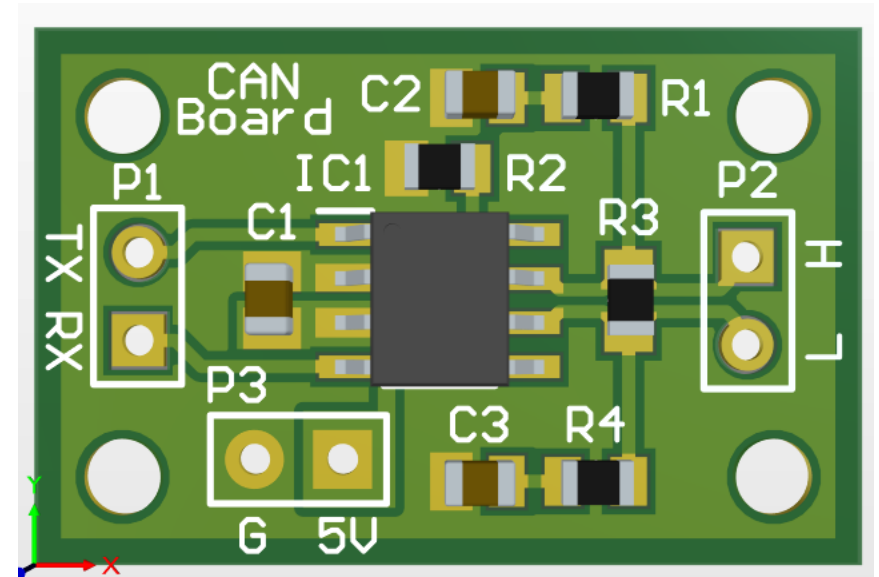


# Welcome to Part 2 of the Altium Designer Workshop!



# Workshop Overview

- ✓ **Part 1: Schematic Design**
  - ✓ Basics of Altium Designer
  - ✓ Adding components
  - ✓ Connecting components
  - ✓ Bonus Material: Hierarchical Schematics
- **Part 2: PCB Design**
  - Moving from schematic to PCB layout
  - Layout tips and best practices
  - Routing techniques
  - Silkscreen tips
  - Bonus Material: Multilayer PCBs
- **Final Deliverable: 2-layer CAN transceiver PCB design**



# Getting Started!

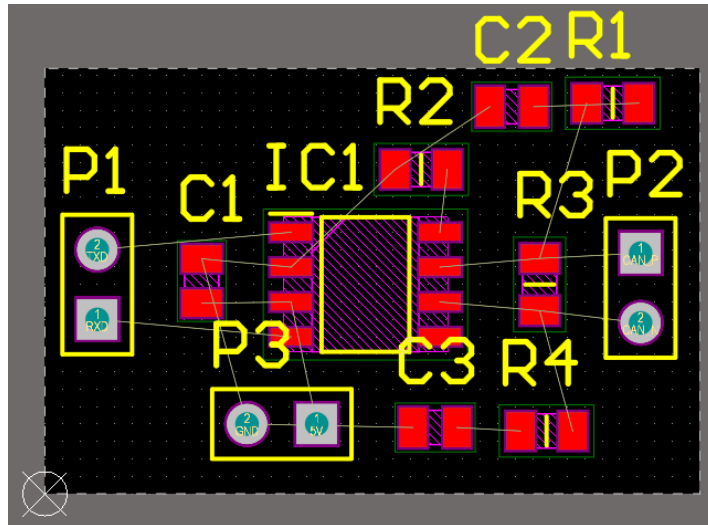
## Tasks:

- <https://tinyurl.com/AltiumFall2022>
- **Make a new PCB file (and save it!)**
- **Set origin to bottom left corner**
- **Edit board shape/size (940 mil x 610 mil)**
  - **Remember to be in Board Planning Mode (1 on your keyboard)**

# Beginning Your Layout

## Tasks:

- Import changes from schematic to PCB
- Do an initial layout (rough draft!)



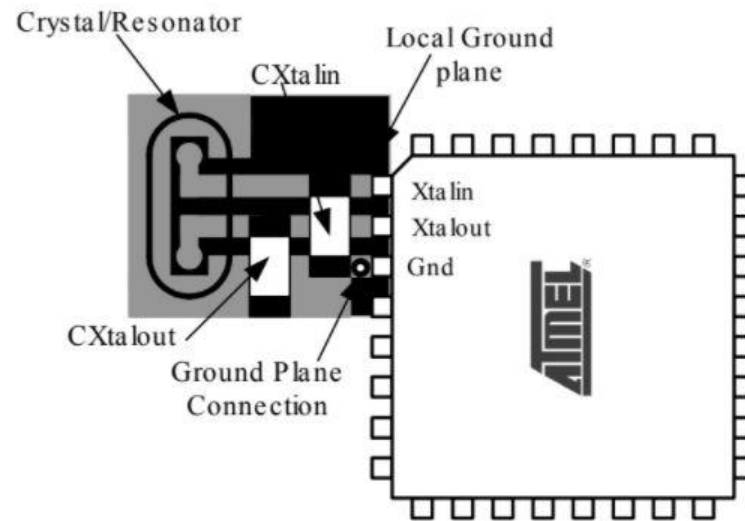
## Shortcuts:

- 1 = Board Planning Mode
- 2 = 2D Mode
- 3 = 3D Mode
- Ctrl + Scroll = Zoom
- Right Click + Drag = Move schematic/PCB
- Click and Hold + Space = Rotate component
- Backspace = Undo Net/Trace Segment
- Esc = Exit active tool
- Right click + drag from right to left = select entire object from part of it
- Right click + drag from left to right = select object that is completely covered
- Clicking mouse wheel + moving up/down = Zoom In/Out
- Scrolling up/down = move view up/down
- Shift + scroll up or down = move left/right

# A Bit of Theory - Clocks

## Important Components on the Board

- **Crystal oscillator (“clock”)**
  - **Keep the clock happy!**
  - **Route XTAL traces close to the same length**
  - **Provide GND shielding around clock and its supporting components**
  - **Have clock and its supporting components close to the IC**



# A Bit of Theory – Analog Components

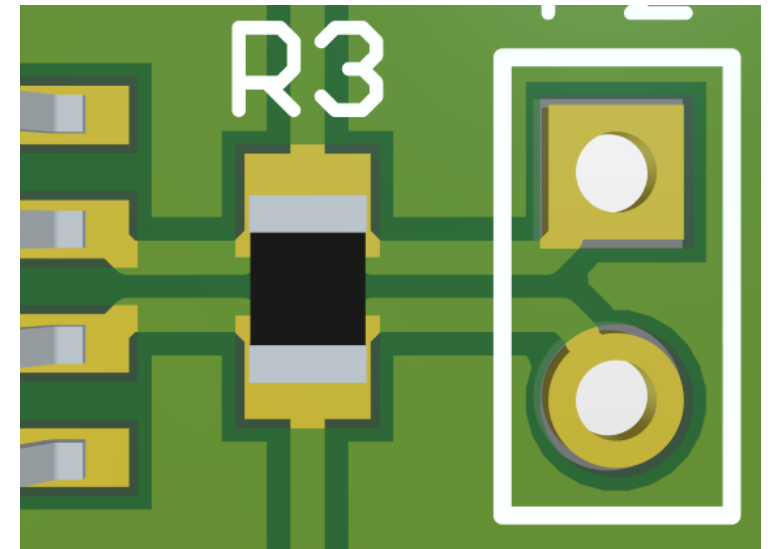
## Important Components on the Board

- **Anything analog**
  - **Analog things are sensitive to noise, so keep them away from digital signals and things that do a lot of switching**
  - **Really just keep sensitive components away from noisy lines (protect the clock, protect analog circuits, etc.)**

# A Bit of Theory

## Important Components on the Board

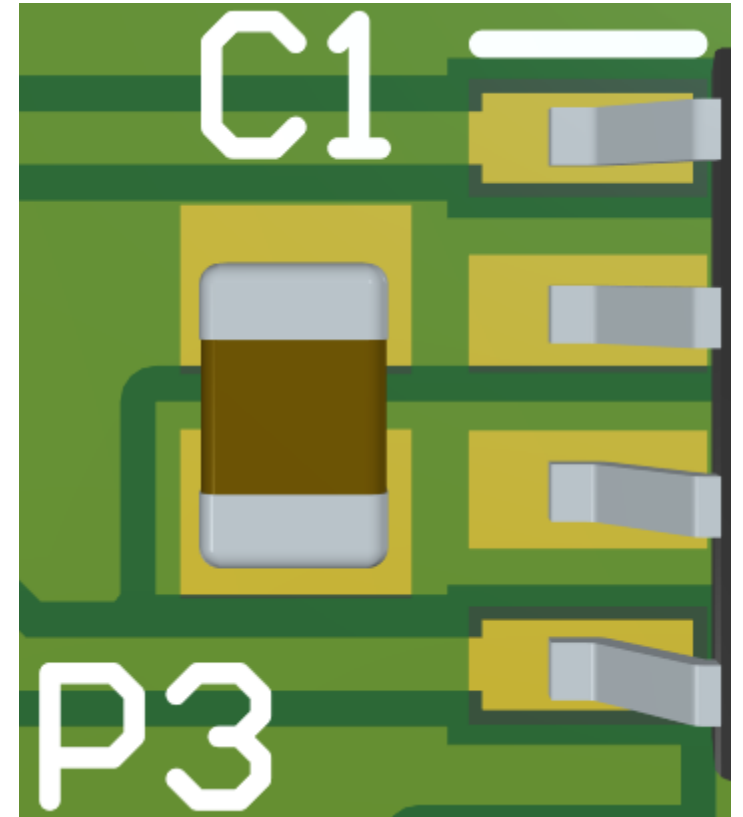
- **Components connected to differential pairs or timing-sensitive signals**
  - Make sure traces are close to the same length
  - Plan ahead by placing the associated components near each other



# A Bit of Theory

## Important Components on the Board

- **Bypass/Decoupling Capacitors**
  - Put them as close to the IC as possible to minimize parasitic inductance





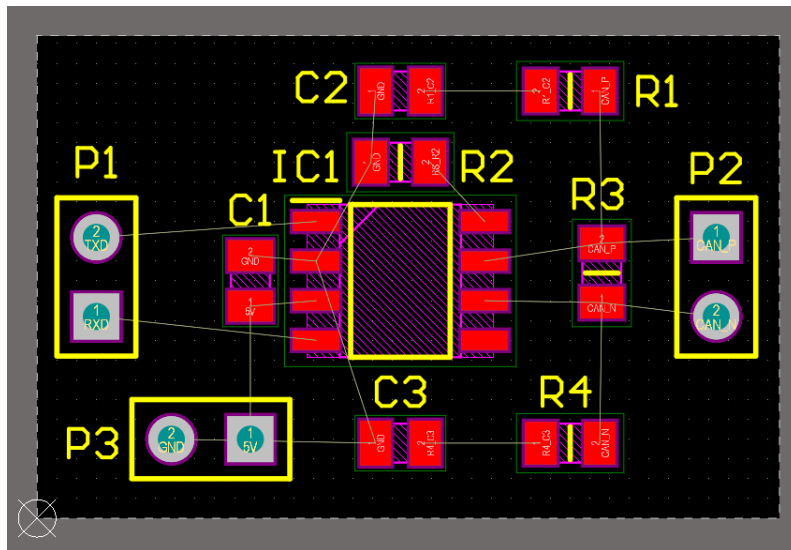
# Tips and Tricks

- **Alignment Tool**
  - Align vertical/horizontal centers
  - Align top/bottom/left/right
- **Selection Filter**
  - Don't forget to turn it off when you're done!

# Fine-Tuning Your Layout

## Tasks:

- **Fine-tune your layout with the tips and theory discussed in the previous slides**



## Shortcuts:

- 1 = Board Planning Mode
- 2 = 2D Mode
- 3 = 3D Mode
- Ctrl + Scroll = Zoom
- Right Click + Drag = Move schematic/PCB
- Click and Hold + Space = Rotate component
- Backspace = Undo Net/Trace Segment
- Esc = Exit active tool
- Right click + drag from right to left = select entire object from part of it
- Right click + drag from left to right = select object that is completely covered
- Clicking mouse wheel + moving up/down = Zoom In/Out
- Scrolling up/down = move view up/down
- Shift + scroll up or down = move left/right

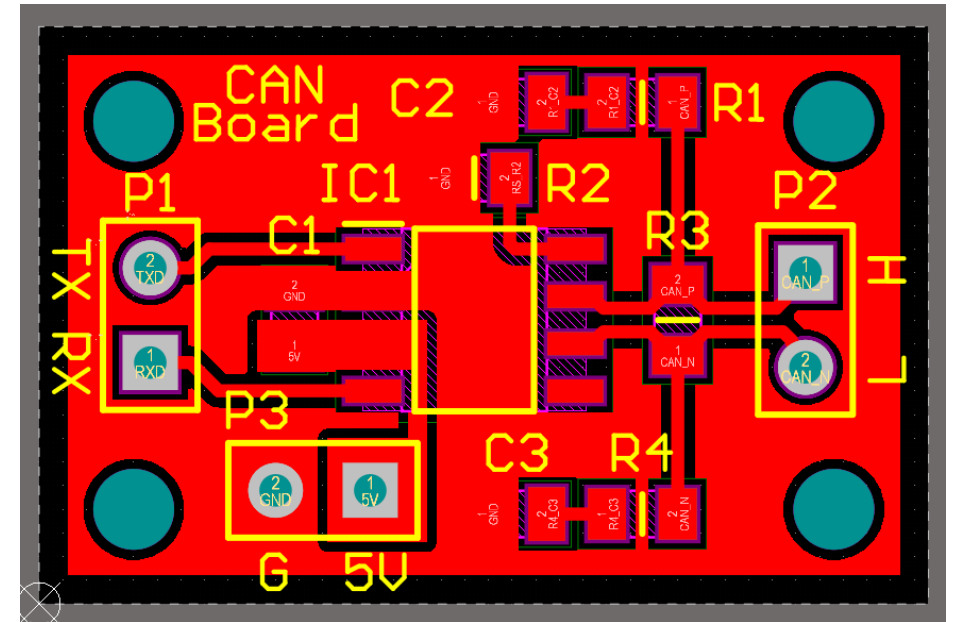
# Updating Rules

- **Adjusting Design Rules**
  - **Default trace width (5 mil min, 15 mil preferred, 20 mil max)**
  - **Differential trace width (15 mil)**
  - **Change polygon connect style to direct connect**
  - **Set board outline clearance so polygons stay 30 mil from edge**



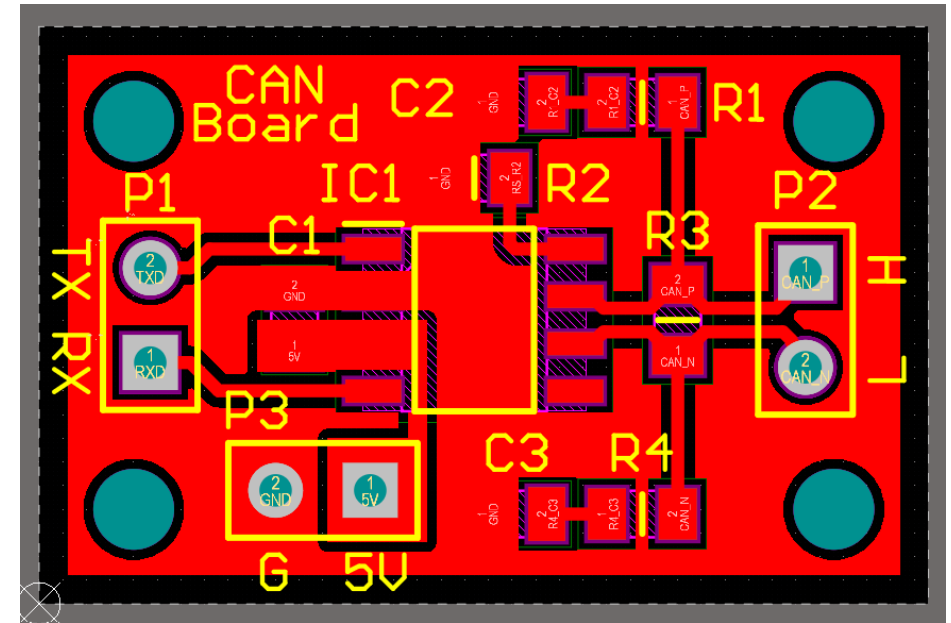
# Route the Board!

- Route differential traces (15 mil trace width)
- Route single traces (15 mil trace width)
- Add polygons for power
  - 5V polygon covering 5V pad of IC, 5V pad of bypass capacitor, and 5V pad of power header
  - GND polygon across top of board



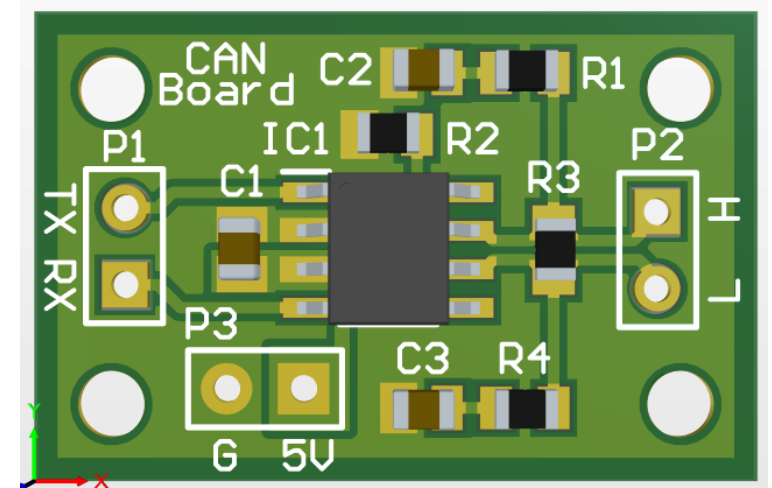
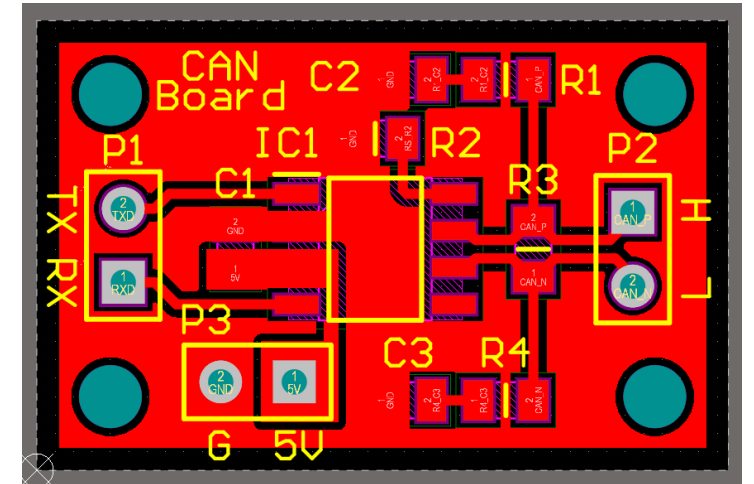
# Add (Useful) Silkscreen

- Add Tx/Rx labels to single-ended signal header
- Add L/H labels to differential pair header
- Add 5V/G labels to power header
- Add board title
- Clean everything up!
  - Make sure the silkscreen is aligned and facing logical directions



# Adding Test Points and Mounting Holes

- Add a test point to the GND net
- Add 4 M2 mounting holes
  - Each mounting hole should be 100 mil from the edges of the corner
  - M2 mounting holes have a hole size of 86.6 mil



# Preparing the Layer Stack

## \*OPTIONAL\* Tasks:

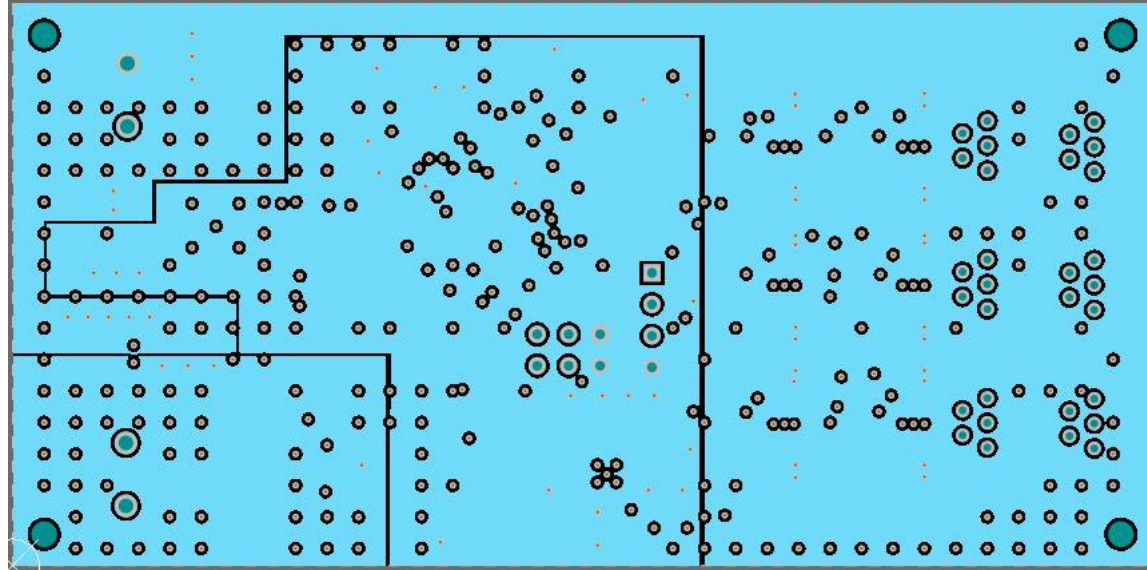
- Set up the layer stack manager so that you have four layers
  - (Top, GND, Power, Bottom)

#	Name	Material	Type	Weight	Thickness	Dk	Df
	Top Overlay		Overlay				
	Top Solder	Solder Resist	Solder Mask		0.4mil	3.5	
1	Top Layer		Signal	1oz	1.4mil		
	Dielectric 2	PP-006	Prepreg		2.8mil	4.1	0.02
2	GND	CF-004	Signal	1oz	1.378mil		
	Dielectric 1	FR-4	Dielectric		12.6mil	4.8	
3	Power	CF-004	Signal	1oz	1.378mil		
	Dielectric 3	PP-006	Prepreg		2.8mil	4.1	0.02
4	Bottom Layer		Signal	1oz	1.4mil		
	Bottom Solder	Solder Resist	Solder Mask		0.4mil	3.5	
	Bottom Overlay		Overlay				

# Preparing the Layers

## \*OPTIONAL\* Tasks:

- Pour a GND polygon across the power layer
- Pour a 5V polygon on the power layer





# Use Vias!

## \*OPTIONAL\* Tasks:

- Try to make a power polygon around one of the 5V pads and place vias in it to connect to the power layer
- Try to use a via to make a trace switch layers before connecting to another pad
- Try to make a ground pour across the top layer and a ground pour across the bottom layer. Add via stitching to the GND net.

# Congratulations!

## You Now Know How To:

- Make a new PCB file
- Lay out components
- Route components using single-ended and differential traces
- Use polygons and vias
- Adjust the layer stack for multilayer boards

If you ever have questions about Altium, feel free to reach out at [emarshall33@gatech.edu](mailto:emarshall33@gatech.edu)!