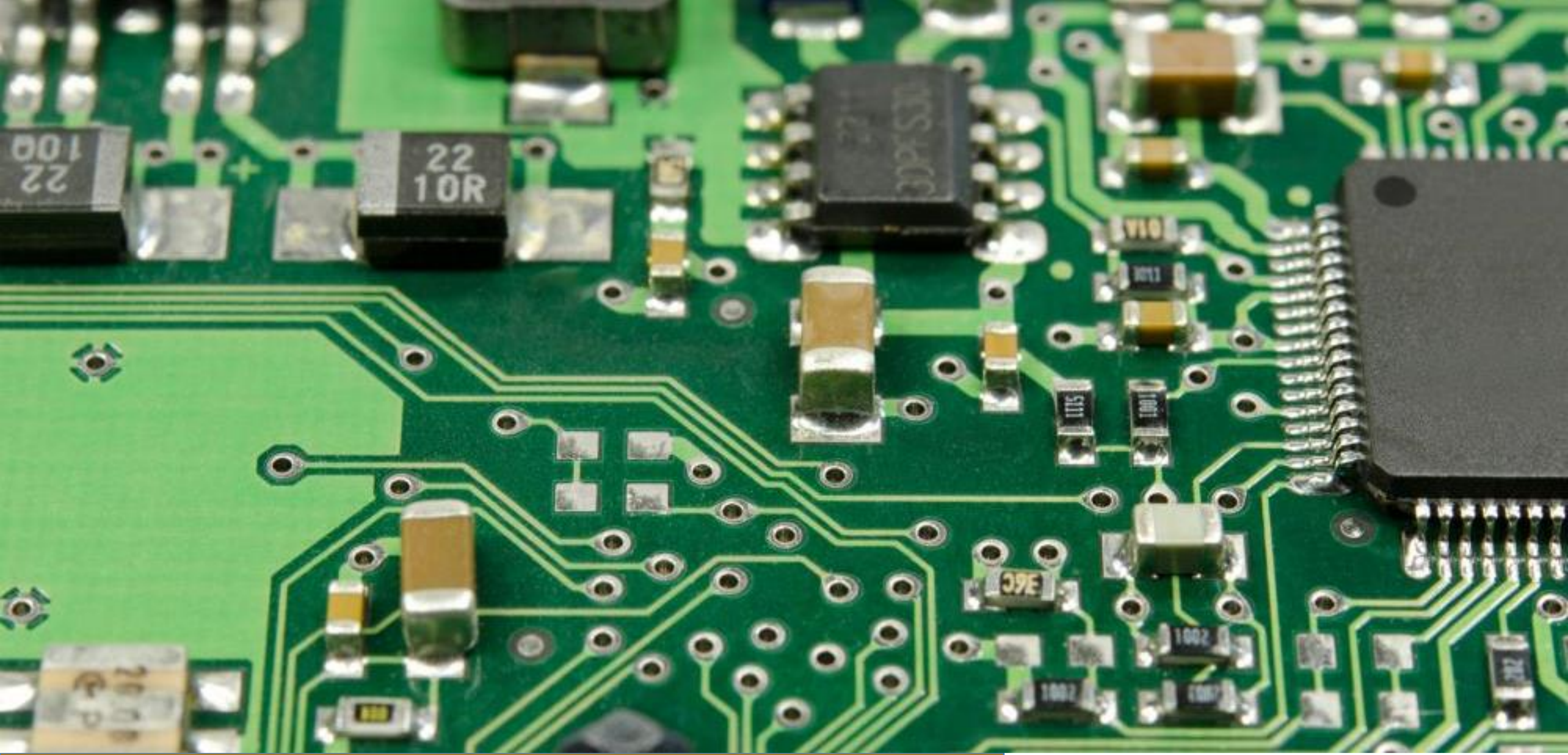


HARDWARE DESIGN TECHNIQUES

PURDUE
UNIVERSITY

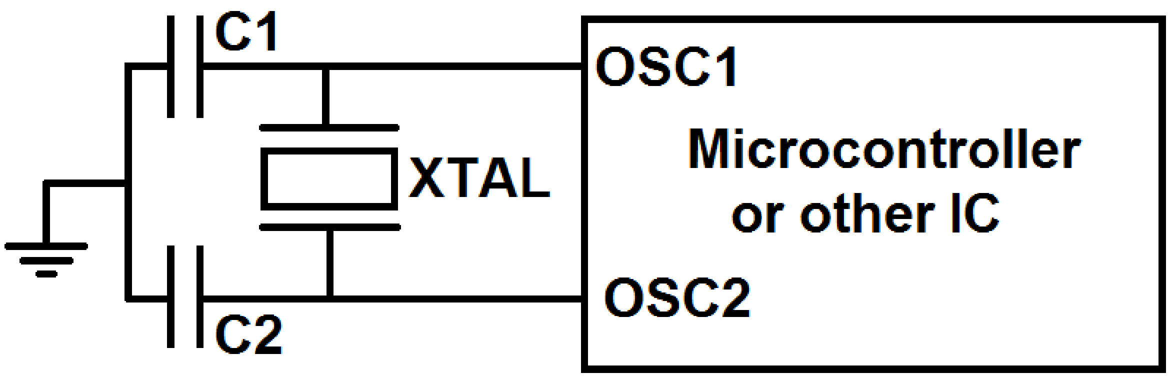
OUTLINE

- IC Support Techniques
- Layout Considerations
- Routing Considerations
- Signal Plane Considerations
- Electromagnetic Interference
- Hardware Debugging Support
- Mechanical Considerations
- Other Considerations

IC SUPPORT TECHNIQUES

Oscillator Circuits

- Many projects may require use of external oscillators (USB, high speed/precision projects, etc.)
- Leave the area beneath and immediately around clock circuits free of graces and components (reduces EMI)
- Clocks should be placed as close to ICs as possible
- Clock traces should be as close to equal length as possible



IC SUPPORT TECHNIQUES

Support Capacitors

- Digital logic necessitates the switching of a circuit between high (energized) and low states quickly. To energize quickly, transistors in digital circuits require access to a small amount of on-demand current
- Decoupling Capacitors: Small capacitors placed physically near digital ICs to provide internal transistors with a temporary source of current during switching (serve as "low pass filter" against high-frequency transient power events)
- Bulk Capacitors: Larger capacitors placed on a circuit board to reliably supply power to decoupling capacitors in the face of power fluctuations

PURDUE UNIVERSITY

IC SUPPORT TECHNIQUES

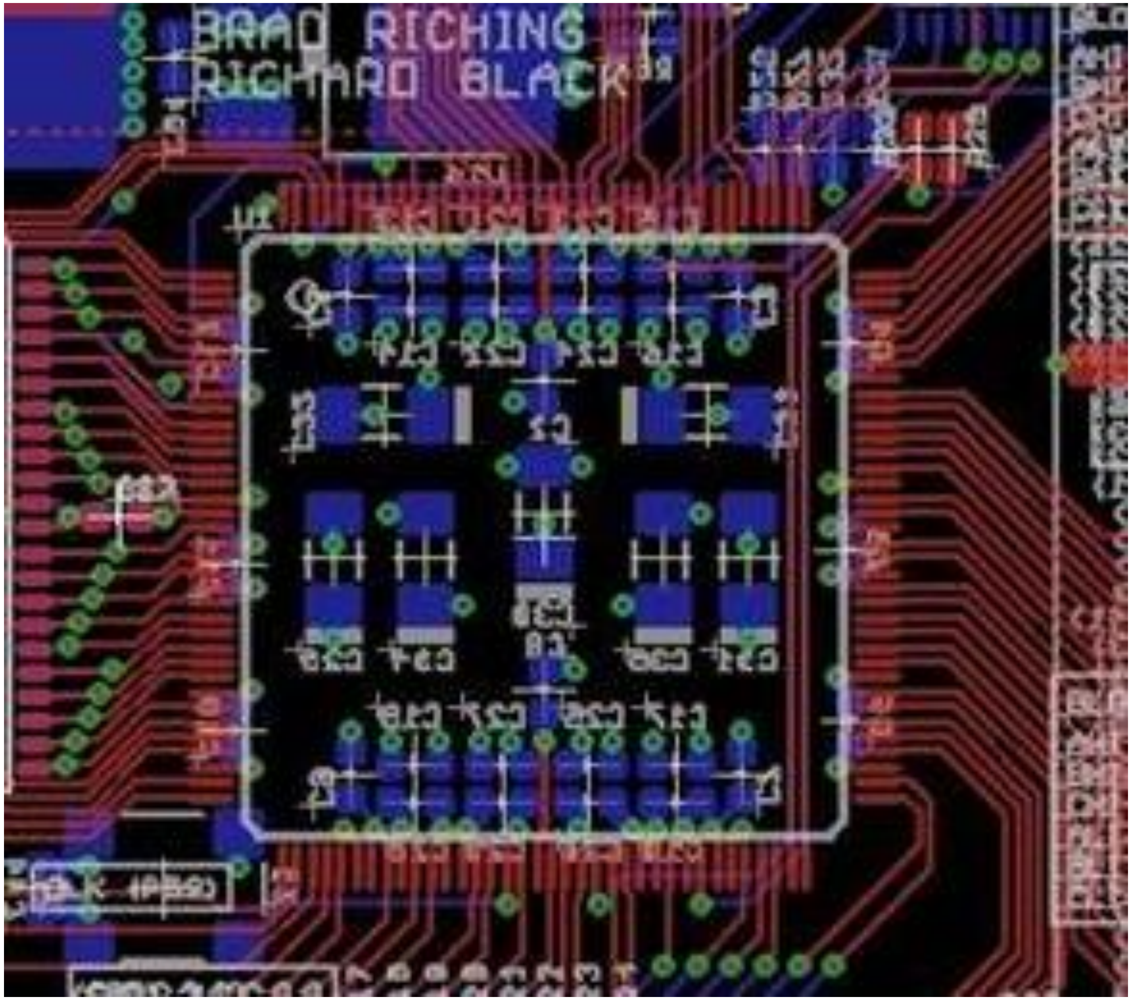
Support Capacitor Guidelines

- Capacitance value:
  - <u>Decoupling capacitors:</u> 100nF – 1µF typical (large enough to store charge but small enough to react quickly)
  - <u>Bulk capacitors:</u> Size not critical, 10uF – 100uF common
- Physical placement:
  - <u>Decoupling capacitors:</u> Place as physically close to IC as possible. Common to place underneath (opposite side of board) from the IC
  - <u>Bulk capacitor:</u> Placement generally not critical

PURDUE UNIVERSITY

IC SUPPORT TECHNIQUES

Support Capacitor Placement



PURDUE UNIVERSITY

IC SUPPORT TECHNIQUES

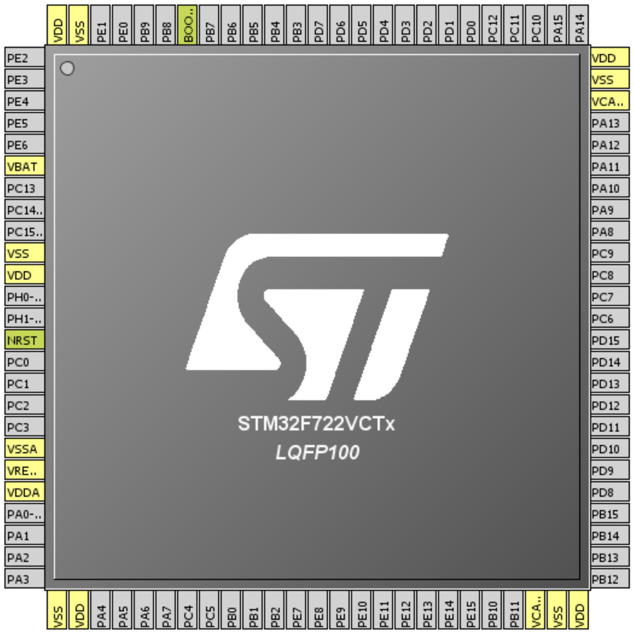
Microcontroller Setup Circuit

- For most boards used in ECE477, the microcontroller constitutes the core element of that board
- Need to know minimum microcontroller setup circuit:
  - Device documentation
  - Development board examples
- At minimum, the core MCU circuit should address:
  - Power connections (V_{DD}, Ground, Analog, Core Voltage)
  - Programmer connections
  - IC decoupling
  - Device-specific connections (Boot pins, etc.)

IC SUPPORT TECHNIQUES

Microcontroller Minimum Setup Example: STM32

- For STM32xx devices, open a given device in STM32CubeMX
- Research all colored pins, their functions, and their required hardware settings for your projects
- Decouple ALL power/ground pin pairs



- In addition to the special pins, identify and connect device programming pins (see MCU reference manual)
- <u>SWD programming pins:</u> nRST, VDD, VSS, SWDIO, SWCLK, SWO

PURDUE
UNIVERSITY

IC SUPPORT TECHNIQUES

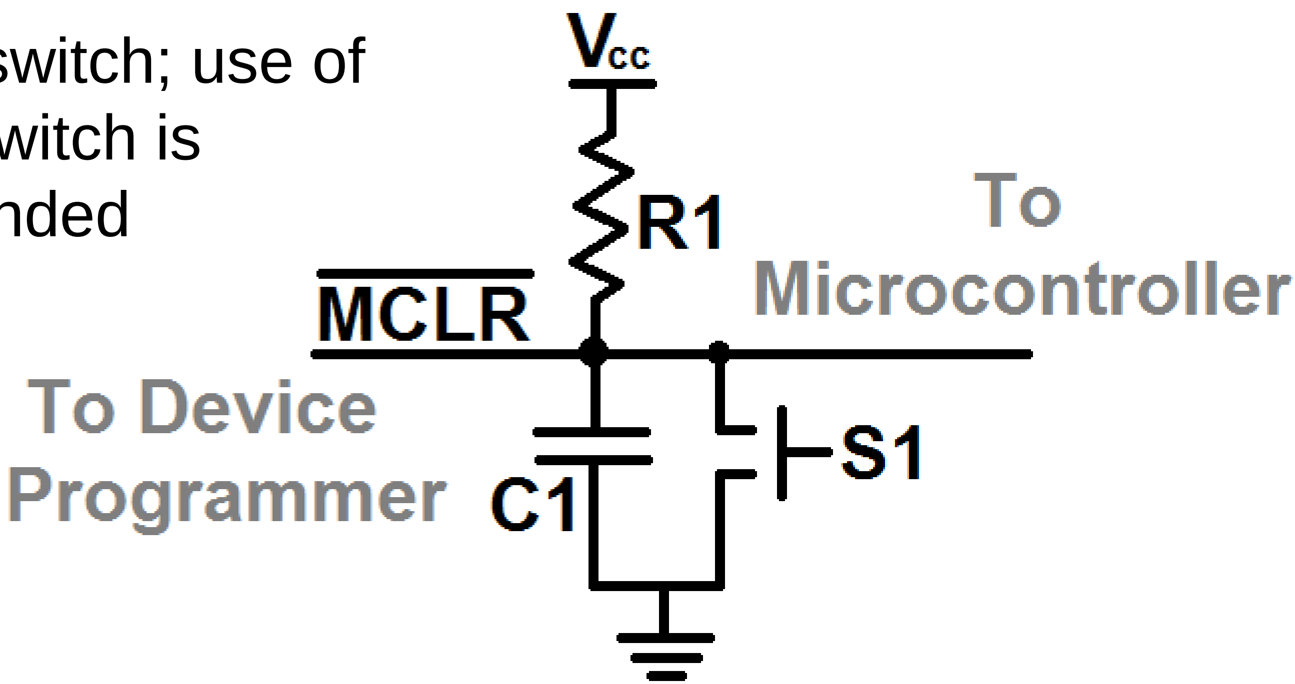
Reset Circuits

- Microcontrollers and other ICs generally must be reset to a known state prior to actions such as programming, and a reset switch is a useful inclusion in nearly all digital circuits
- Failure to properly handle device reset logic may result in spurious resets, failure to program the device, and unpredictable device behavior
- Reset lines are generally active low, and pulled high during normal device operation

IC SUPPORT TECHNIQUES

Reset Circuit Example

- Example microcontroller reset circuit
- R1 should be chosen to pull reset line high (10kΩ typical)
- C1 decouples reset line from high-frequency transient events
- S1 reset switch; use of a tactile switch is recommended

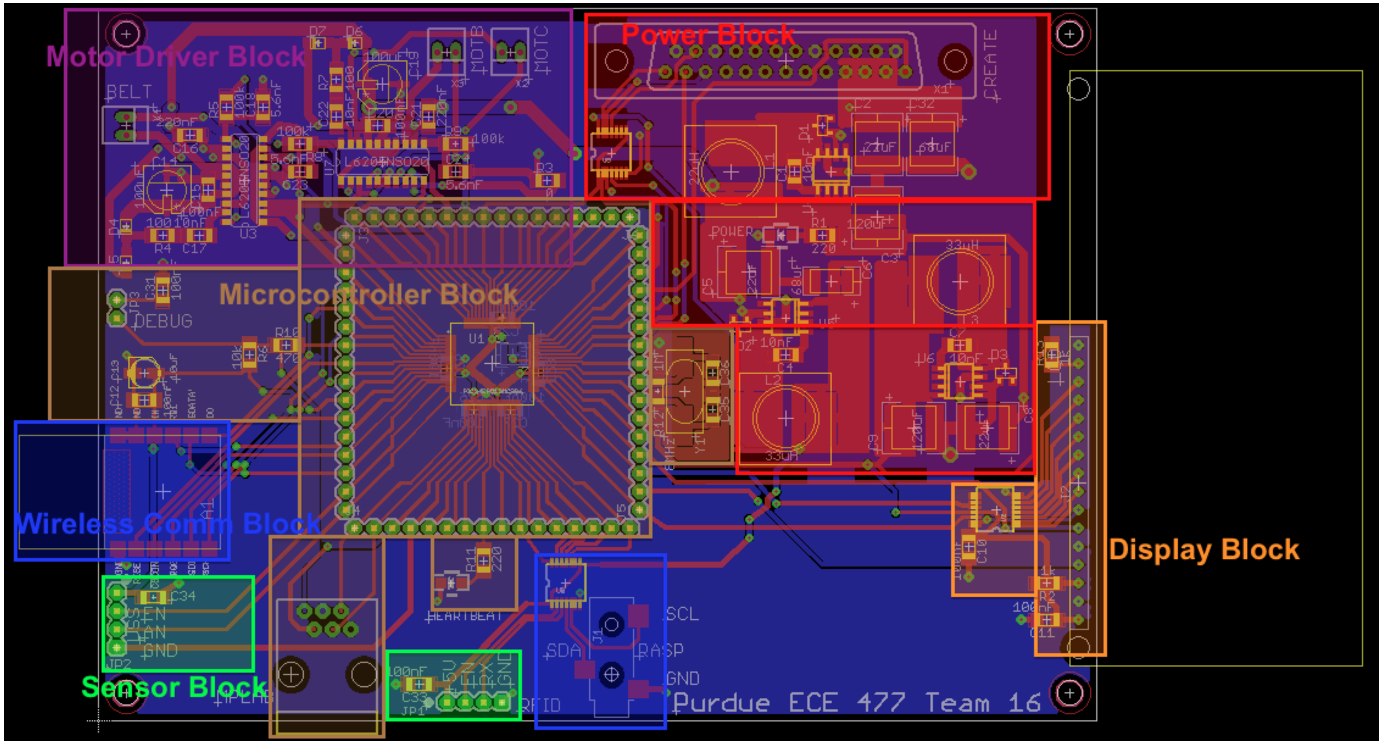


PURDUE UNIVERSITY

GENERAL LAYOUT CONSIDERATIONS

Board Organization and Separation of Circuits

- A critical element of good board design is board organization
- ALWAYS place and organize parts prior to routing a board



PURDUE
UNIVERSITY

ROUTING CONSIDERATIONS

Trace Widths and Current Capacity

- As with all wires, PCB traces are capable of carrying a certain amount of current without overheating
- Trace current capacity is dependent on:
  - Width of trace
  - Thickness of copper used in circuit board (1oz default)
- When routing a board, consider current carried by traces in determining trace widths
- Rules of thumb:
  - 8-16 mil traces for general signals
  - 24+ mil traces for power busses
- Utilize online trace width calculators to size traces (link)

PURDUE UNIVERSITY

ROUTING CONSIDERATIONS

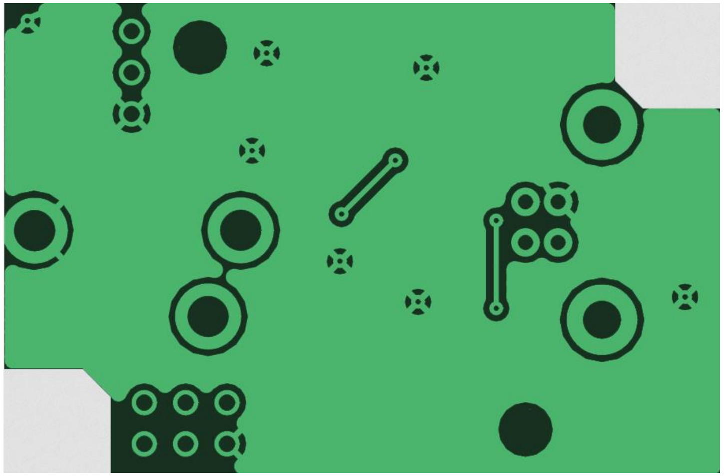
Routing Order

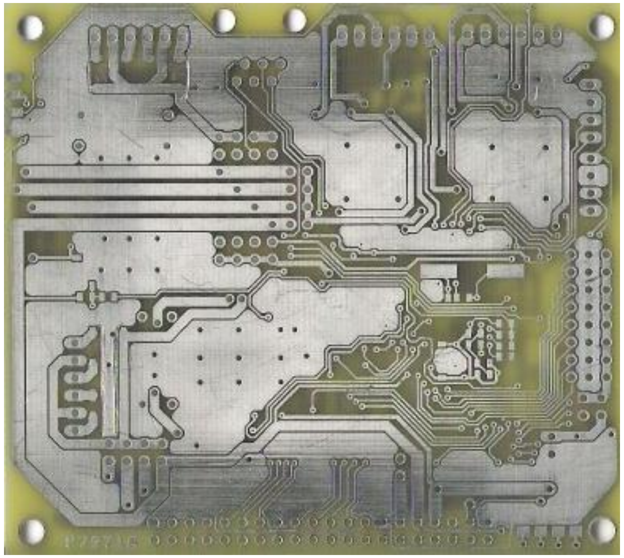
- When routing a circuit board, route systems in order of most performance critical to least
- General routing order recommendations:
  1. Oscillators and clocking circuitry
  2. Decoupling capacitors
  3. Microcontroller reset circuitry
  4. Power system circuitry
  5. General system signal routing
  6. Board-wide VDD/VSS signals

SIGNAL PLANES AND VIAS

Signal Planes

- Useful for creating large, low resistance signal paths
- Allow easy connection for common signals (e.g. GND)
- Saves copper (unused copper on a circuit board must be etched away)
- Can be used as PCB heatsinks (reduce or eliminate the need for external, dedicated heatsinks)

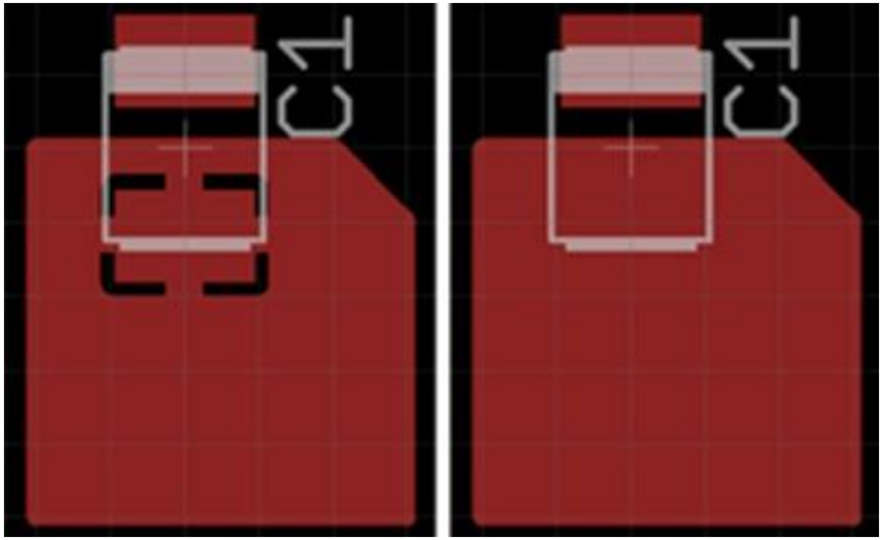




SIGNAL PLANES AND VIAS

Thermal Relief

- Electronic components must be soldered to boards; in order to solder (or desolder) a part, the copper of a component pin/pad must be raised to a sufficient temperature
- If a pin/pad is given a direct connection to a thick trace or large signal plane, the trace or plane may act as a heatsink, preventing the pin/pad from reaching soldering temperature
- <u>Thermal Relief:</u> Isolate a pin/pad from direct connection with large trace/plane (heatsink)

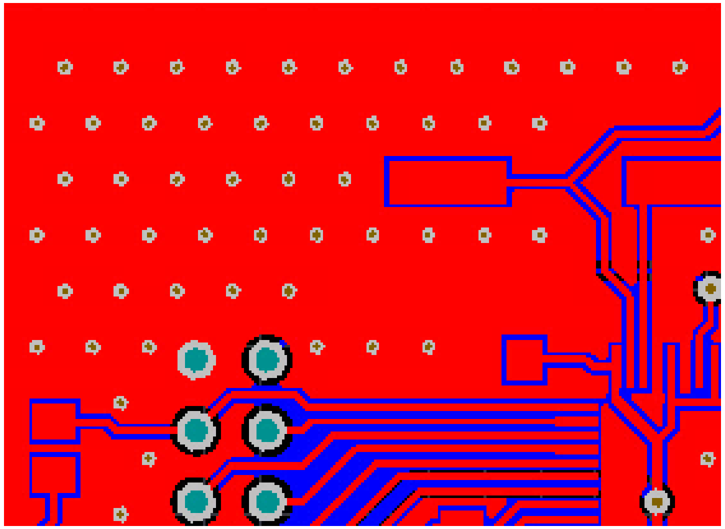


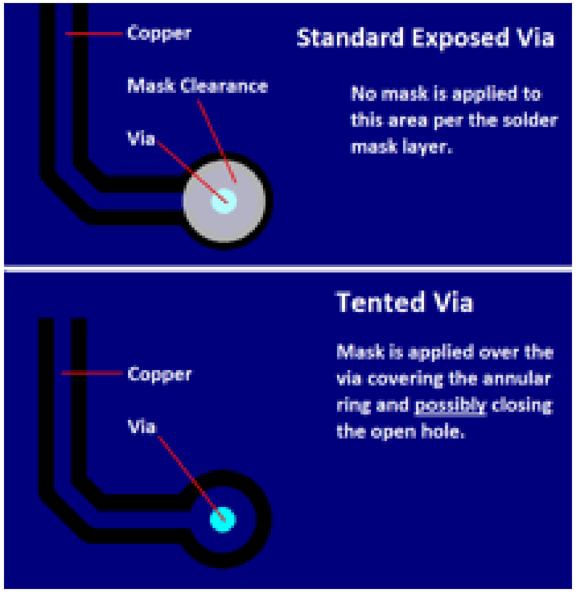
PURDUE
UNIVERSITY

SIGNAL PLANES AND VIAS

Stitching Vias and Tented Vias

- Signal planes for a given signal on opposite sides of the board only meet at cross-layer connections. Use <u>stitching vias</u> to connect a signal plane which exists on both sides of a board (also improves noise performance)
- <u>Tented Vias:</u> Design the board to allow the solder mask to run over the top of a via (provides unbroken surface for more legible silkscreen, etc.)





PURDUE UNIVERSITY

ELECTROMAGNETIC INTERFERENCE

Motivating Question

- Circuit boards often involve traces in close proximity to each other and switching digital waveforms

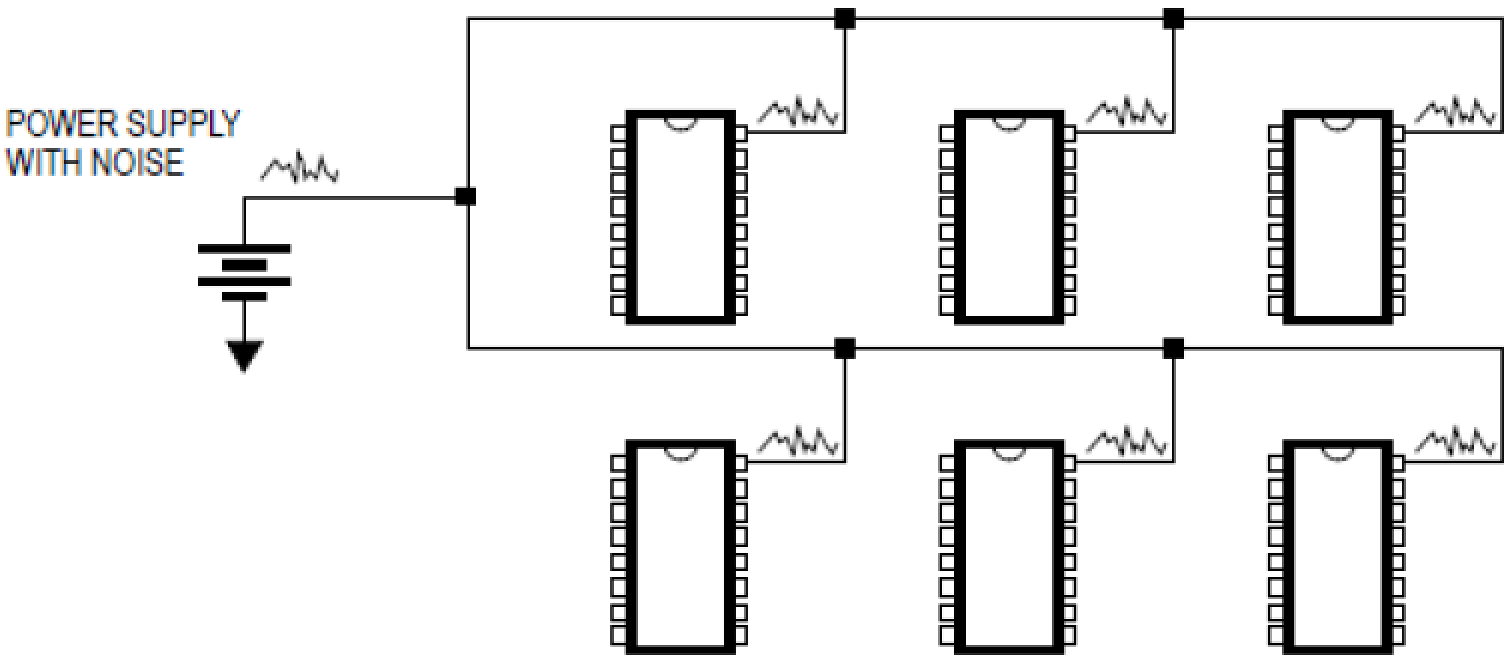
- Q1: If noise is present on a conductor, what will the current/voltage characteristics be of any connected conductors?
- Q2: What does the frequency response of a pulse function look like?
- Q3: If two wires are placed close together and current flows through one of them, what will happen to the current/voltage of the second wire?

PURDUE UNIVERSITY

ELECTROMAGNETIC INTERFERENCE

Electromagnetic Interference

- A1: Noise on a conductor will propagate to any shared conductors

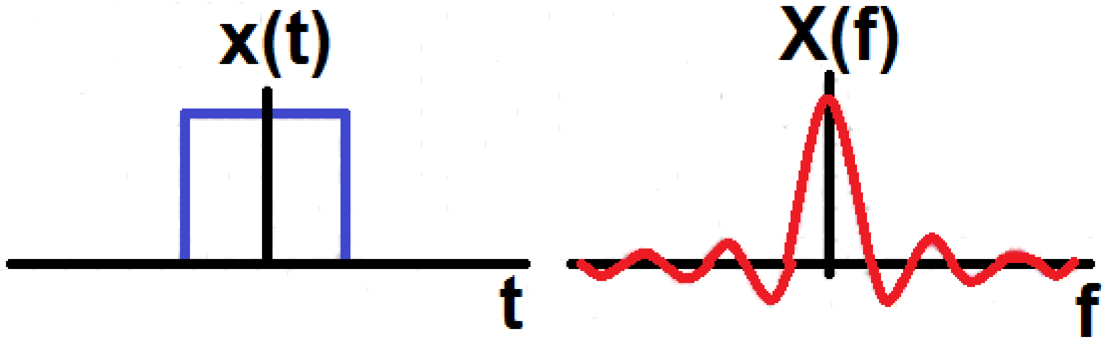


PURDUE
UNIVERSITY

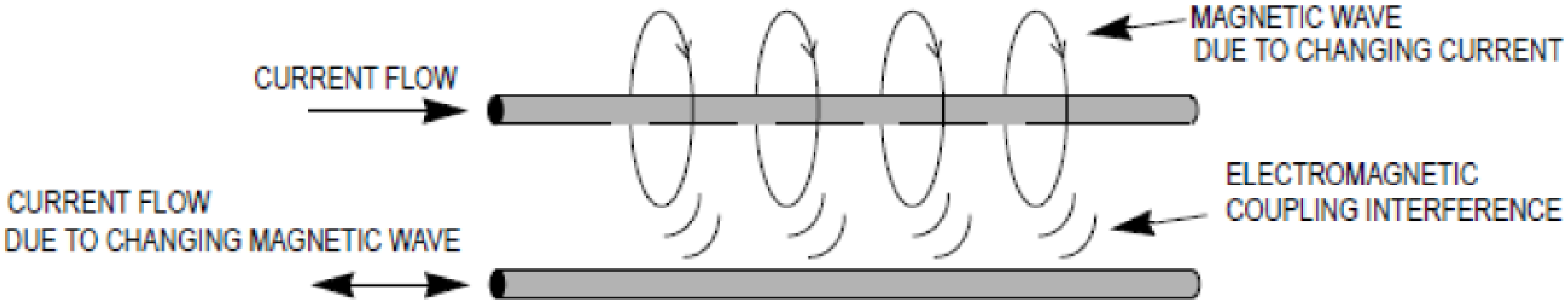
ELECTROMAGNETIC INTERFERENCE

Electromagnetic Interference

- A2: Frequency content of a pulse function = wideband noise



- A3: When wires are placed close together, electromagnetic coupling can occur



PURDUE
UNIVERSITY

ELECTROMAGNETIC INTERFERENCE

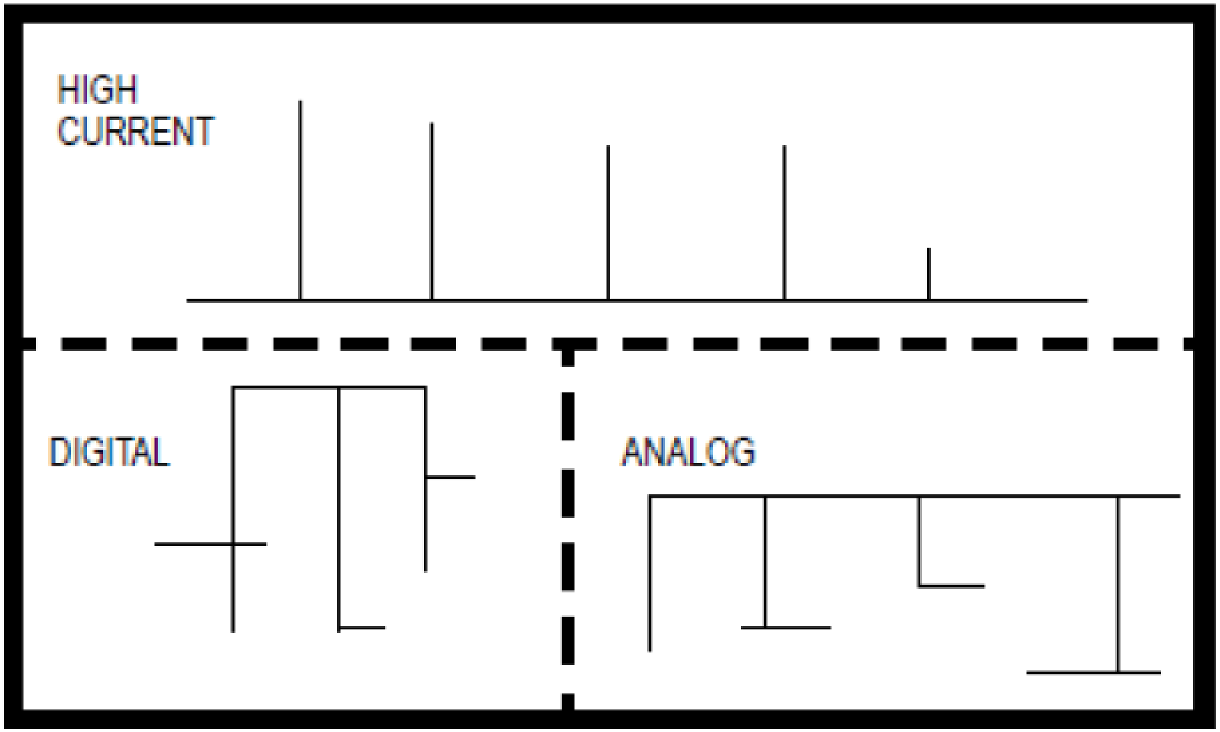
Design for Electromagnetic Compatibility (EMC)

- A circuit is electrically compatible if it does not affect or become affected by its environment
- Remedies:
  - Decrease emissions – can be suppressed at the source through proper system design
  - Shielding – protect sensitive areas of the circuit from emissions
  - Increase noise immunity – susceptibility to noise can be decreased by “hardening” the circuit’s design

ELECTROMAGNETIC INTERFERENCE

Design for Electromagnetic Compatibility (EMC)

- Separation of circuits on a PCB
  - Where possible, separate out circuits by function or type, and minimize interactions between subsystems:



PURDUE
UNIVERSITY

ELECTROMAGNETIC INTERFERENCE

Design for Electromagnetic Compatibility (EMC)

- Perpendicular Routing:
  - In spots where analog and digital lines must be routed through the same area of the board, routing them perpendicular to one another will help to minimize cross coupling
- Ground plane shielding
  - Surrounding sensitive traces with a ground plane can help minimize noise introduction into the trace through other noise sources

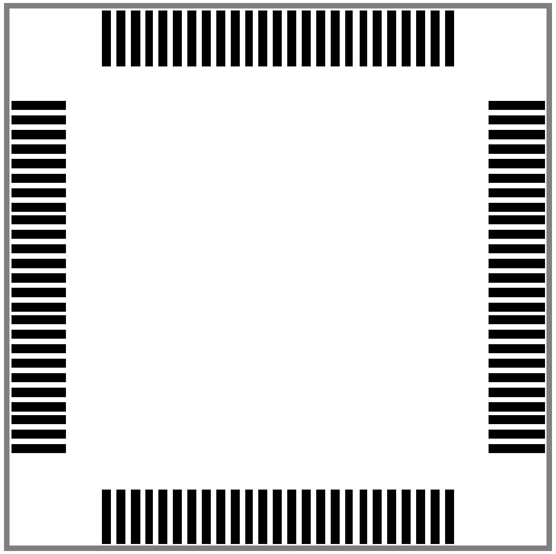
Design for Electromagnetic Compatibility (EMC)

- Analog and Digital Grounding:
  - Generally, analog and digital circuits should be given separate analog ground (AGND) and digital grounds (GND). These circuits can then be joined together at a finite number of points.
  - Access to the analog ground can be controlled through the use of through-hole jumpers or solder bridges (sometimes referred to as a 0Ω resistor)

DEBUGGING AND VERIFICATION

Chip Polarity

- IC packages usually involve some degree of rotational symmetry. In the footprint to the right, which pin is pin 1?
- For every device that involves a specific polarity (diodes, LEDs, ICs, connectors, etc.) pin 1 should be clearly marked in a way that is easy to determine



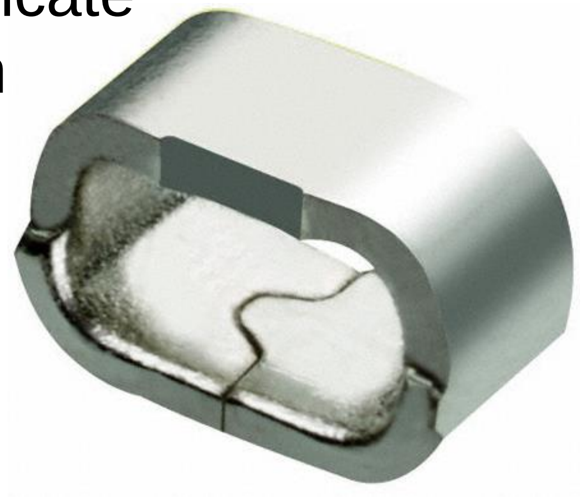
- Markings for device polarity should be visible even after components are soldered, for ease of debugging and verification (if pin marking is under an IC, then the IC will cover it up and slow down debugging later)

PURDUE UNIVERSITY

DEBUGGING AND VERIFICATION

Breakout Pins, Test Points, and LEDs

- Hardware can be incorporated into designs to greatly improve the ease with which a project can be debugged
- Breakout pins: Provide access to signals, debugging interfaces, and future functionality not yet in design
- Test Points: Vias, pads, or other parts to provide access to signals with measurement tools
- Debugging LEDs: Can be used to indicate code execution has reached a certain state in the code, indicate power is applied to the board (power led) and indicate the device is alive and executing (heartbeat LED)



PURDUE UNIVERSITY

MECHANICAL CONSIDERATIONS

Space Conflicts and Accessibility

- 2D layout packages (such as Eagle) can sometimes miss important considerations in the Z-axis
- Consider:
  - If 2+ boards are stacked, what will you be able to access in the covered area? (Answer: nothing)
  - How will your board fit into your packaging when you consider the height of “tall” components such as connectors and capacitors?

PURDUE
UNIVERSITY

MECHANICAL CONSIDERATIONS

Space Conflicts and Accessibility Solutions

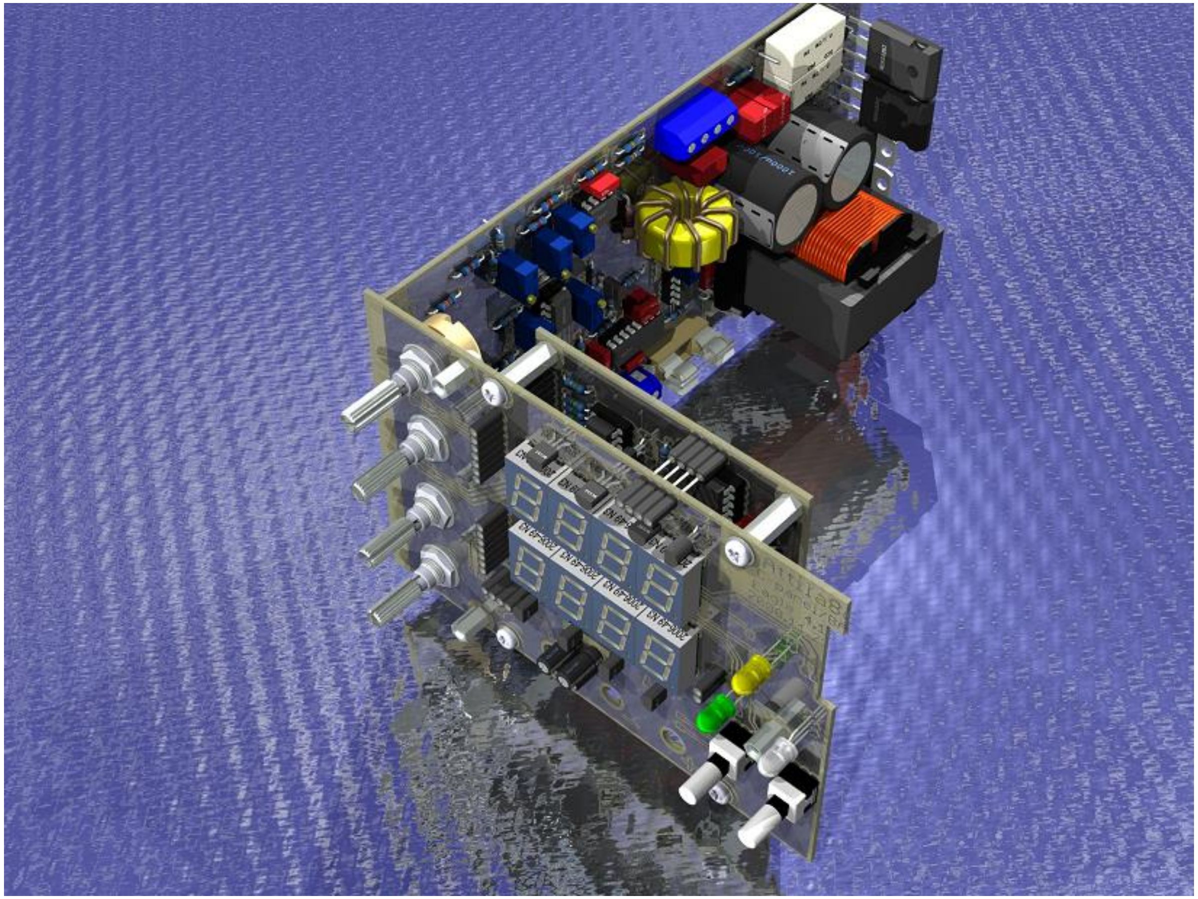
- A few techniques to deal with space conflicts and accessibility:
  - When performing board layouts, ask yourself “How will this component work in 3 dimensional space? What could it run into?”
  - In 2D layout packages, include the X/Y dimensions of every component in the documentation (or other) layer. (For example, include the dimensions of a breakout board rather than a simple pin header to access it)
  - Use a 3D tool such as Eagle3D to generate a three dimensional model of your board to spot check for issues

PURDUE
UNIVERSITY

MECHANICAL CONSIDERATIONS

Space Conflicts and Accessibility Solutions

- Example Eagle3D Output:



PURDUE UNIVERSITY

MECHANICAL CONSIDERATIONS

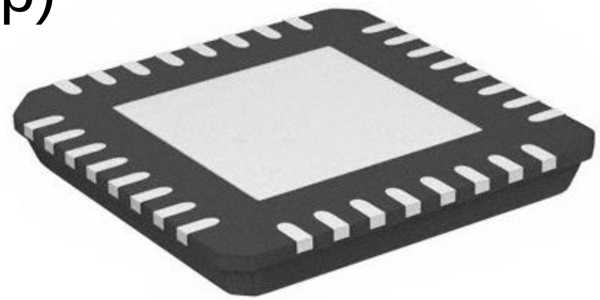
Connectors and Standoffs

- Standoffs: Most circuit boards should include 3+ mounting holes to allow the board to be securely fastened to its project packaging
  - Surrounding mounting hole connections with grounded vias is a common practice for emissions and mechanical stability
- Connectors: Place connectors and other user I/O (such as SD card slots or buttons/switches) on the edges of the board to enable easy access
  - Adding isolation and current limiting to offboard connections are common practices

OTHER CONSIDERATIONS

Under Chip Pads

- Some ICs feature exposed (under chip) pads on the bottom of the package for thermal, mechanical, or electrical reasons



- Accessing these pads for soldering purposes generally requires the use of reflow soldering techniques

- A workaround for hand soldering is to place vias with large diameter holes into the exposed chip pads. The holes must be large enough to allow the tip of a soldering iron to pass through, enabling the part to be soldered.

PURDUE UNIVERSITY

OTHER CONSIDERATIONS

Silkscreen Topics

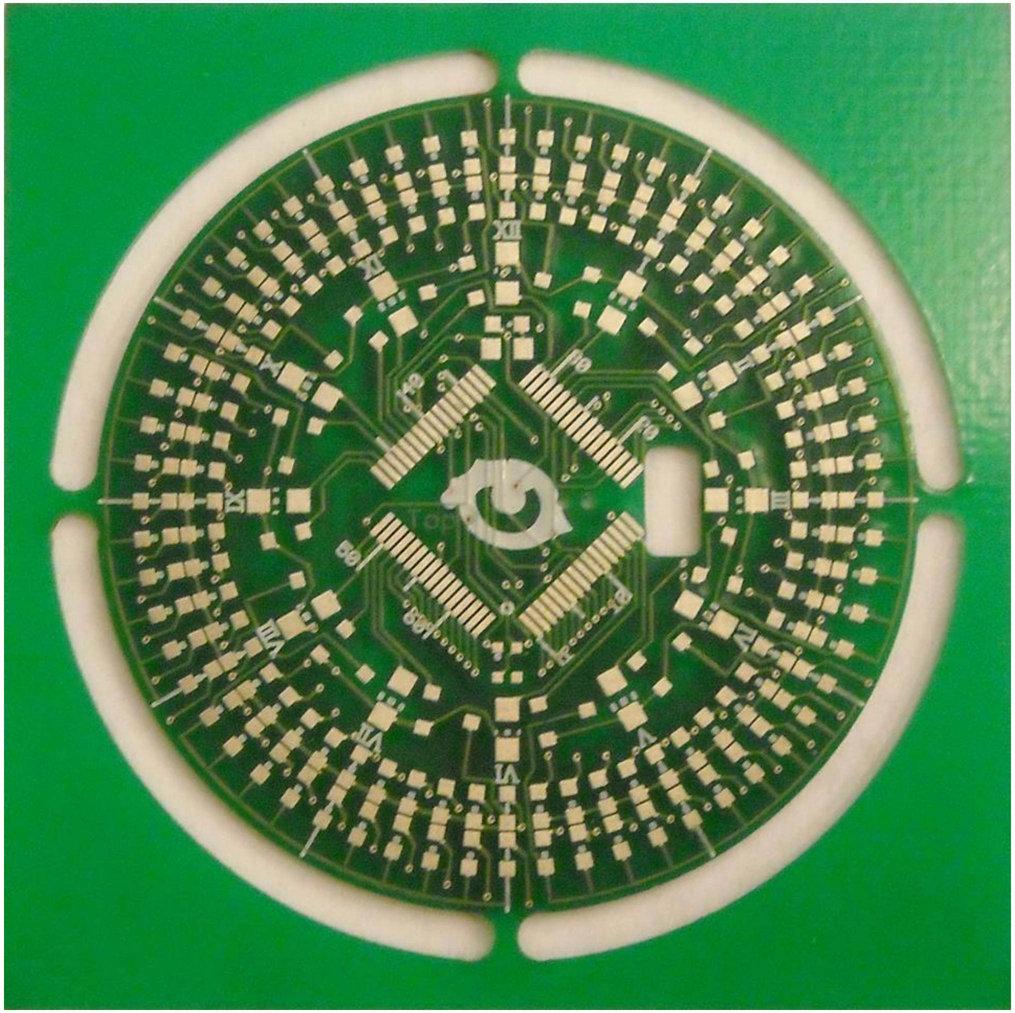
- Every silkscreen layer should include:
  - Name of author(s) (ECE477 team members)
  - Name to identify the circuit board
  - Date last modified or revision number
- Placing component values in silkscreen may be unnecessary; consider instead giving components IDs (R1, C1, L1, IC1, D1, S1, etc.) and using table of values
- Custom silkscreen images can enable addition of custom logos and image to circuit boards

PURDUE UNIVERSITY

OTHER CONSIDERATIONS

Polar Layouts and Component Rotations

- <u>Polar Layouts:</u> For use in circular designs and RF
- <u>Part Rotations:</u> May simplify board routing to rotate to non-90° angle



Questions?