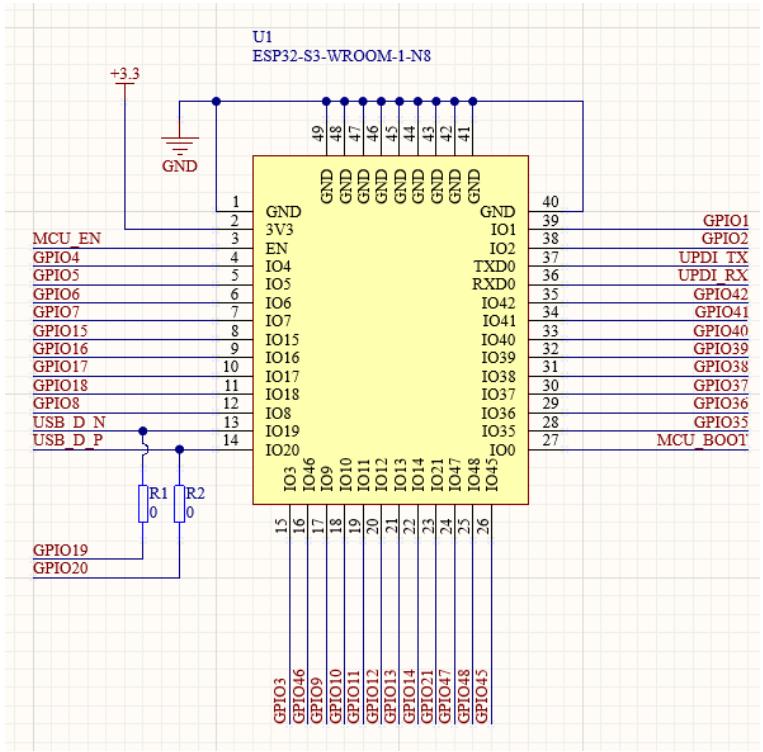


- The antenna portion of the module may be placed completely on the board, however there must be no GND plane below.
- Use Keepout regions to facilitate this.

Important Pins / GPIO

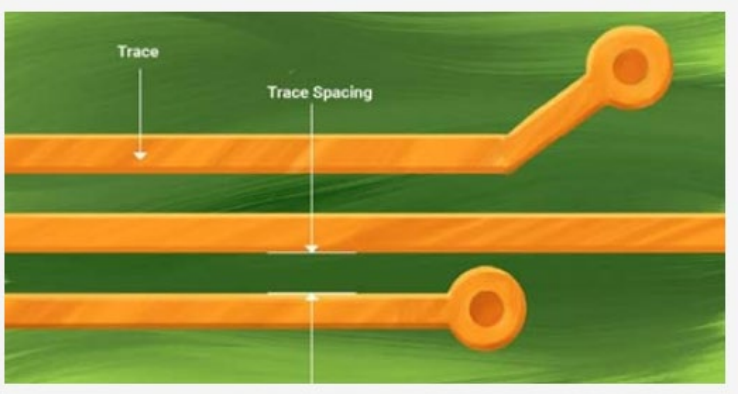
- PIN 3
  - MCU Enable
- PIN 13 and 14
  - USB- and USB+ (respectively)
- PIN17
  - Boot Mode Bootstrapping Pin
- PIN 36 and 37
  - UART RX and TX (respectively)

Note: These refer to physical pins, not GPIO Number



PCB Manufacturing – DFM and DRC

- DRC (Design Rule Check) Analysis is for checking that the minimum requirements can be met.
- DFM (Design for Manufacturing or Design for Manufacturability) Analysis is a process undertaken to ensure that the PCB layout has minimal problems.
  - Attempts to minimize problems to be encountered during manufacturing and / or assembly.
- Some aspects that might be checked are:
  - Drill Hole Size
  - Drill to Copper Clearance
  - Trace Spacing
  - Solder Mask Clearance



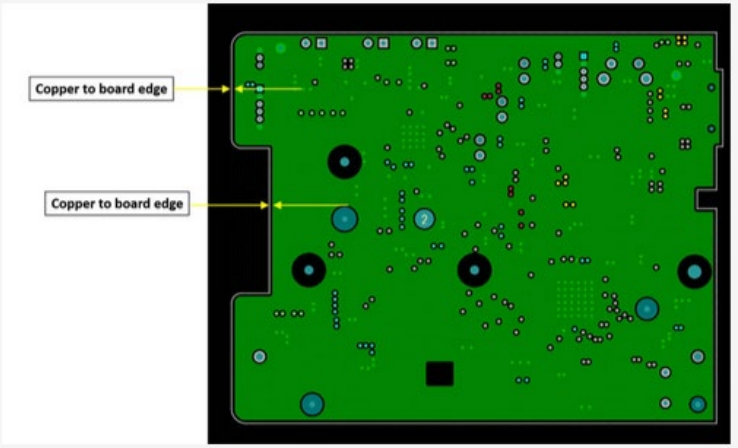


Image credit: Sierra Circuits
https://www.protoexpress.com/kb/pcb-manufacturing-overview/

DFM vs DRC

Why do I need to use both?

- DRC's goal is to detect any discrepancies or errors.
  - Any errors found in the DRC verification will be present in all manufactured boards.
- DFM's goal is to identify any aspects that could lead to issues later on during the manufacturing process.
  - Errors found in the DFM may appear in some boards during the manufactured lifetime.
  - This could manifest as variability in the manufacturing process.
  - Accounting for DFM can reduce cost and complexity.
  - Key for later production stages

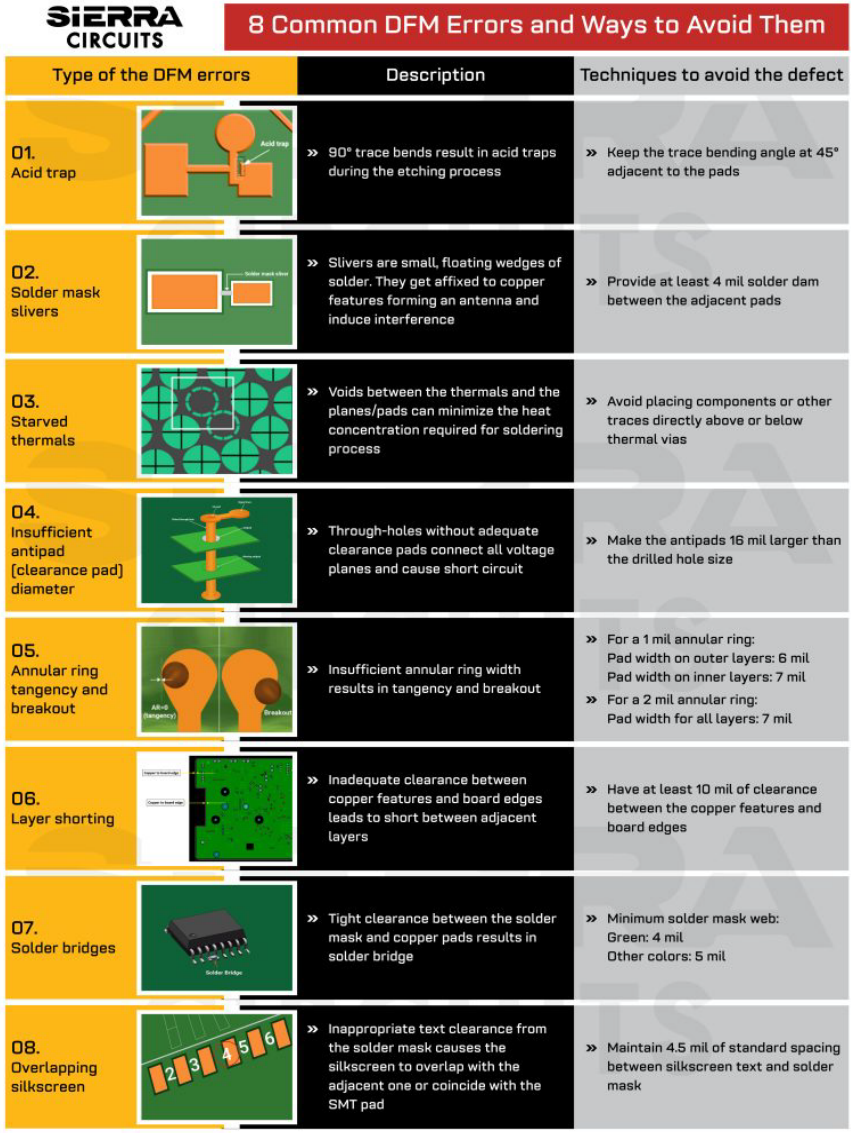


Image credit: Sierra Circuits
https://www.protoexpress.com/blog/dfm-issues-pcb-manufacturing/

Gerber Files

- Gerbers (or Gerber Files) are the files sent to the manufacturer that will be used to make the boards
- Engineers at the fab review files for manufacturability and then send it off to production.
- Gerber files can be generated from EDA software but needs to be reviewed by us before sending it off.

- There are other versions of PCB design files such as:
  - Gerber X2
  - OBB++

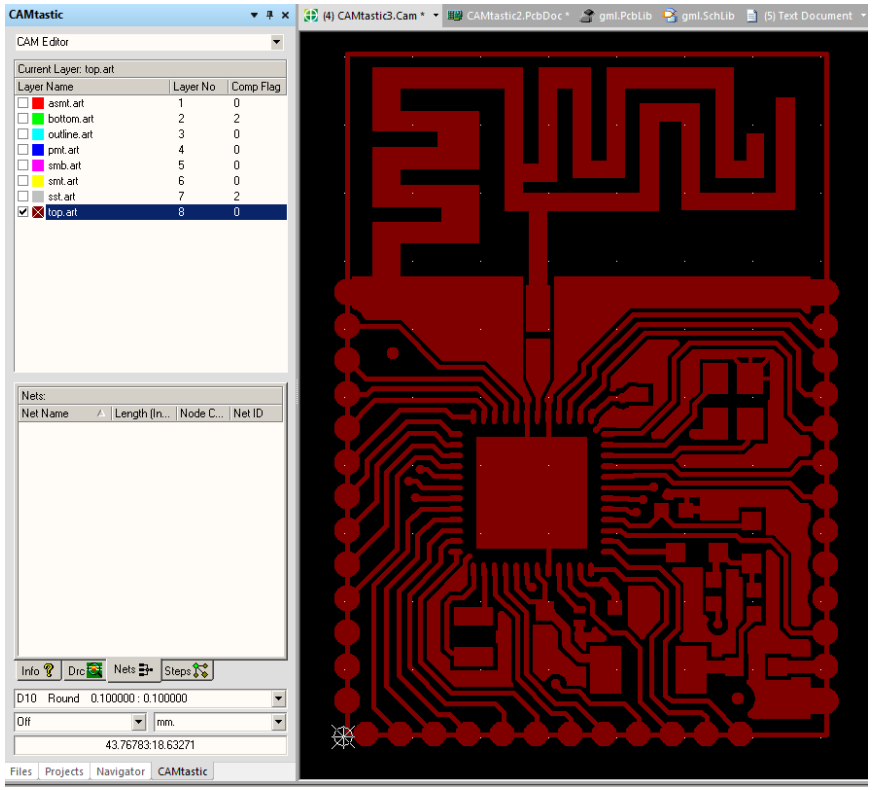
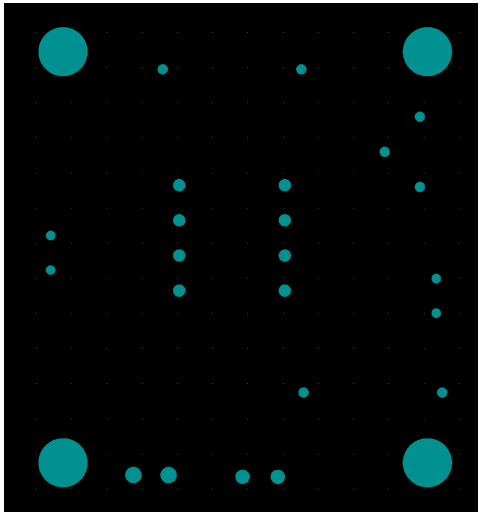


Image credit: One Thesis
https://www.onethesis.com/gerbers-to-footprint/

NC Drill Files

- NC Drill Files contain the information for the drill holes of the PCB.
  - These are generated separately from the Gerber Files but are just as necessary for the manufacturing process.



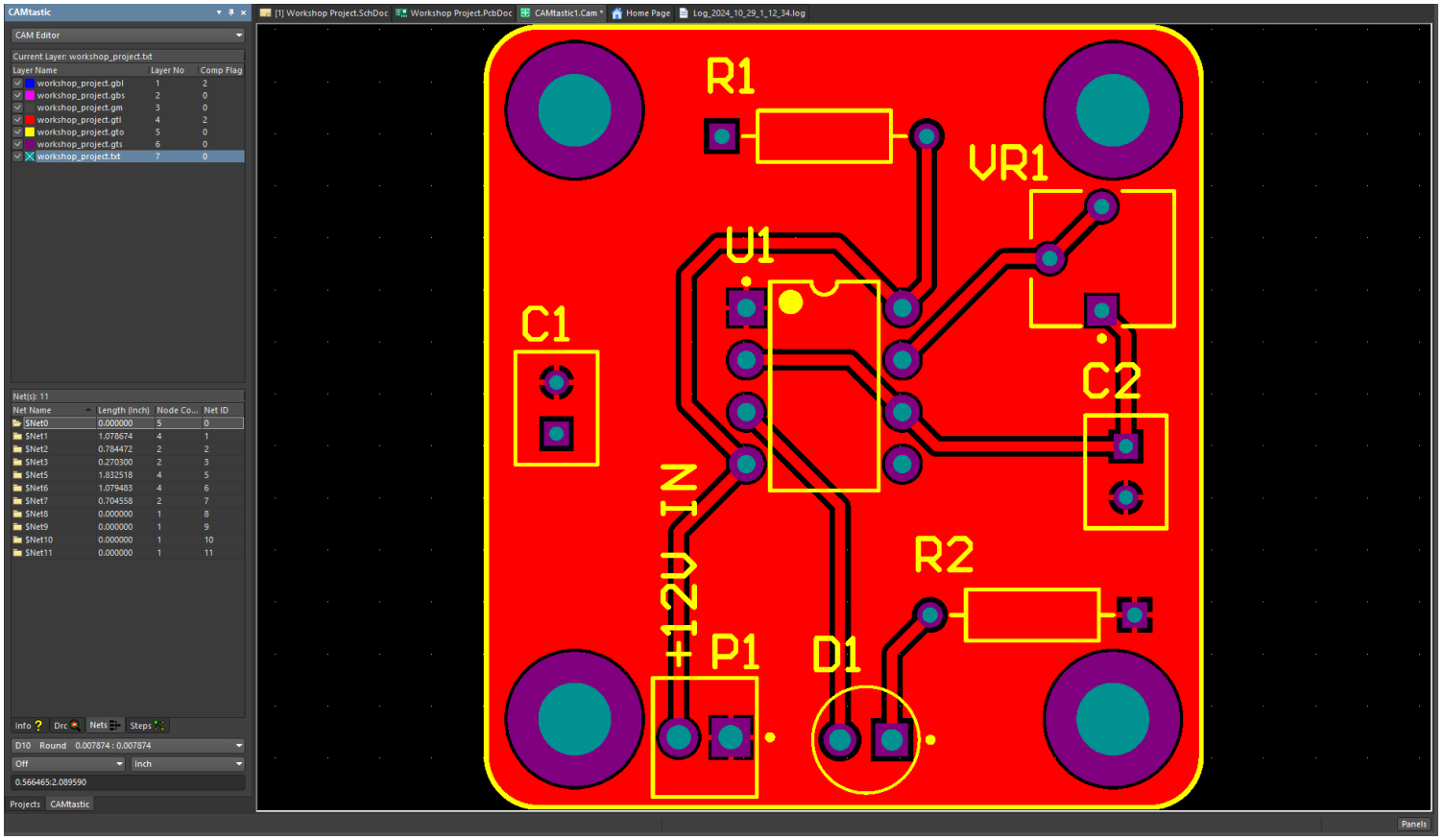
```
NCDrill File Report For: Workshop Project.PcbDoc   10/29/2024  1:09:38 AM

Layer Pair : Top Layer to Bottom Layer
ASCII RoundHoles File : Workshop Project.TXT

Tool       Hole Size              Hole Tolerance              Hole Type       Hole Count   Plated        Tool Travel
T1      28mil (0.7mm)                                          Round           4          PTH        1.29inch (32.82mm)
T2      30mil (0.75mm)                                         Round           7          PTH        2.02inch (51.35mm)
T3      35mil (0.9mm)                                          Round           8          PTH        0.90inch (22.86mm)
T4      41mil (1.05mm)                                         Round           2          PTH        0.10inch (2.54mm)
T5      47mil (1.19mm)                                         Round           2          PTH        0.10inch (2.54mm)
T6      140mil (3.556mm)                                       Round           4          PTH        3.24inch (82.30mm)
Totals                                                                         27

Total Processing Time (hh:mm:ss) : 00:00:00
```

Altium CAM Viewer / CAMTastic



- Altium has a built-in CAM viewer and editor called CAMTastic which can be used to verify design files.
- CAMTastic isn't very intuitive to use but can be useful to verify designs files within Altium.

Questions?

Questions?

Other Topics for Today

- Design Reuse Schematics
- Microcontroller Layout Guidelines (ESP32)
- Rooms / Grouping
- Exporting Gerber Design Files
- Work Time and Design Review / Discussion

- Free Work Time and Design Reviews

Download Today's Project Files

Navigate to the workshop GitHub and download today's files listed under Week04

https://github.com/AdrianSucahyo/IEEE-PCB-Workshop-Resources-2025