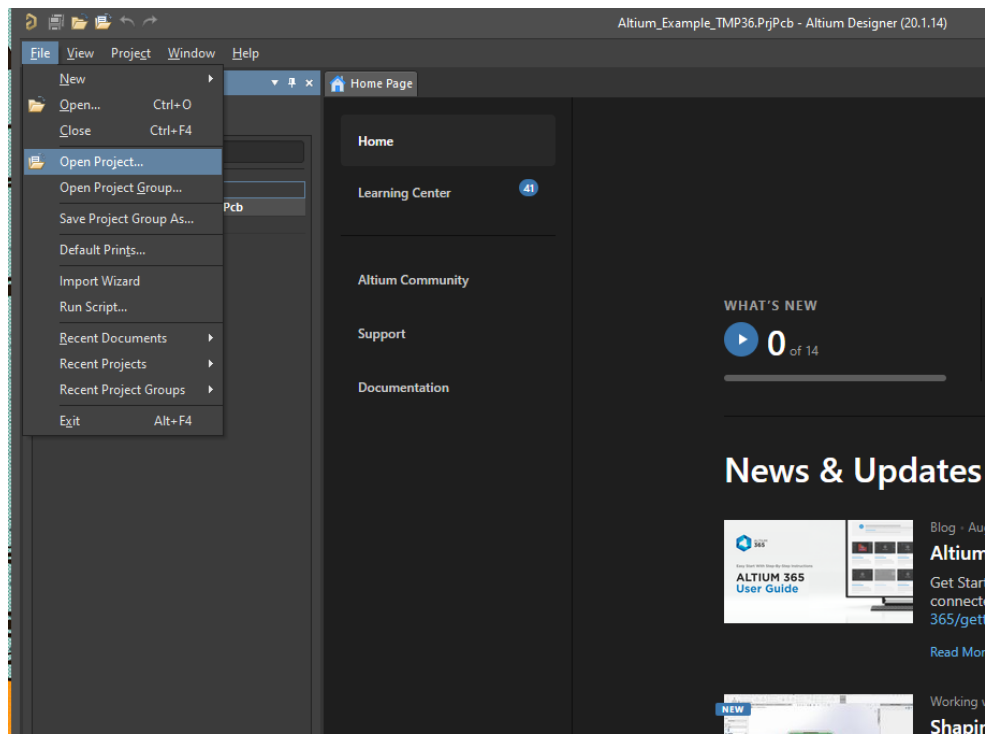
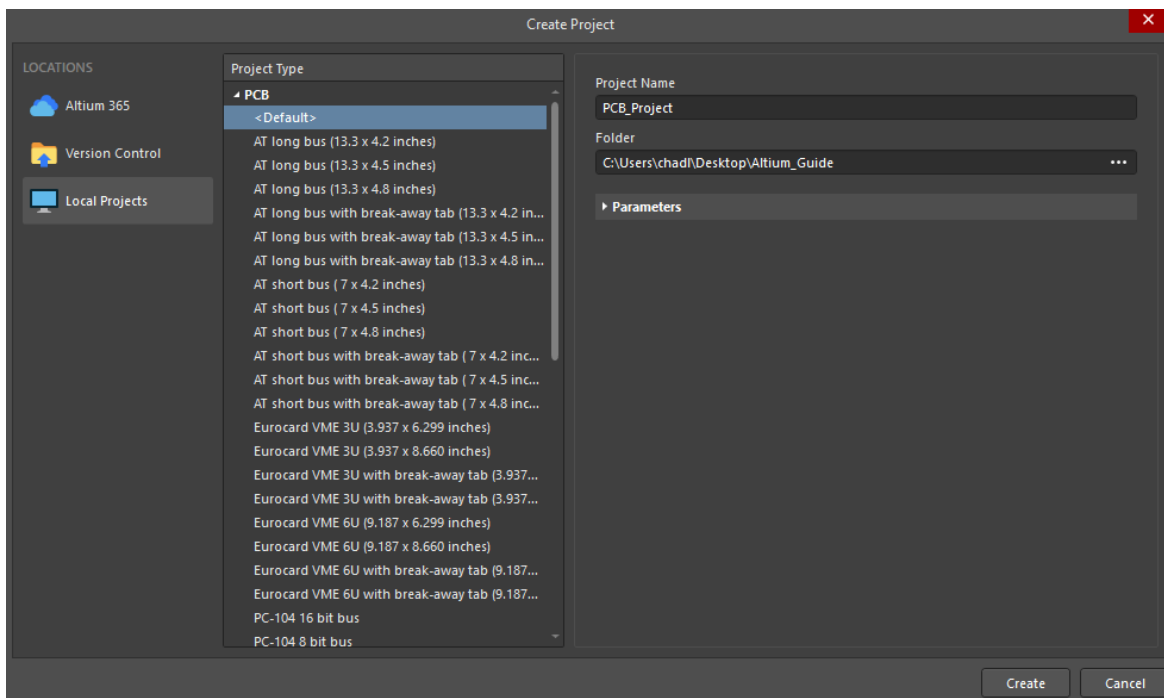


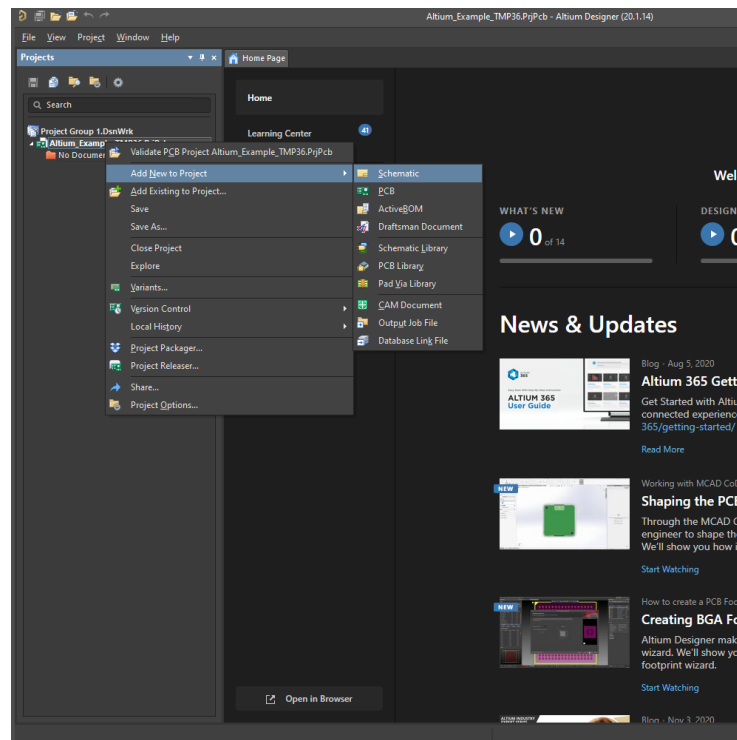
1. Create a project in Altium:



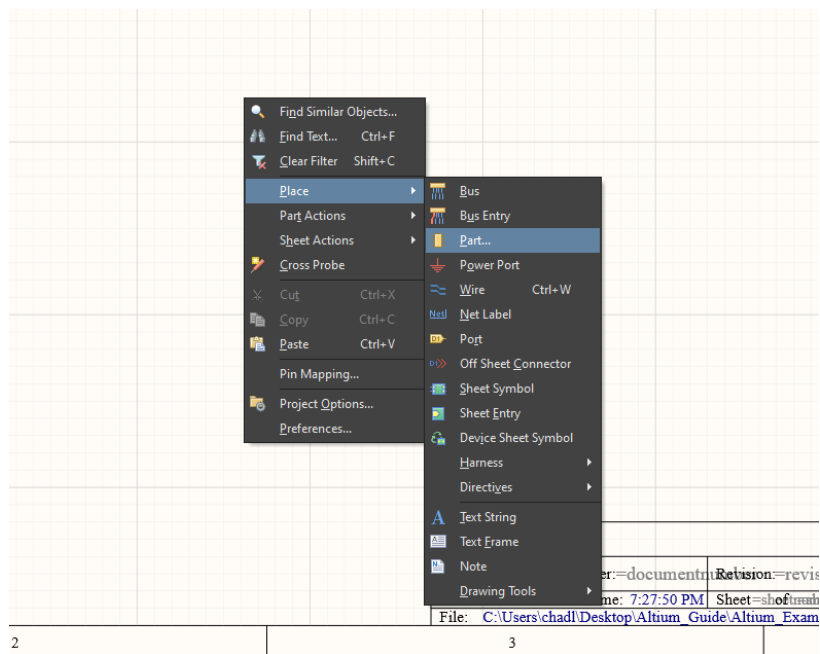
Choose a place for your project:



2. Add schematic file to project:



Place part by right clicking the schematic > place > part (or hit “p” key twice)



Place Part list: Choose or search for a part:

The screenshot displays the Altium Designer interface. The main workspace shows a blank PCB grid with a coordinate system (A, B, C, D) and a part list table at the bottom. The Components panel on the right is open, showing a search bar and a list of components. The selected component is 2N3904, an NPN General Purpose Transistor.

Part List Table:

Design Item ID	Description	Footprint
2N3904	NPN General Purpose...	TO-92A
2N3906	PNP General Purpos...	TO-92A
ADC-8	Generic 8-Bit A/D Co...	SOT403
Antenna	Generic Antenna	PIN1
Battery	Multicell Battery	BAT-2
Bell	Electrical Bell	PIN2
Bridge1	Full Wave Diode Bri...	D-38
Bridge2	Diode Bridge	D-46_6
Buzzer	Magnetic Transduce...	ABSM-1
Cap	Capacitor	RAD-0...
Cap Feed	Feed-Through Capa...	VR4
Cap Pol1	Polarized Capacitor...	RB7.6-1
Cap Pol2	Polarized Capacitor...	POLAR...
Cap Pol3	Polarized Capacitor...	C0805
Cap Semi	Capacitor (Semicon...	C1206
Cap Var	Variable or Adjustab...	C1210...
Cap2	Capacitor	CAPR5...
Circuit Breaker	Circuit Breaker	SPST-2
D Schottky	Schottky Diode	SMB
D Tunnel1	Tunnel Diode - RLC...	3.2X1.6
D Tunnel2	Tunnel Diode - Dep...	DIODE-
D Varactor	Variable Capacitanc...	SOT23
D Zener	Zener Diode	DIODE-
DAC-8	Generic 8-Bit D/A Co...	SOT402
Diac-NPN	DIAC	TO-262
Diac-PNP	DIAC	SOT89
Diode	Default Diode	SMC
Diode 10TQ035	Schottky Rectifier	TO-220
Diode 10TQ040	Schottky Rectifier	TO-220
Diode 10TQ045	Schottky Rectifier	TO-220
Diode 11DQ03	Schottky Rectifier	DO-204
Diode 18TQ045	Schottky Rectifier	TO-220
Diode 1N4001	1 Amp General Purp...	DO-41
Diode 1N4002	1 Amp General Purp...	DO-41
Diode 1N4003	1 Amp General Purp...	DO-41
Diode 1N4004	1 Amp General Purp...	DO-41
Diode 1N4005	1 Amp General Purp...	DO-41
Diode 1N4006	1 Amp General Purp...	DO-41
Diode 1N4007	1 Amp General Purp...	DO-41
Diode 1N4148	High Conductance...	DO-35
Diode 1N4149	Computer Diode	DO-35
Diode 1N4150	High Conductance...	DO-35
Diode 1N4448	High Conductance...	DO-35
Diode 1N4934	1 Amp Fast Recovery...	DO-41
Diode 1N5400	3 Amp General Purp...	DO-201
Diode 1N5401	3 Amp General Purp...	DO-201
Diode 1N5402	3 Amp General Purp...	DO-201
Diode 1N5404	3 Amp General Purp...	DO-201
Diode 1N5406	3 Amp General Purp...	DO-201

Component Details for 2N3904:

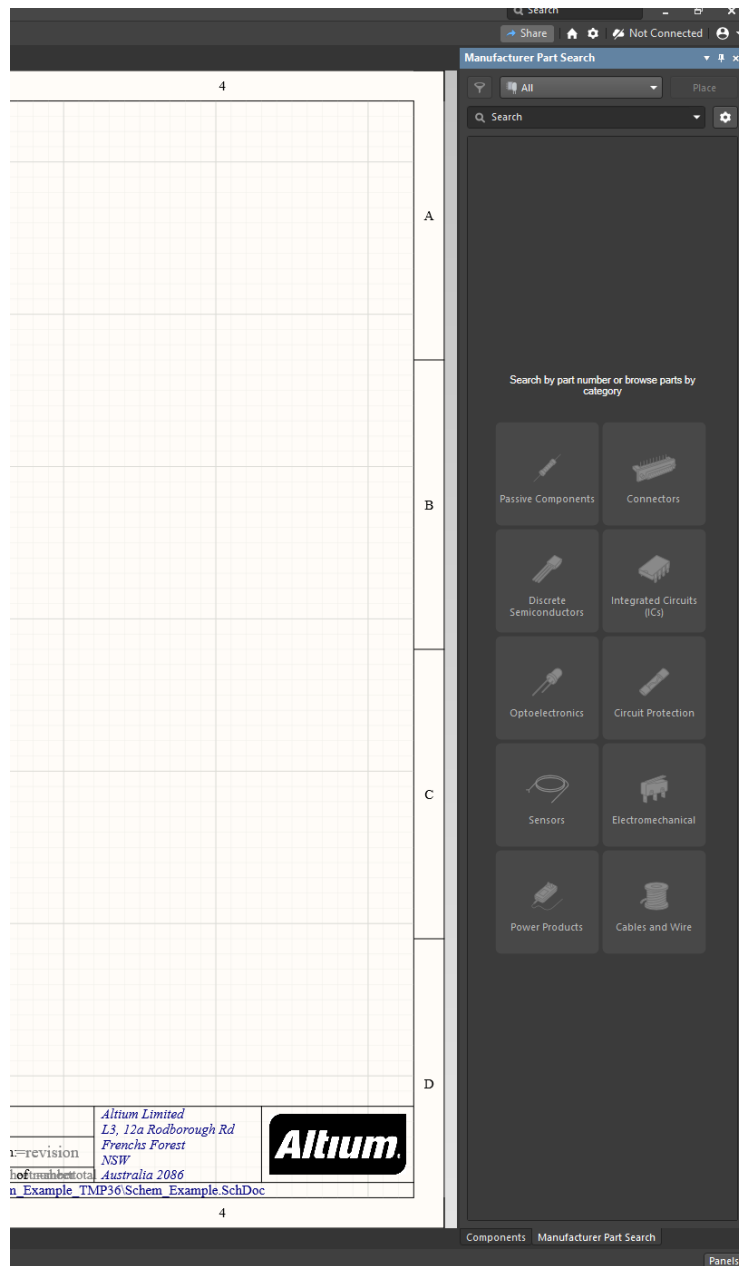
Property	Value
Description	NPN General Purpose...
LatestRevisionNote	PCB Footprint 'BCY-W...
PackageDescription	TO, Flat Index; 3 In-Lin...
PackageReference	TO-92A
PackageVersion	Sep-1998

Altium Limited
L3, 12a Rodborough Rd
Frenchs Forest
NSW
Australia 2086

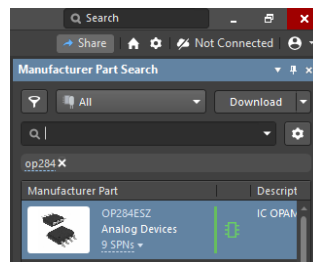
Altium

Time: 7:27:50 PM Sheet: 4 of 4
esktop\Altium_Guide\Altium_Example_TMP36\Schem_Example.SchDoc

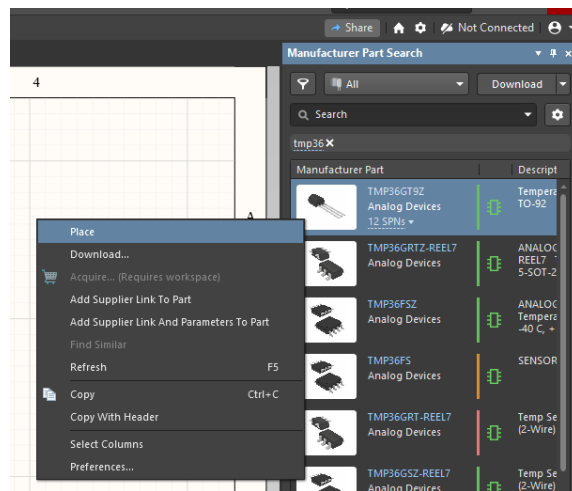
To open Manufacturer part search > click on “Panels” in the bottom right:



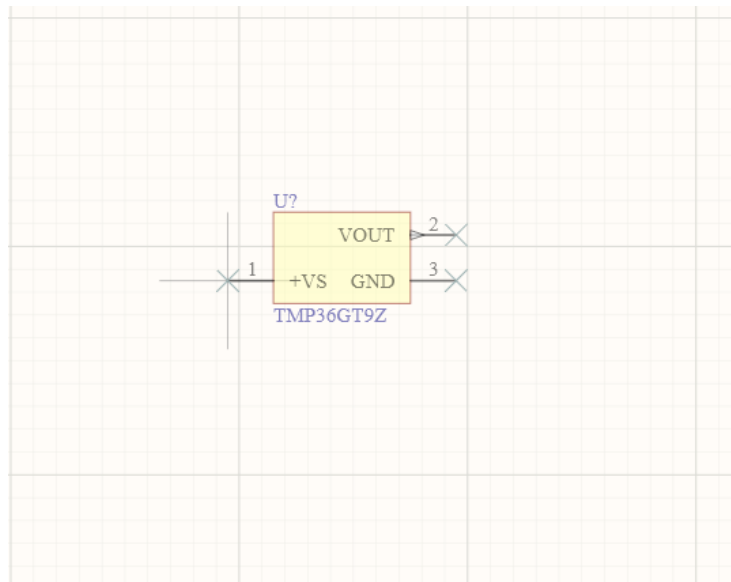
Search for a part:



Right click the part and hit “place”:

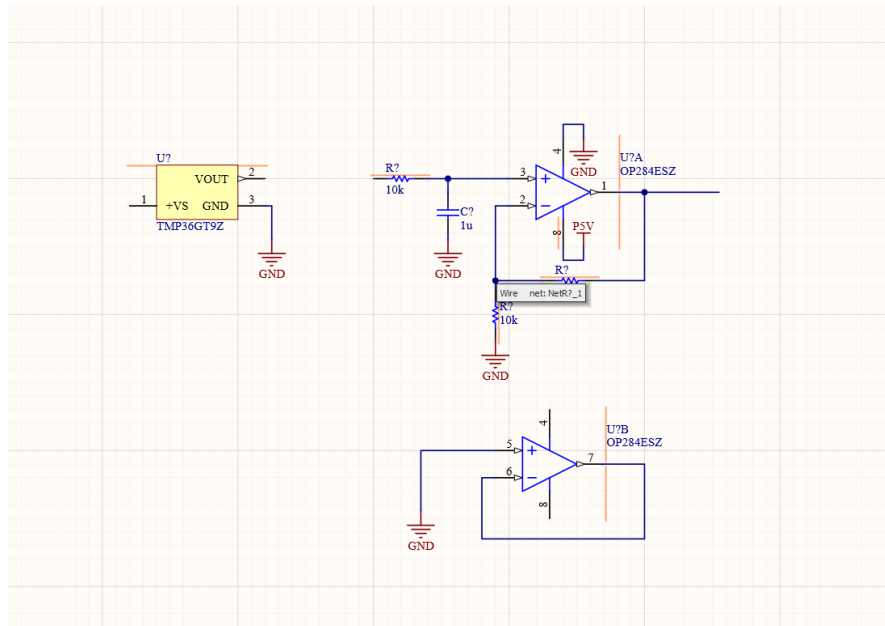


The part will now appear with your cursor in your schematic, click to place the part:

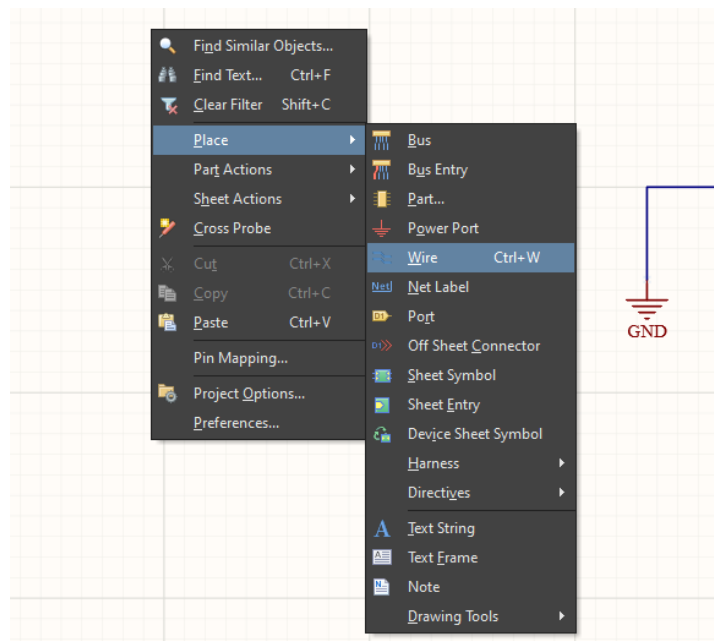


Note that the part is not named yet (U?) but we will deal with this later

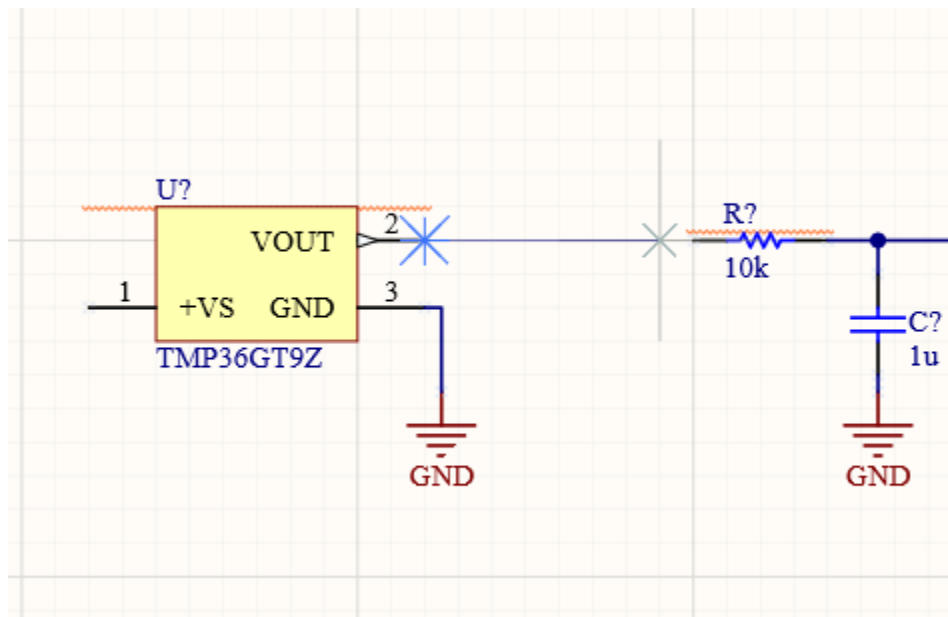
Place the other parts you will need:



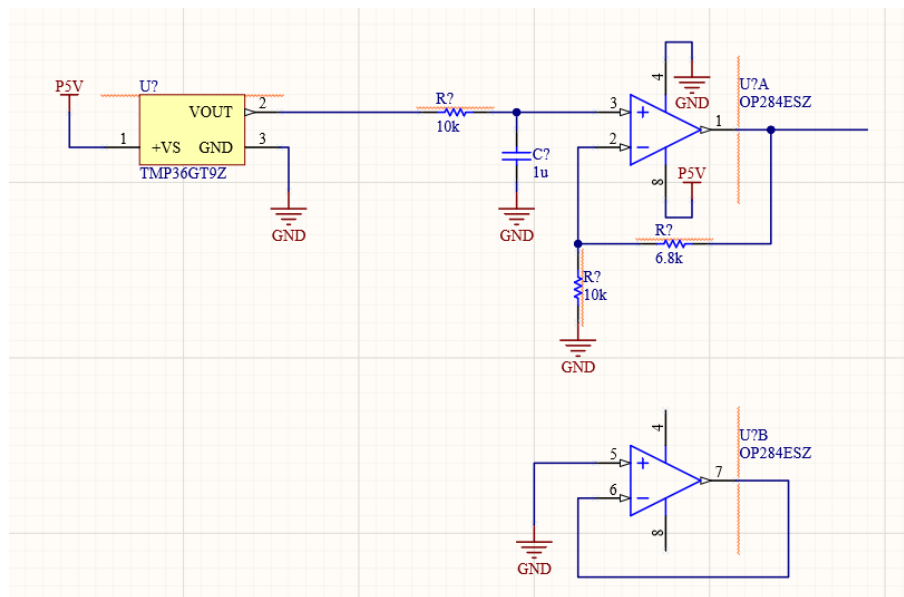
To place wire, right click > place > wire or hit CTRL + W:



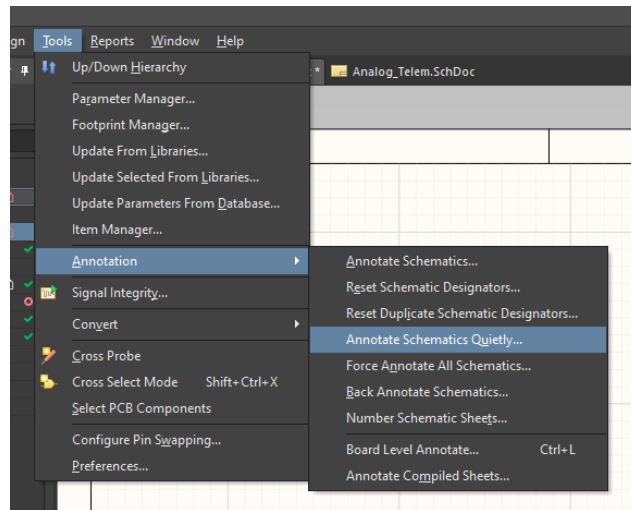
Placing the wire: Click once to “start” a wire” & then move the cursor and click again to “finish” the wire



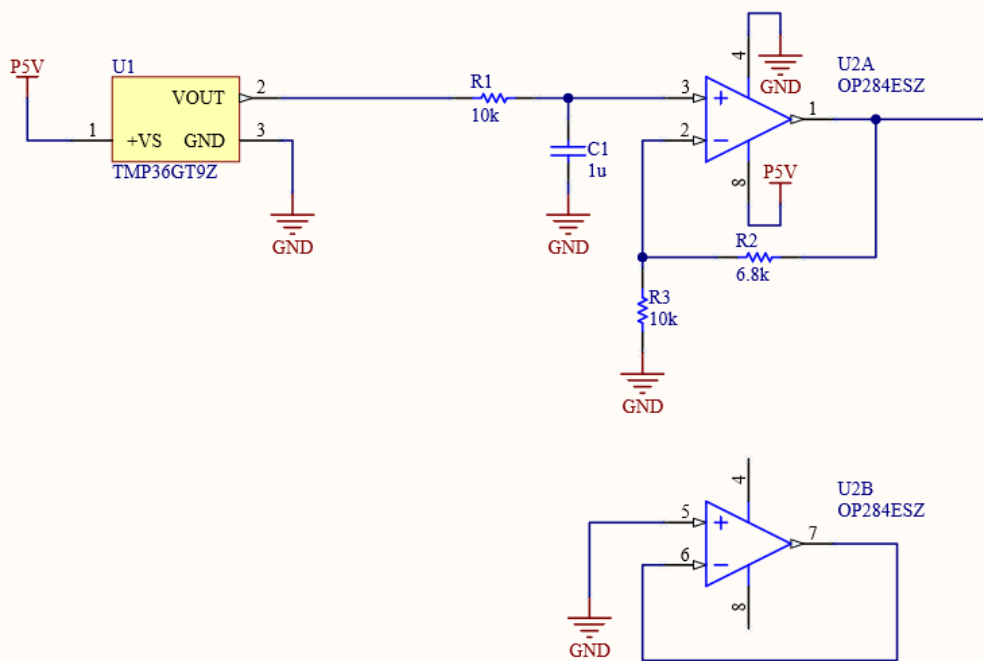
Finished Schematic:



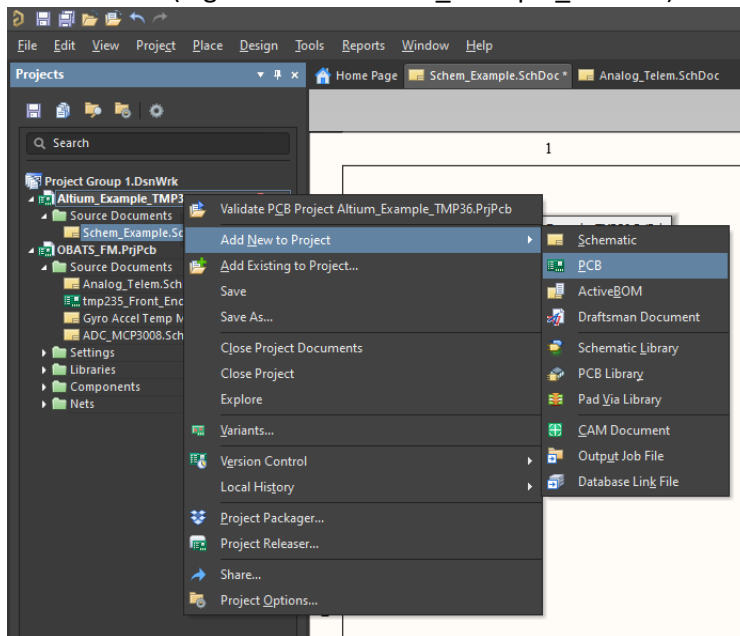
Naming parts on the schematic: (Tools > Annotation > Annotate Schematics Quietly)



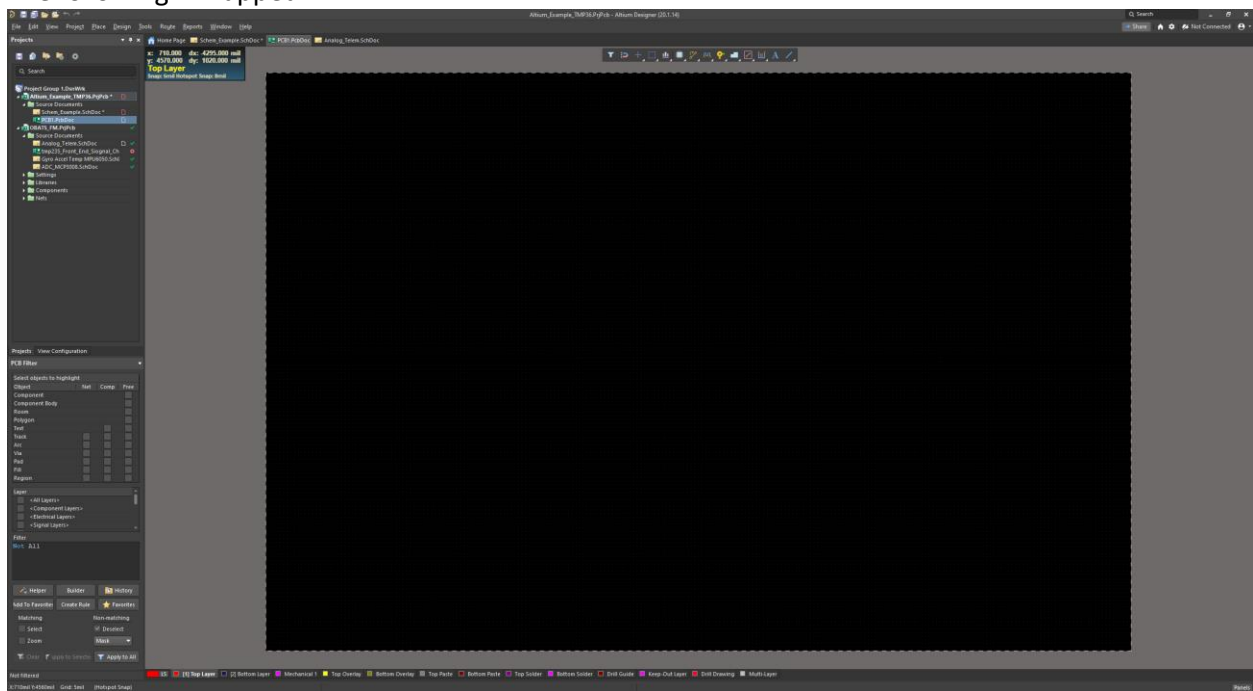
Complete Schematic:



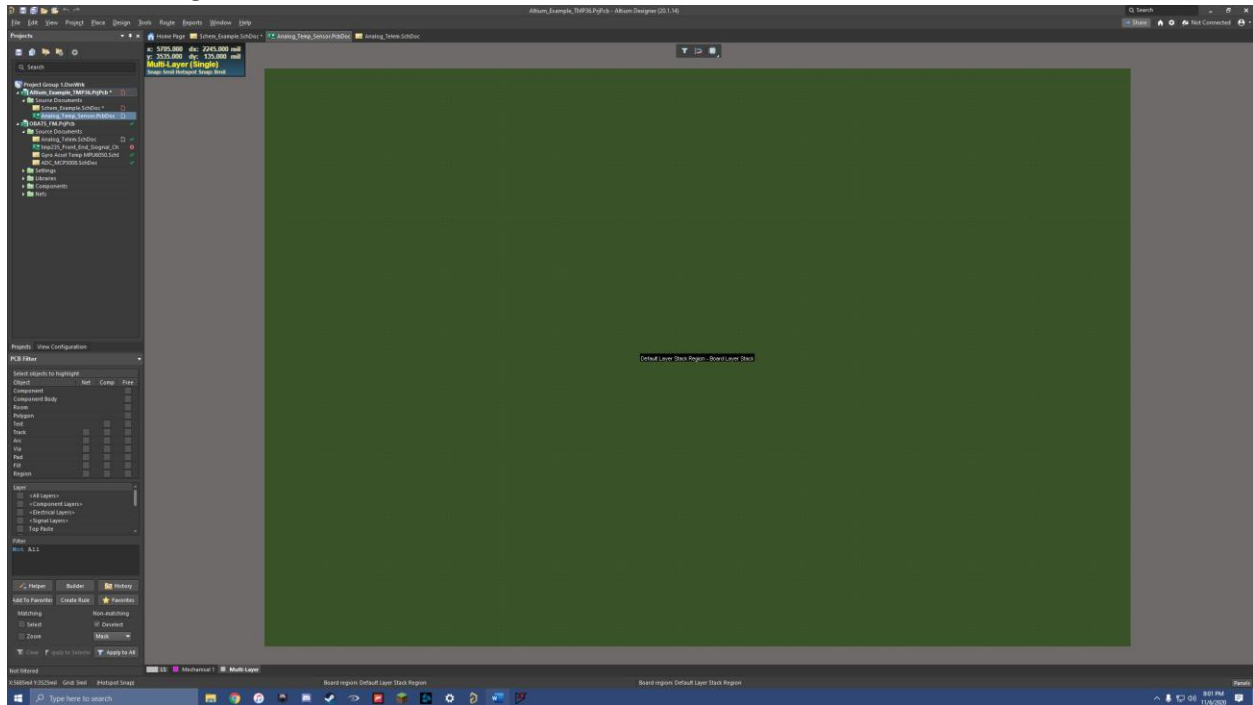
3. Add PCB file: (Right click on “Altium_Example_TMP36”)



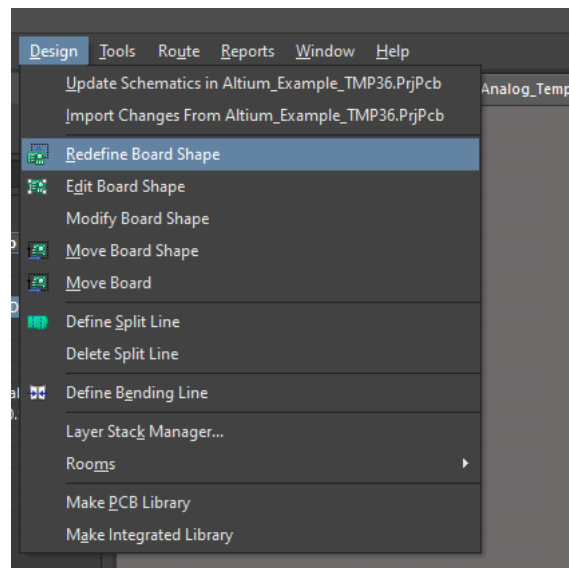
The following will appear:



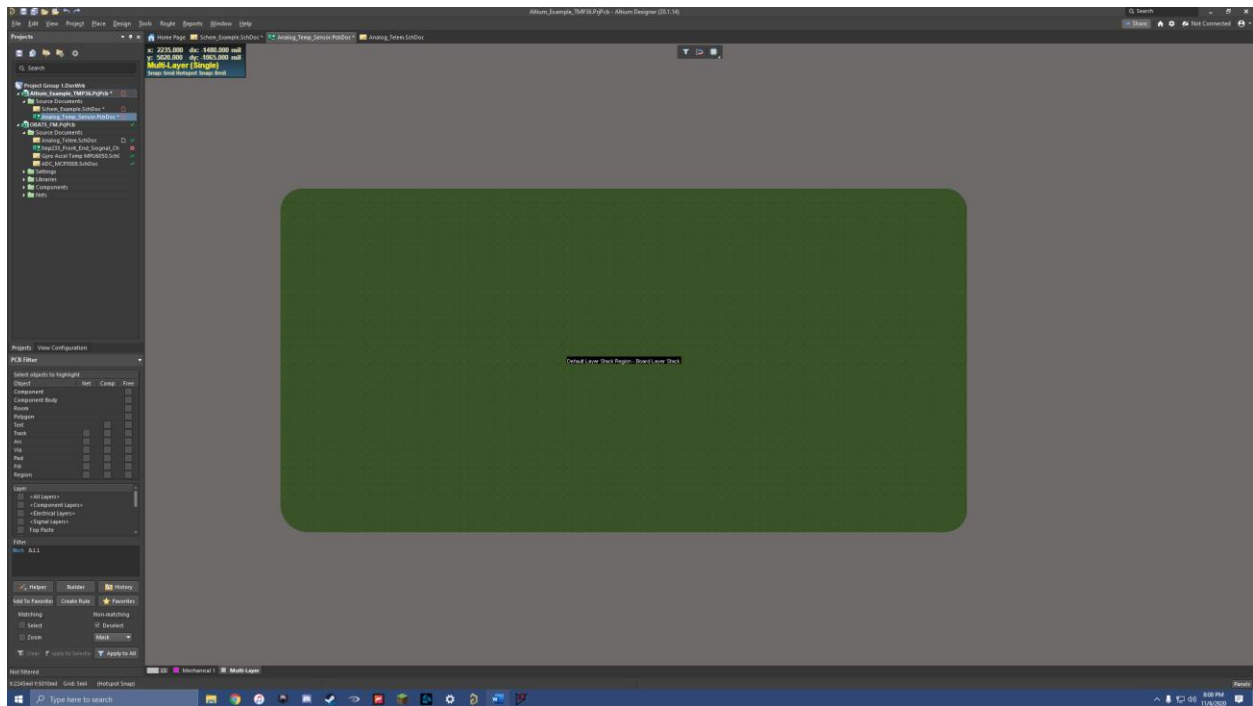
Press 1 to change view:



Hit Design > Redefine board shape

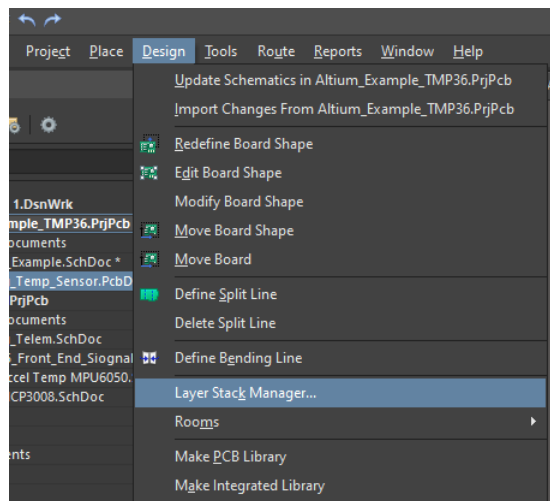


Finished Board:

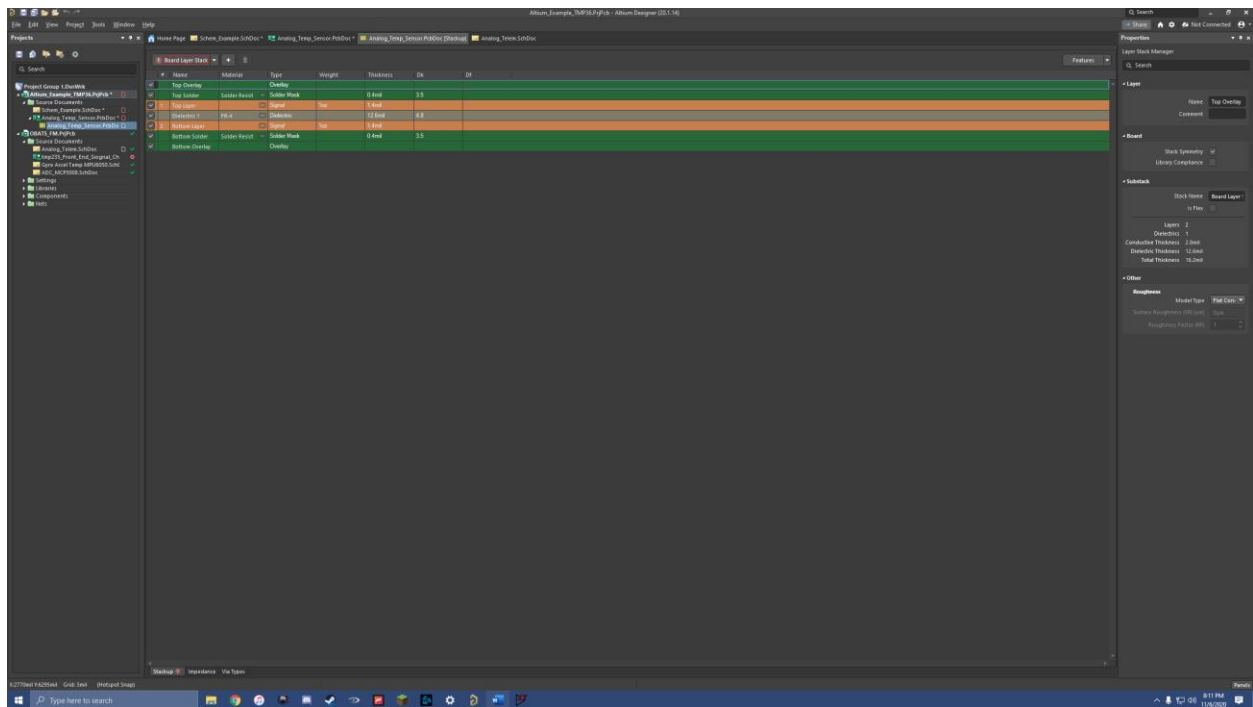


Add Layers to board: (we will do 2 layers for simplicity):

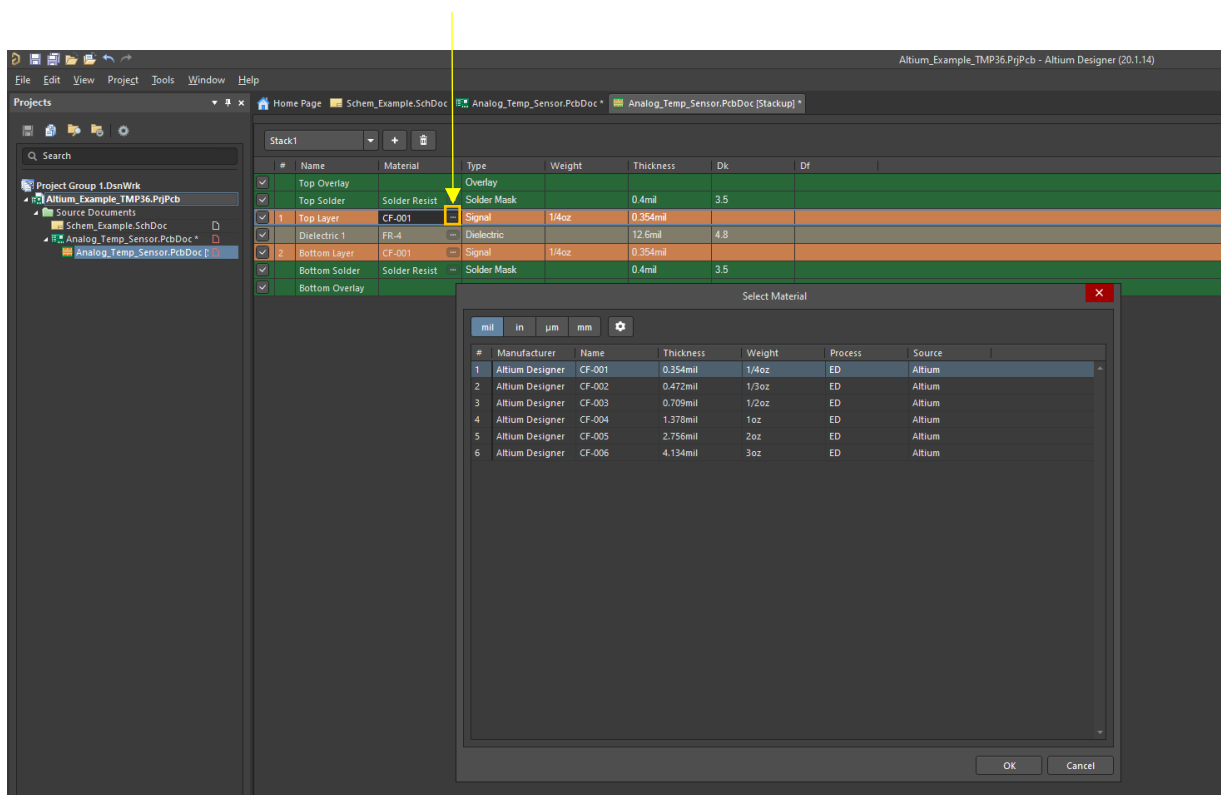
Hit Design > Layer Stack Manager:



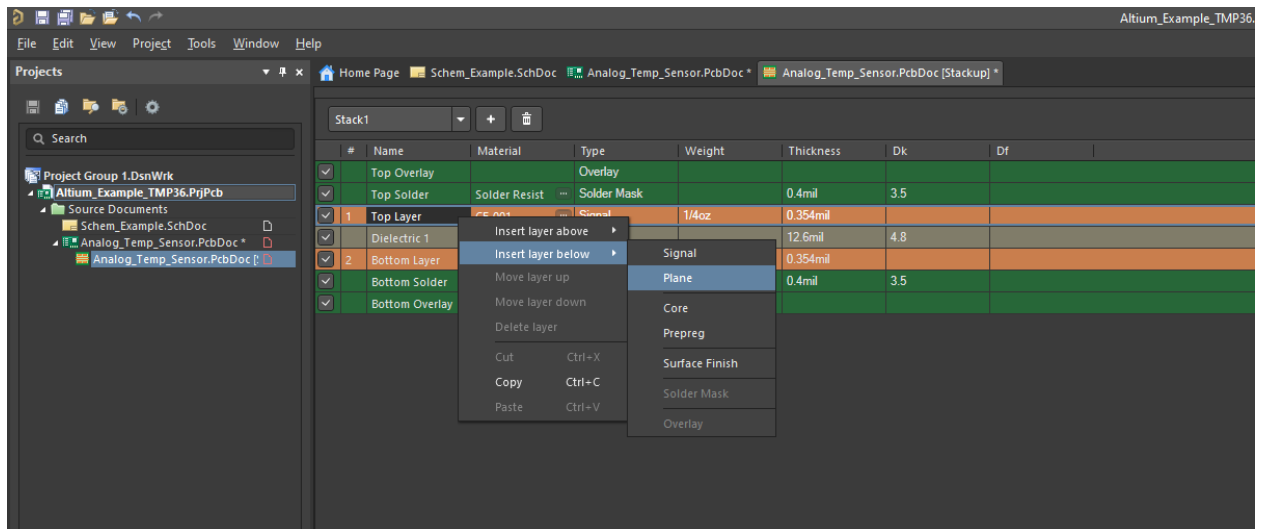
Following page will open:



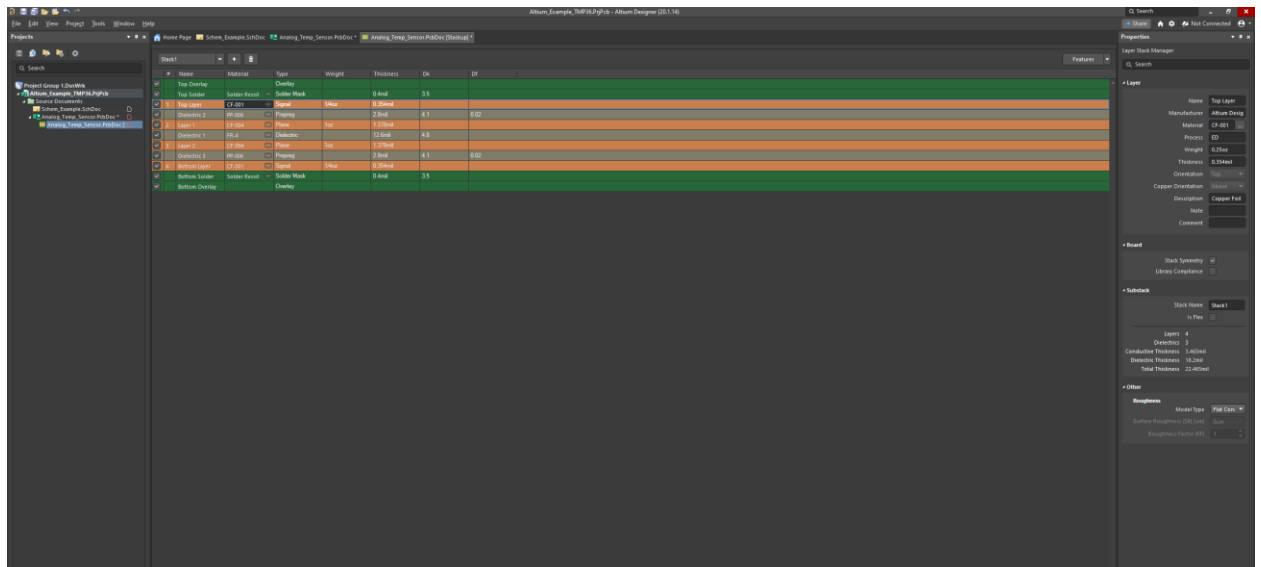
Select Material dimensions for insulators signal layers:



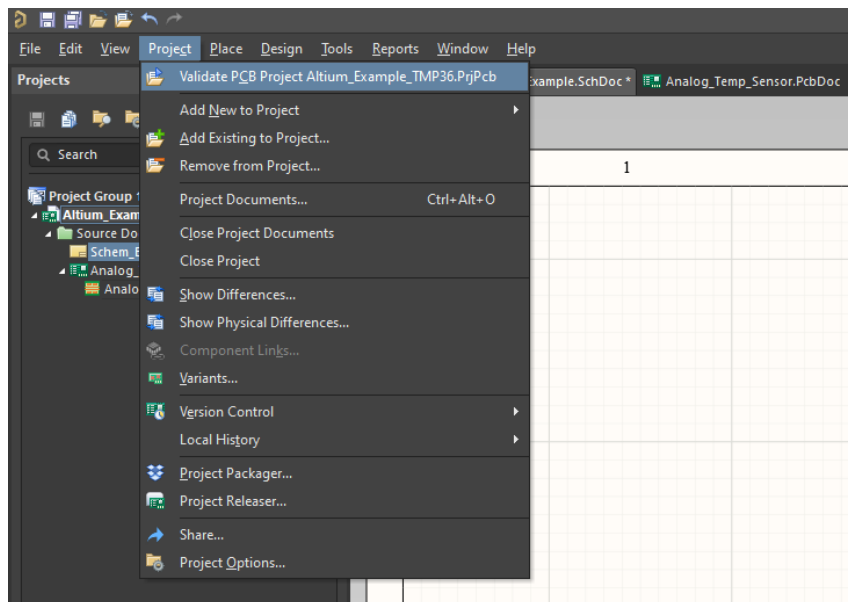
Add a plane layer: (Right click anywhere on layer “1”)



New Layer Stack:

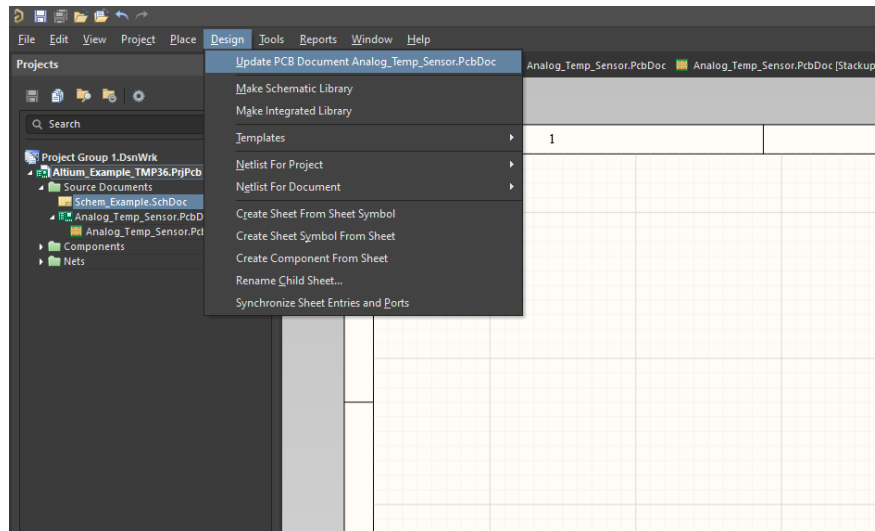


4. Run (Validate) DRC on Schematic (Design Rule Check):

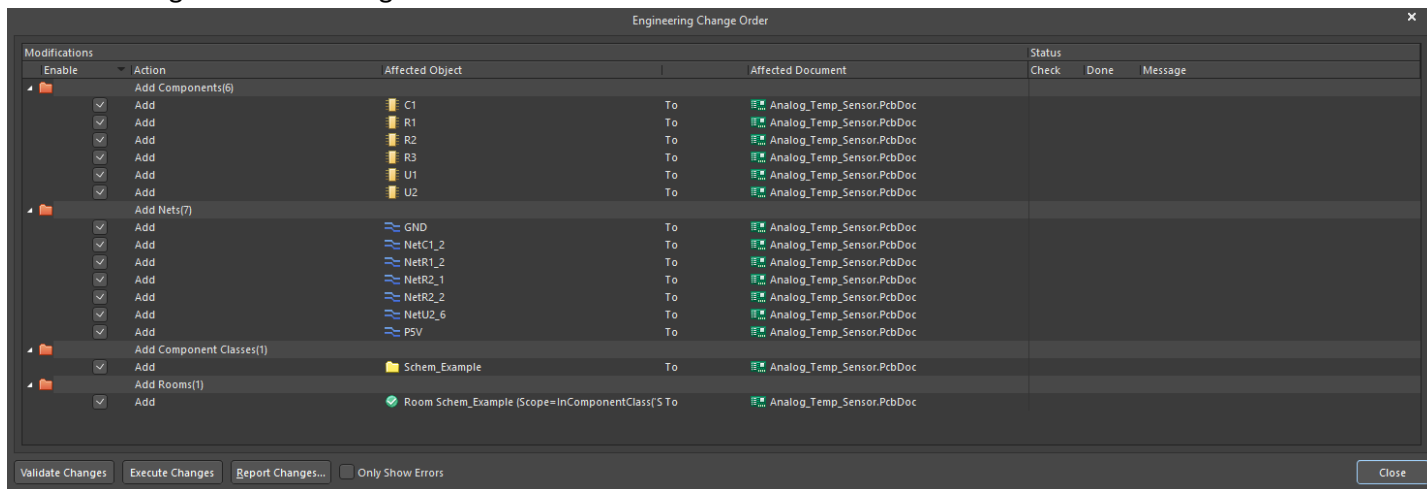


If there is nothing wrong with schematic, nothing will happen

5. Update PCB Document with Schematic components:



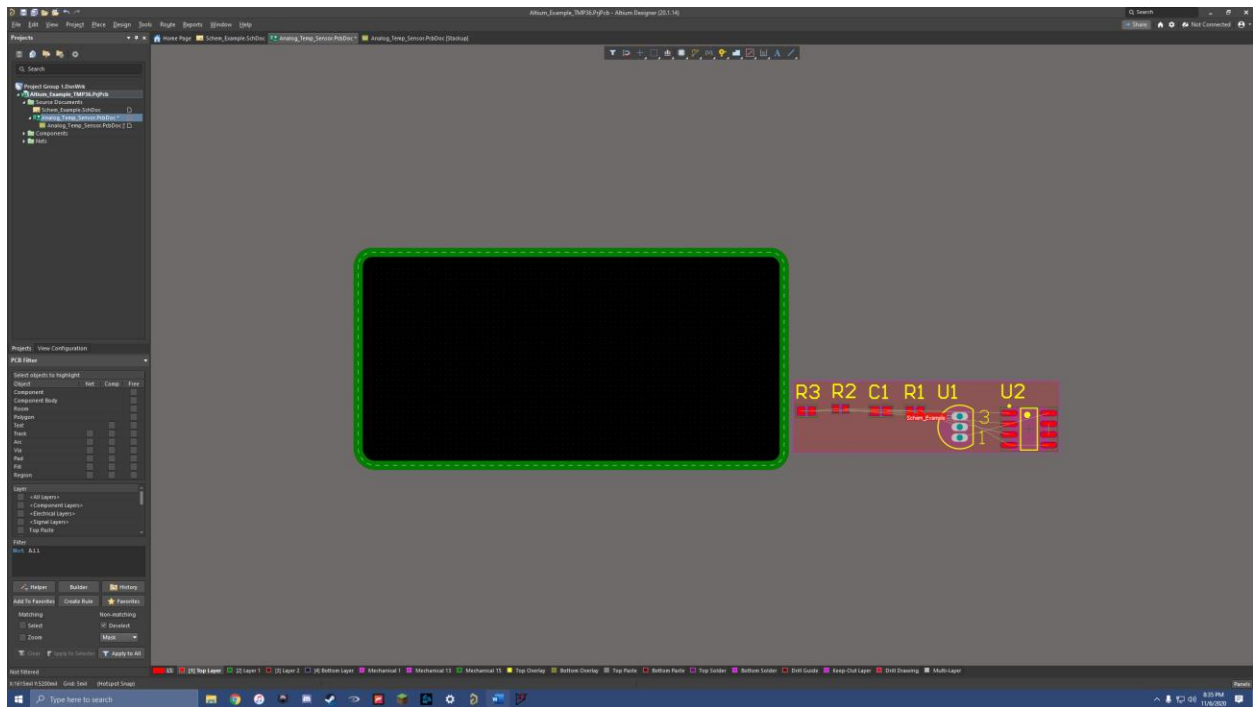
Validate changes before moving to PCB file:



Hit “Validate Changes” and then “Execute Changes”

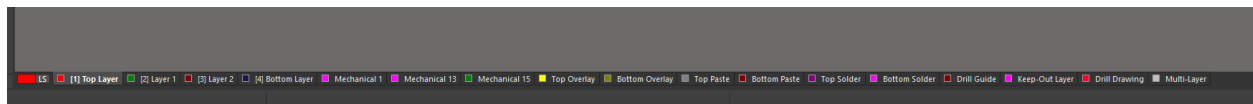
This should open the PCB document with the schematic components

Hit the “2” key to switch to 2D mode:



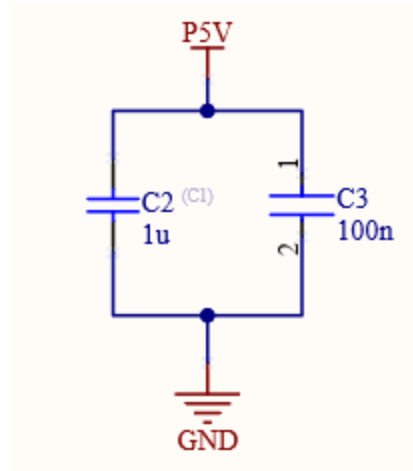
Before we start placing:

The bottom of the screen can be used to select layers of the board:



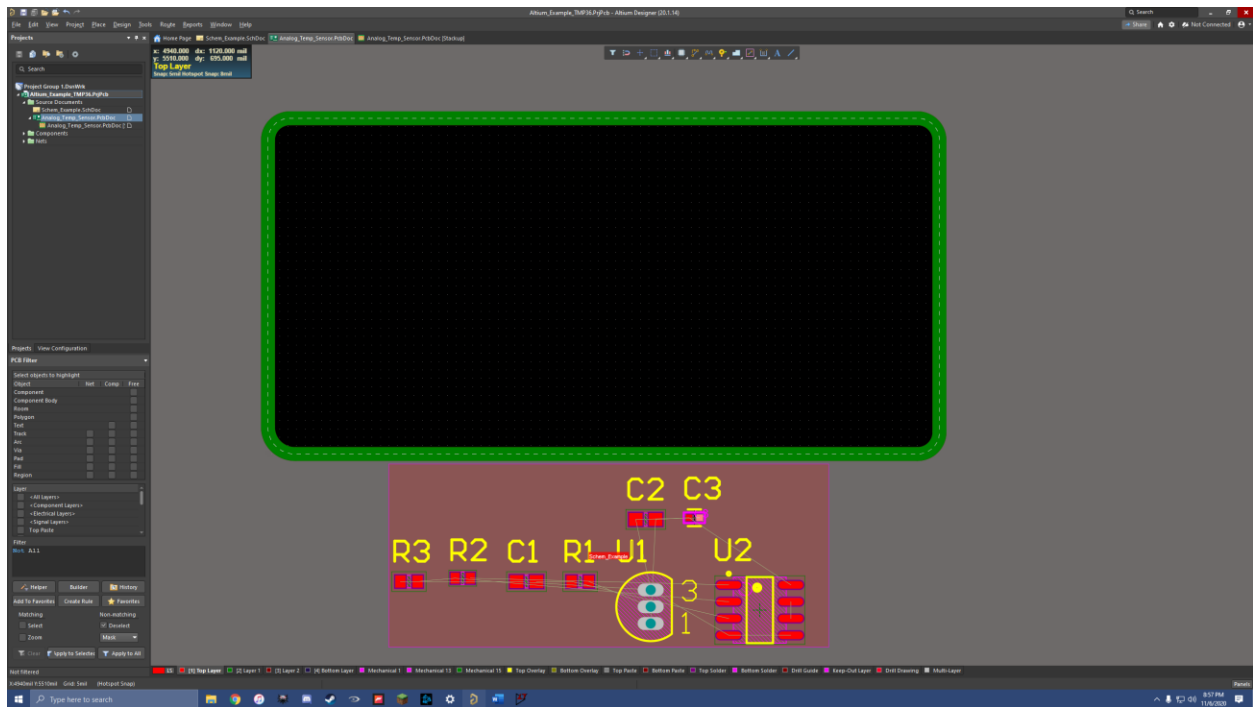
***While making this portion of the guide, I realized that I forgot to connect power and GND to capacitors and a solder pad

Schematic Parts Added:

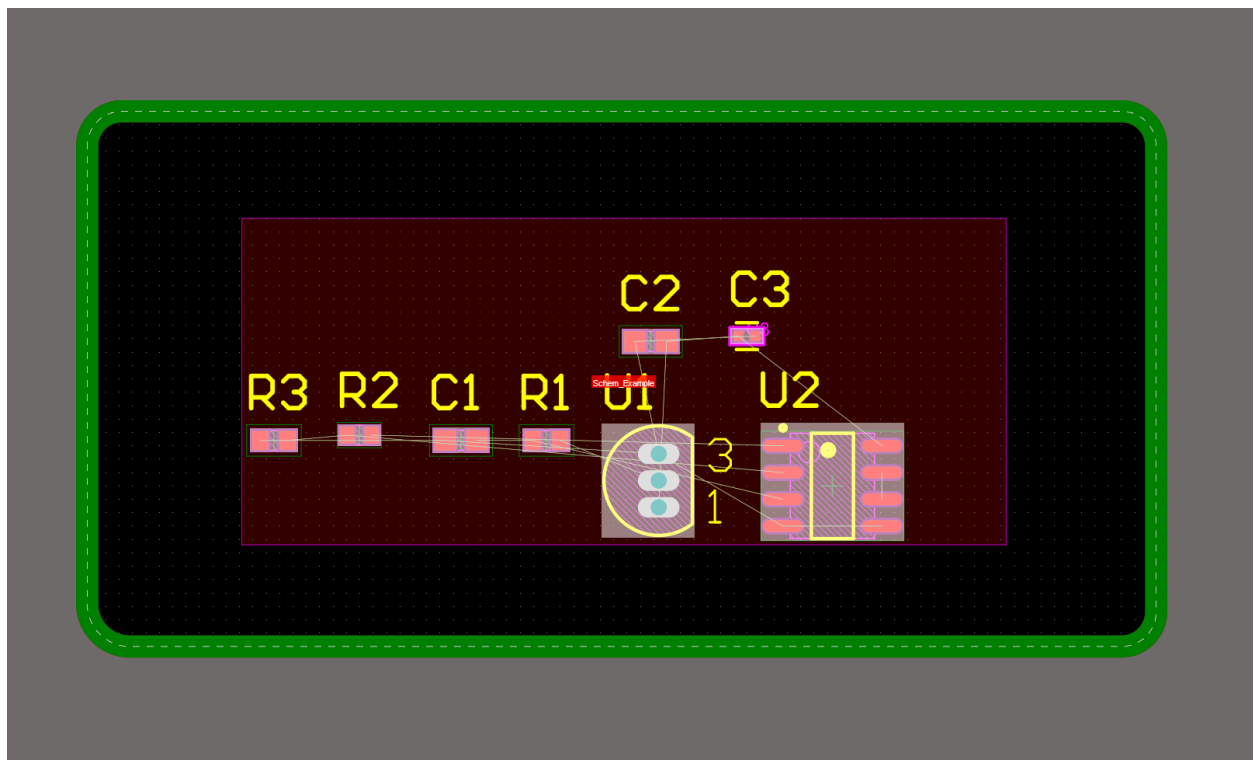


***Make sure to update changes to PCB after editing schematic file

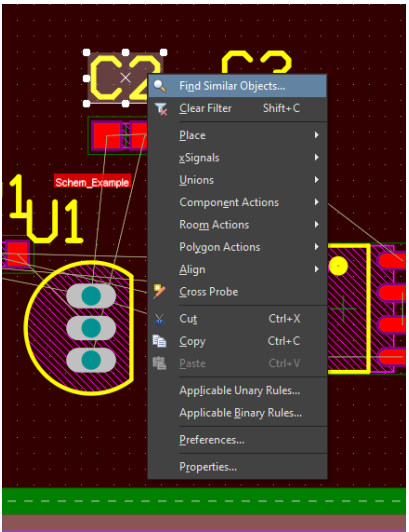
6. Place Parts Onto PCB



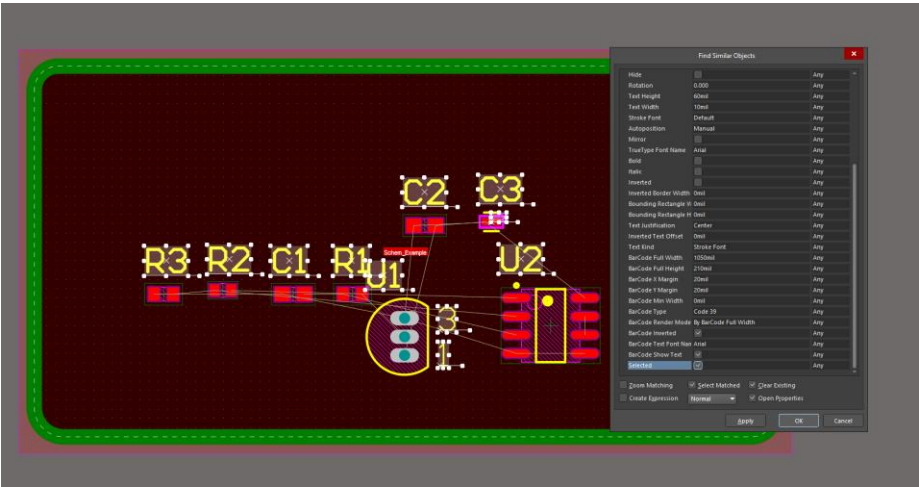
Move Keep out layer along with parts onto board:



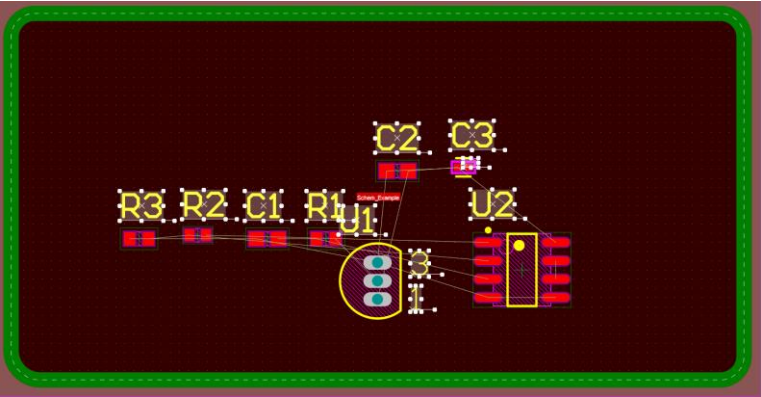
Before moving parts, lets make these giant designers smaller:



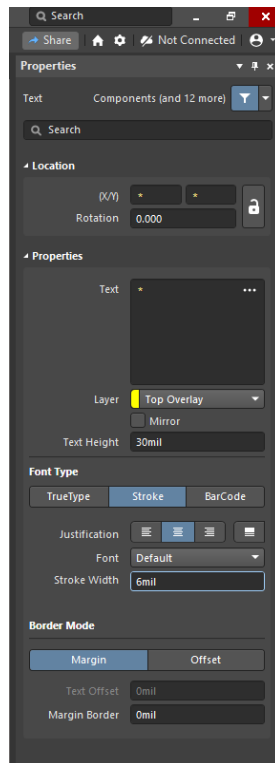
Hit “okay”



Now all designators are selected:



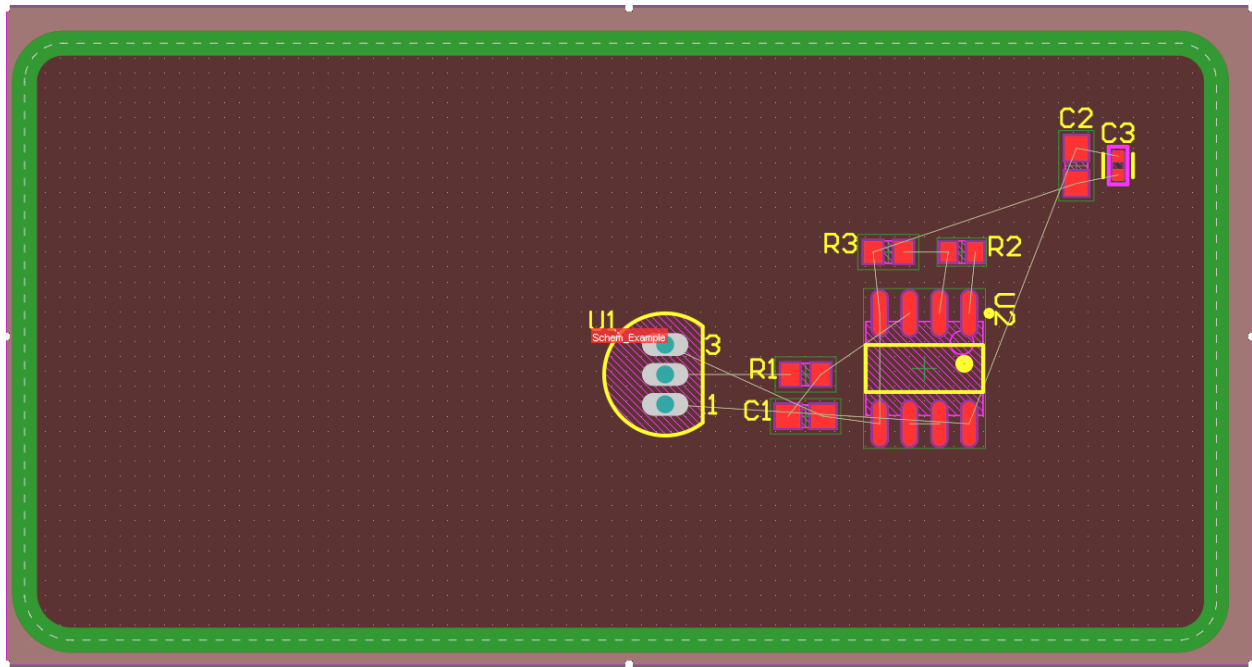
Look towards the right side of the screen, change “Text Height” & “Stroke Width”:



(I chose 30mil height and 6 mil width)

Now Designators are an appropriate size

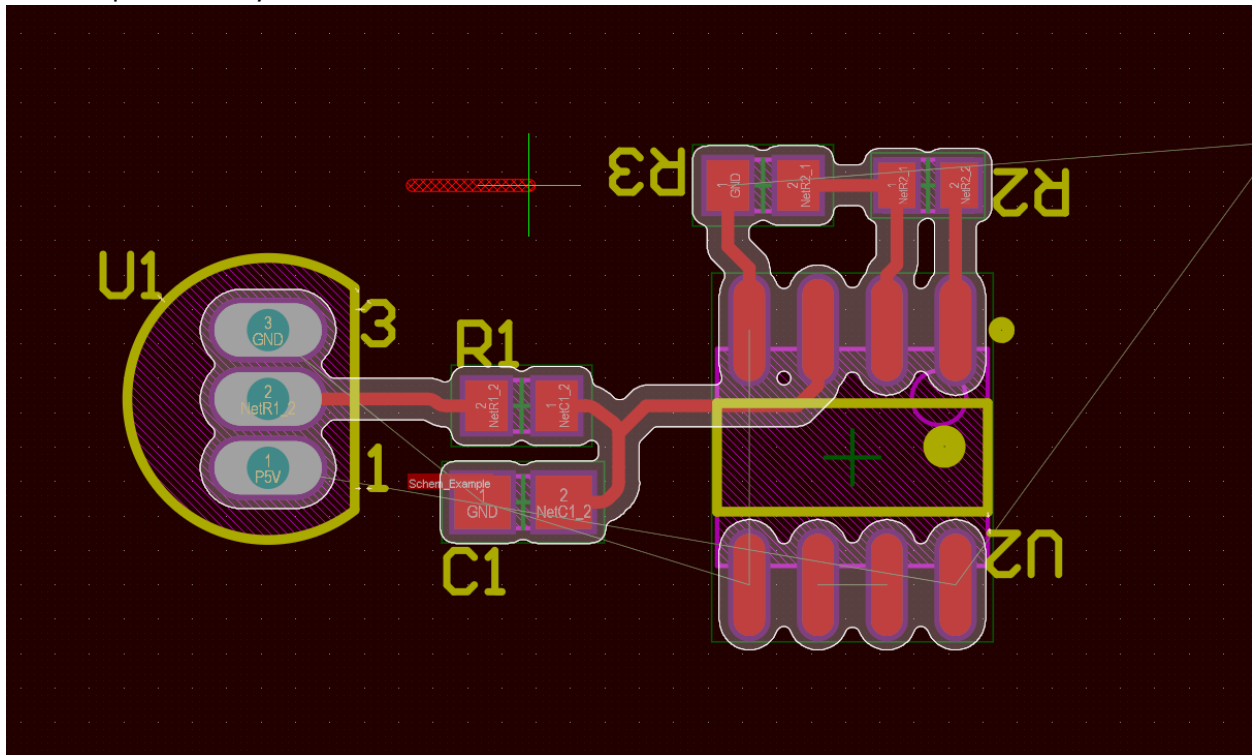
Arrange Parts onto PCB that allows for connections (disregard power pins for now):



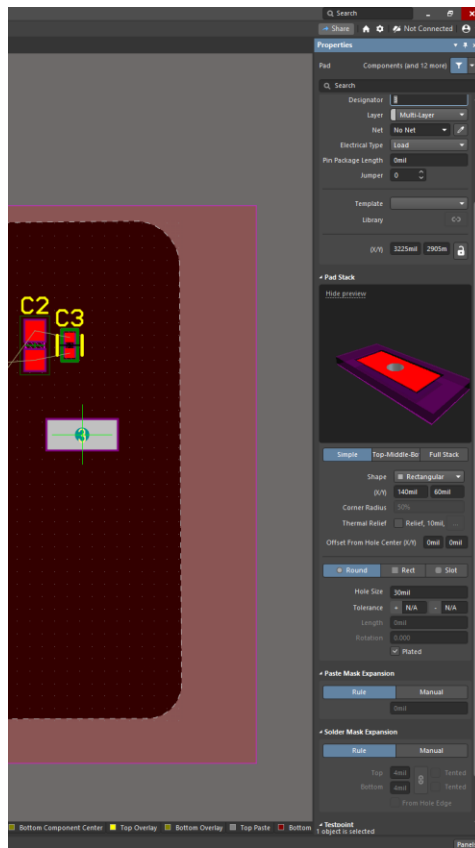
Note that lines appear to show you “net” connections

Place Tracks using “p” and then “t” keys:

Connect parts as they should be connected:

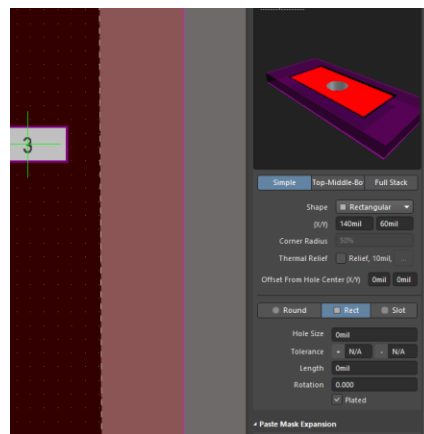


Add solder pads: hit “p” twice, and then hit TAB key:

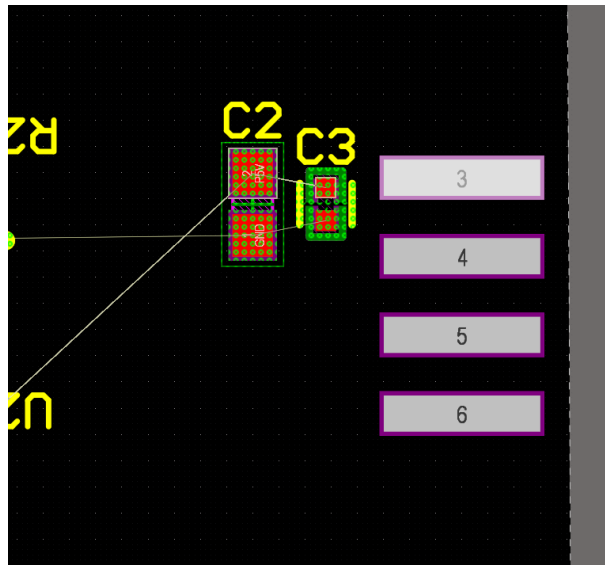


A solder pad will appear, adjust the Shape and then X/Y to make a rectangular like shape (this pad will be solder to a wire)

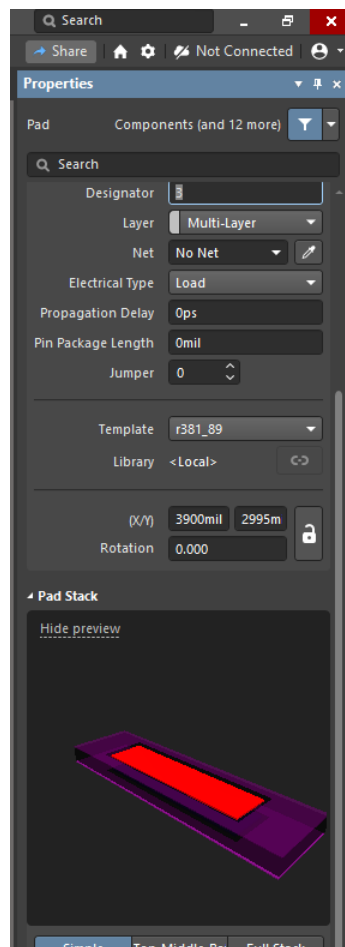
Then, Change hole size of pad to zero:



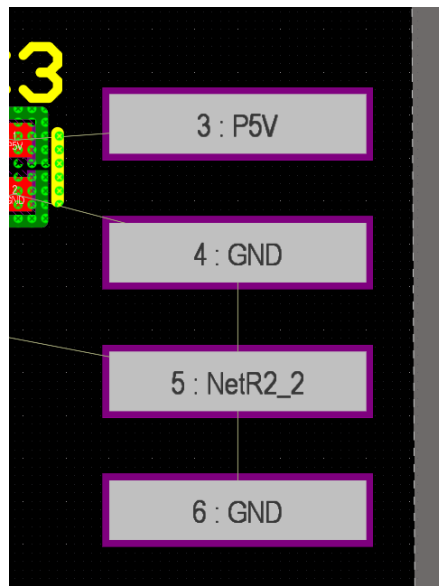
Place pads:



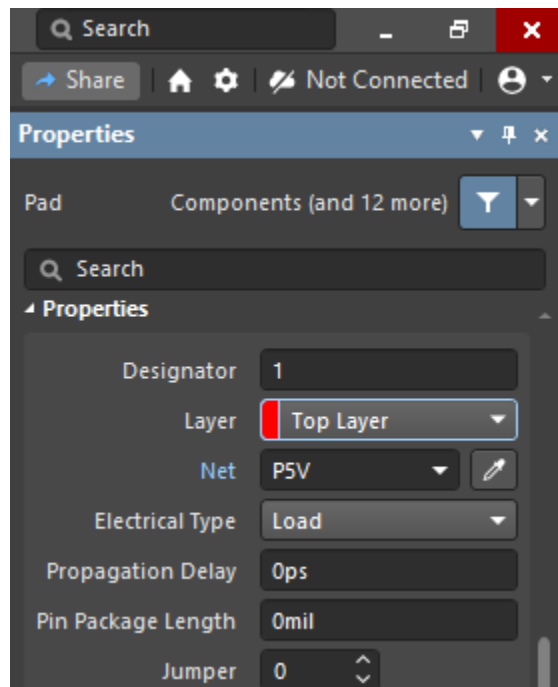
Assign nets to solder pads:



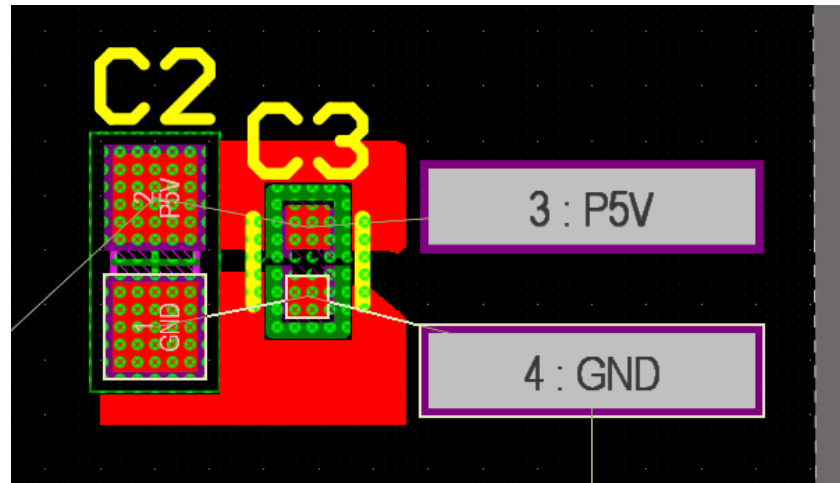
Netted Pads:



Make pads single layer:



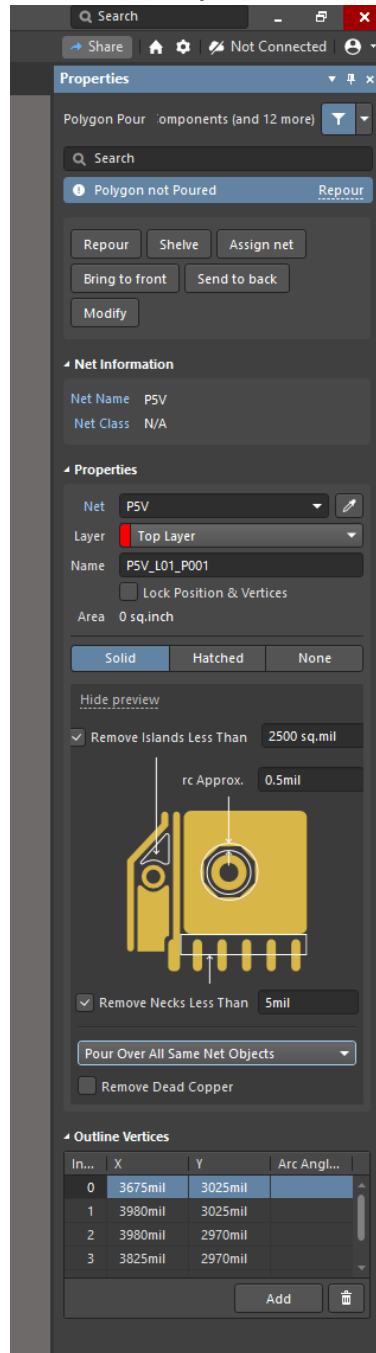
Polygon Pours type “p” and then “g” and then drag over objects: (SHIFT + SPACE to change curve)



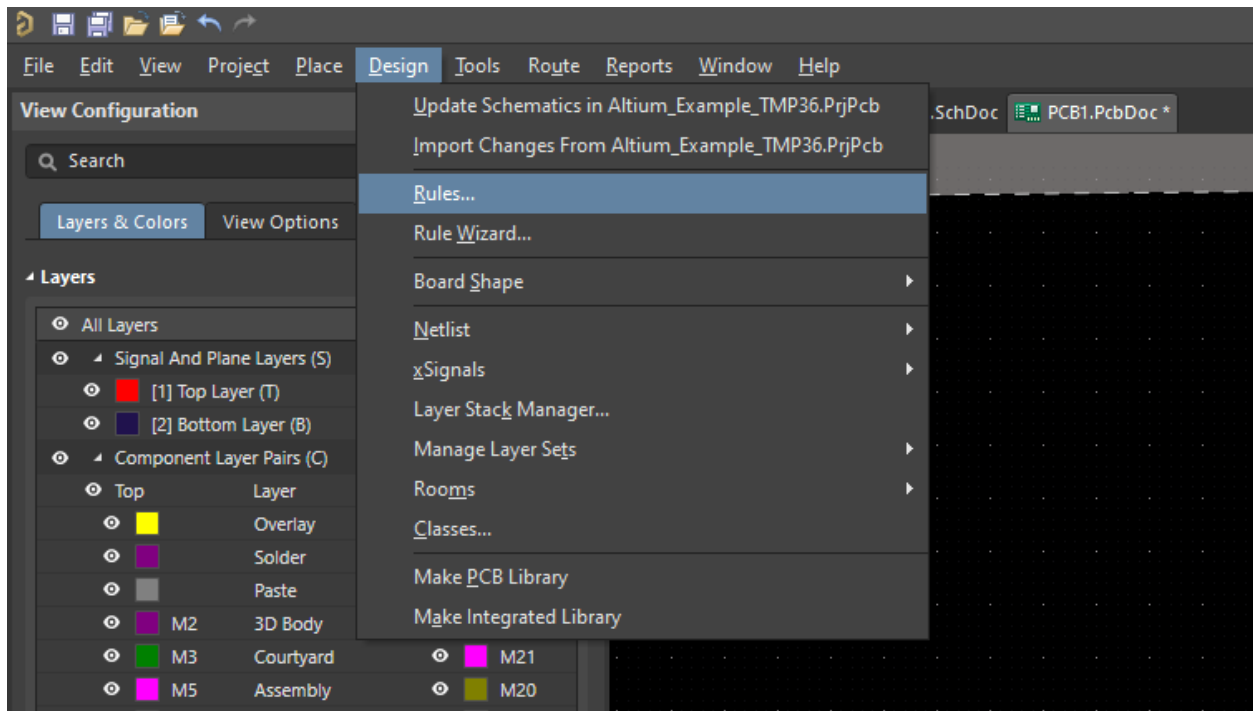
Assign poly pour nets to pads and then repour all polygon (hit “t” “g” “a”)

If pours come out weird:

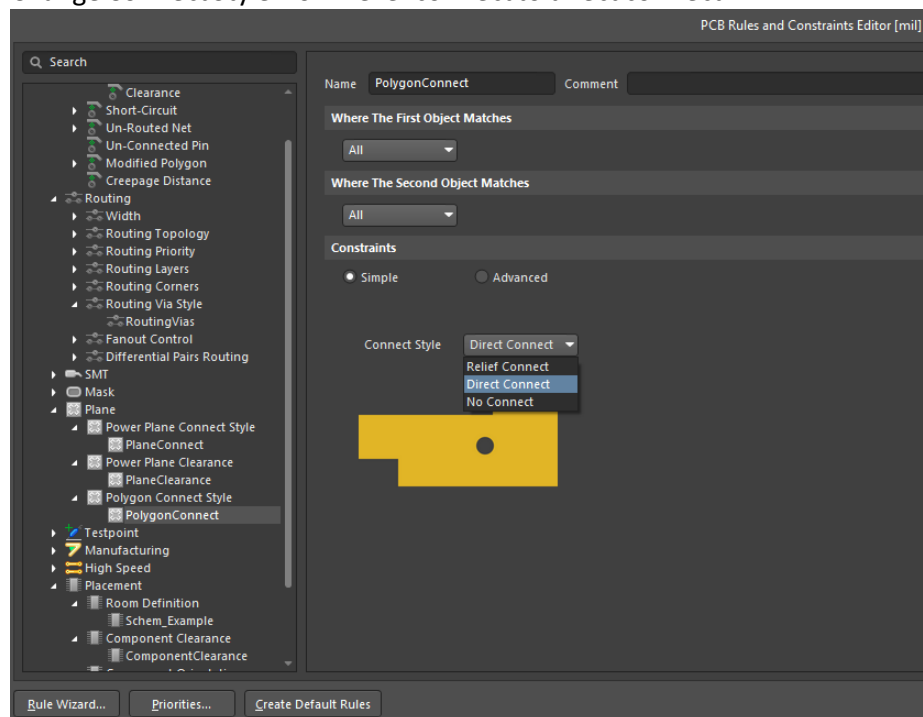
Change pour settings to “Pour Over All Same Net Objects”



If still weird, perform the following:

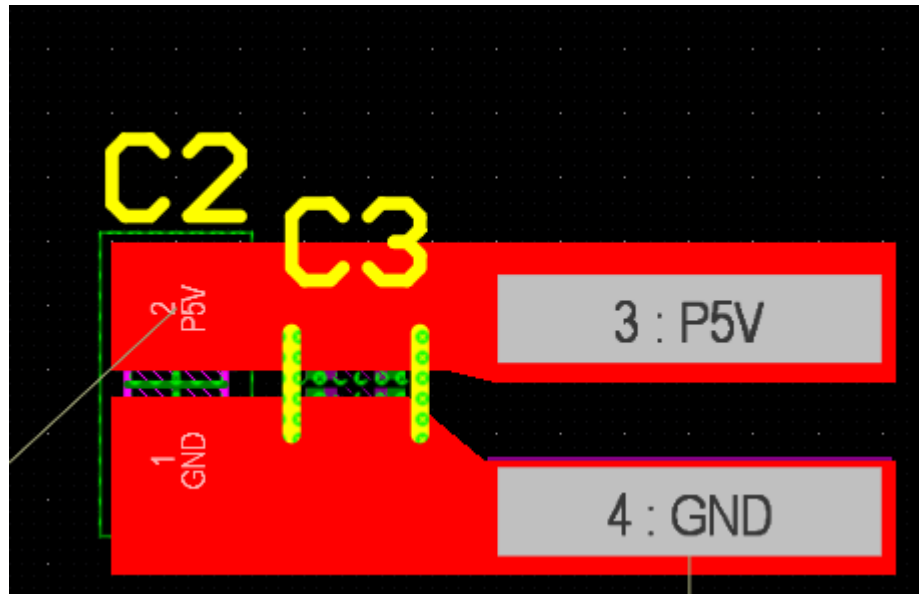


- Plane > Polygon Connect
 - Change Connect Style from Relief connect to direct connect:



Hit Apply and then Okay

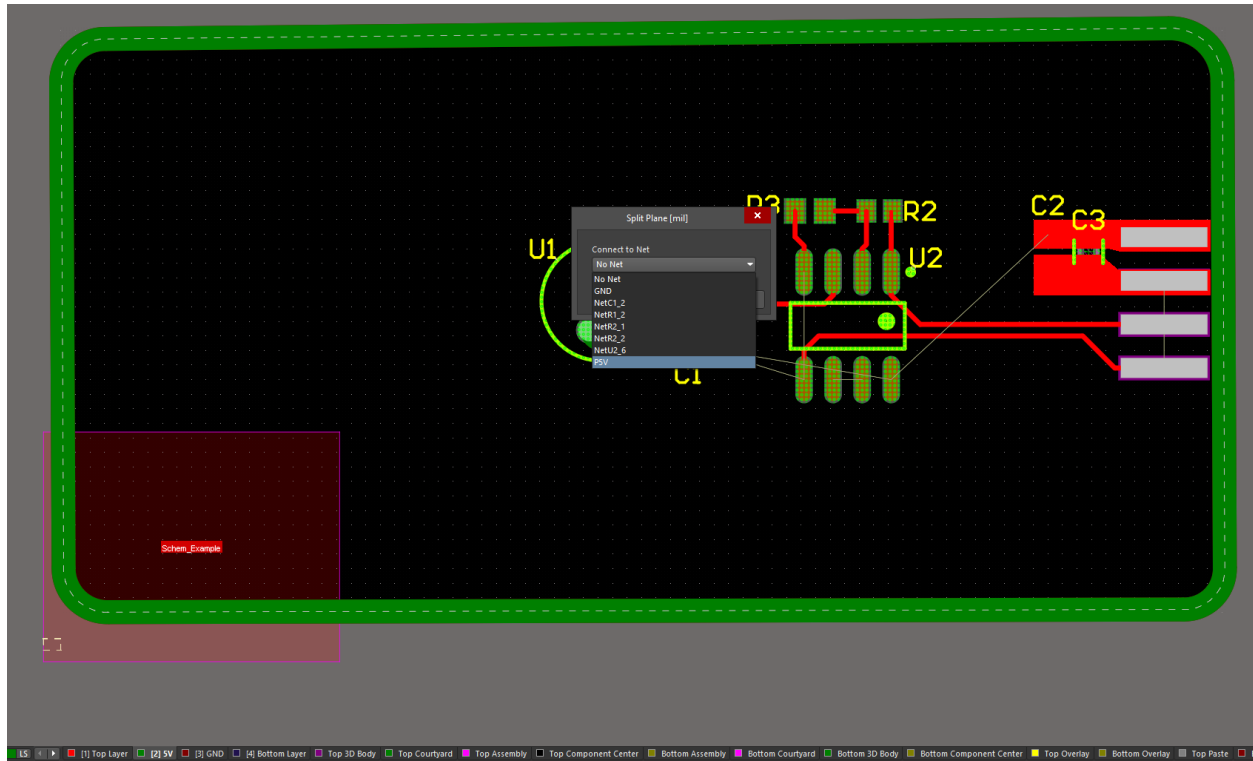
Repour using "t" "g" "a":



Now all tracks and resistors are placed (Time for Vias!!!!)

Before Vias, we must net layers of the board (we are connecting power to layers, 5V & GND)

Select the layer in the bottom that you wish to net, right click the board (anywhere with empty black space), hit Properties, and then choose the net you want:



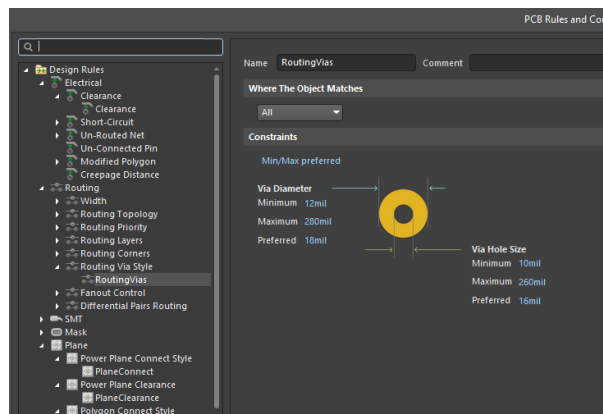
Repeat for any other layers

OSHPARK Via Requirements:

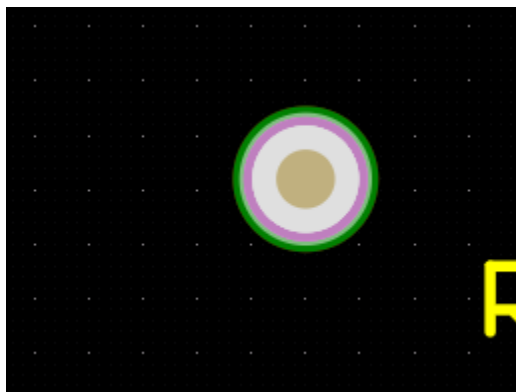
4 Layer Services

Spec Type		Note
Minimum Hole size	10 mil (0.254mm)	Holes belows this size will be rounded up
Maximum Drilled Hole Size	260 mil (6.604mm)	Holes exceeding this size will be milled
Annular Ring	4 mil (0.1016mm)	
Via Plating Thickness	1 mil (0.0254mm)	
Fabricated Hole Size Tolerance	+/-2.5mil max (0.0635mm)	+/-1 mil typical (0.0254mm)
Fabricated Hole Position Tolerance	+/-2.5mil max (0.0635mm)	+/-1 mil typical (0.0254mm)

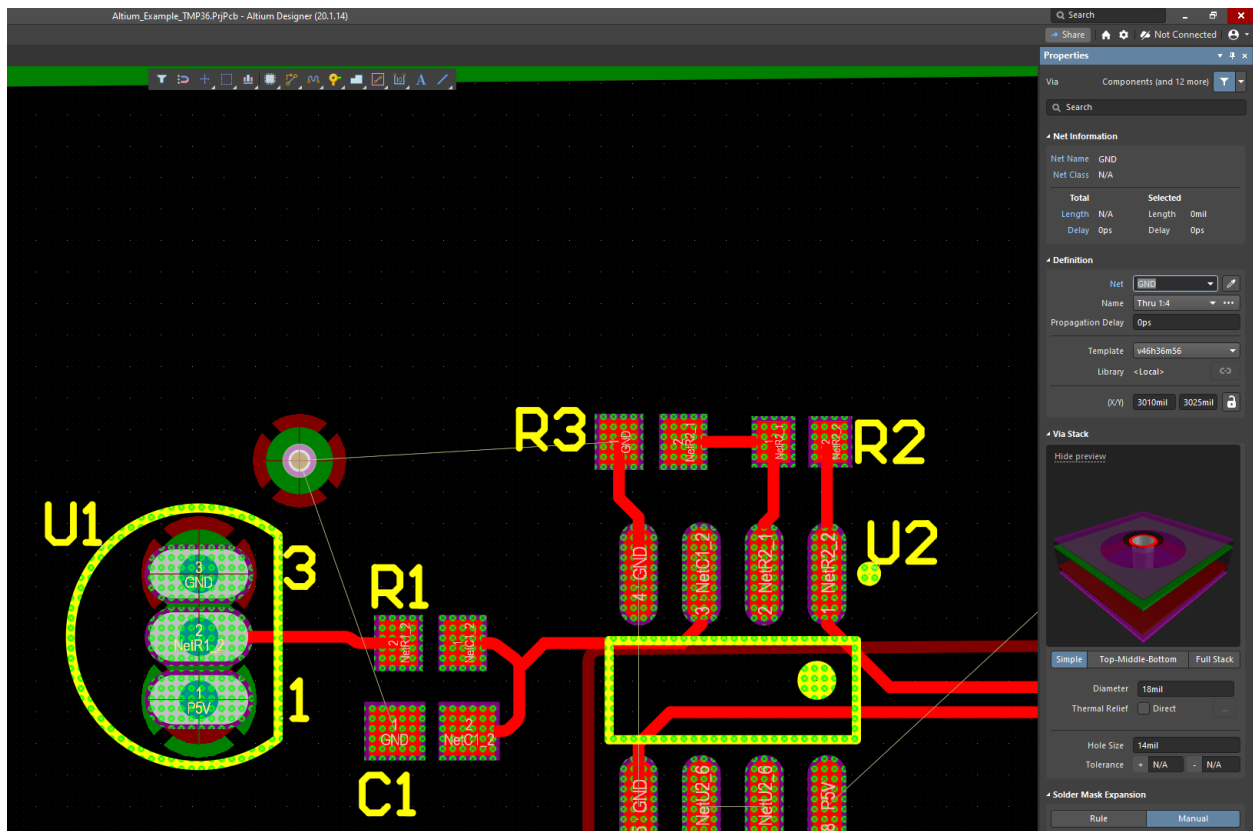
Settings in Altium (Design > Rules > Routing Via Style > Routing Vias: Adjust Settings:



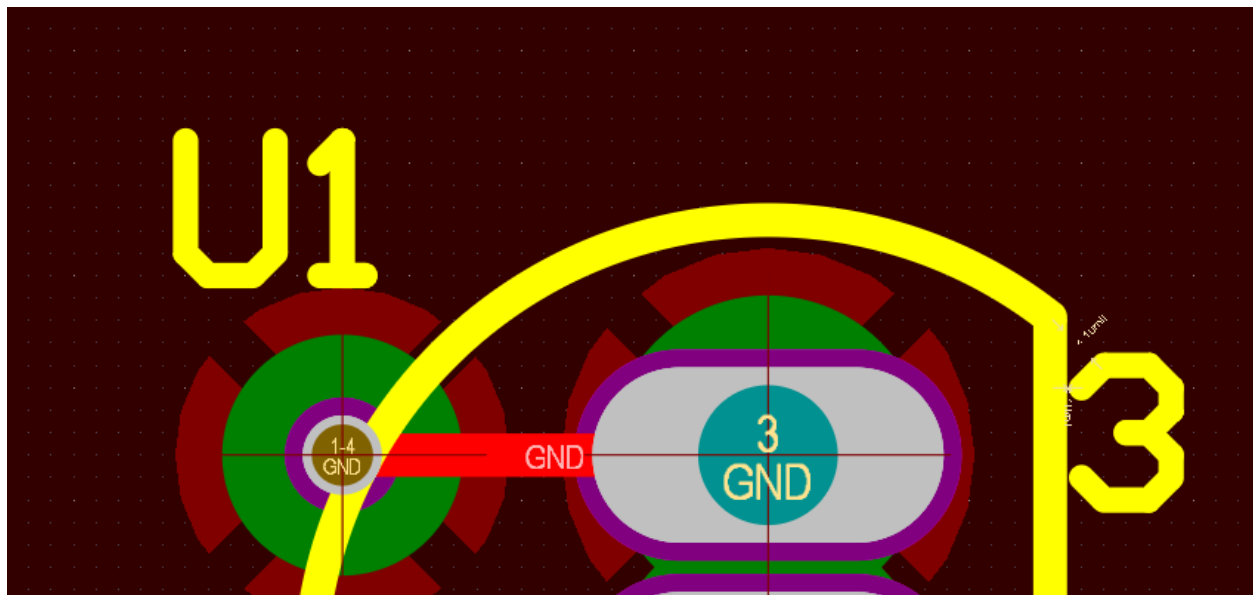
Placing Vias: Hit “p” “v” and then click:



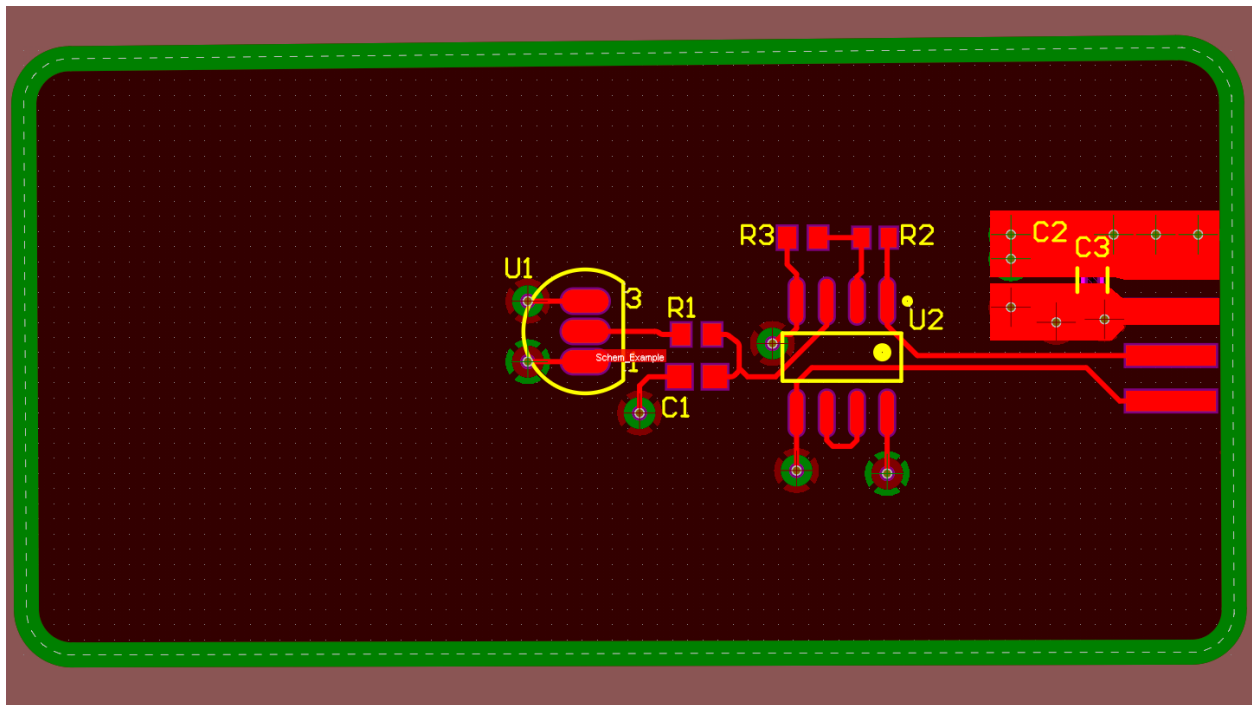
Assign net to Via:



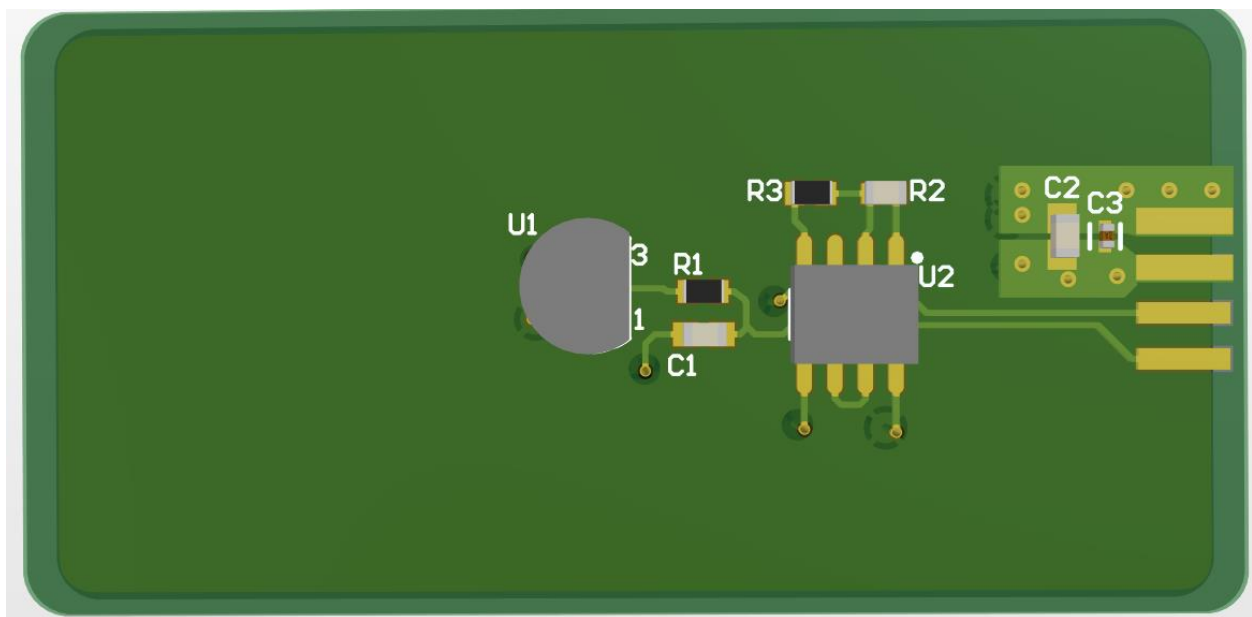
Connect to via using track:

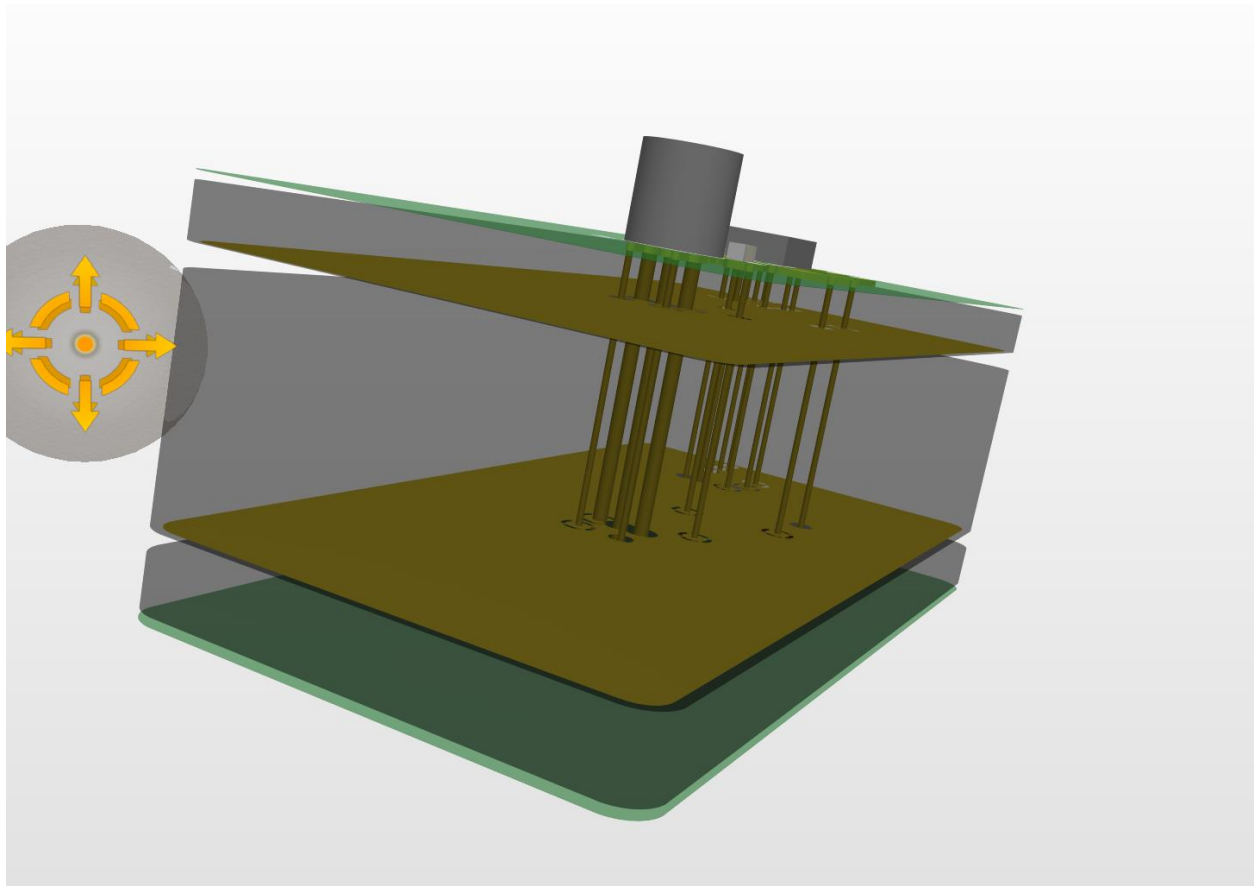


Place and connect all Vias:

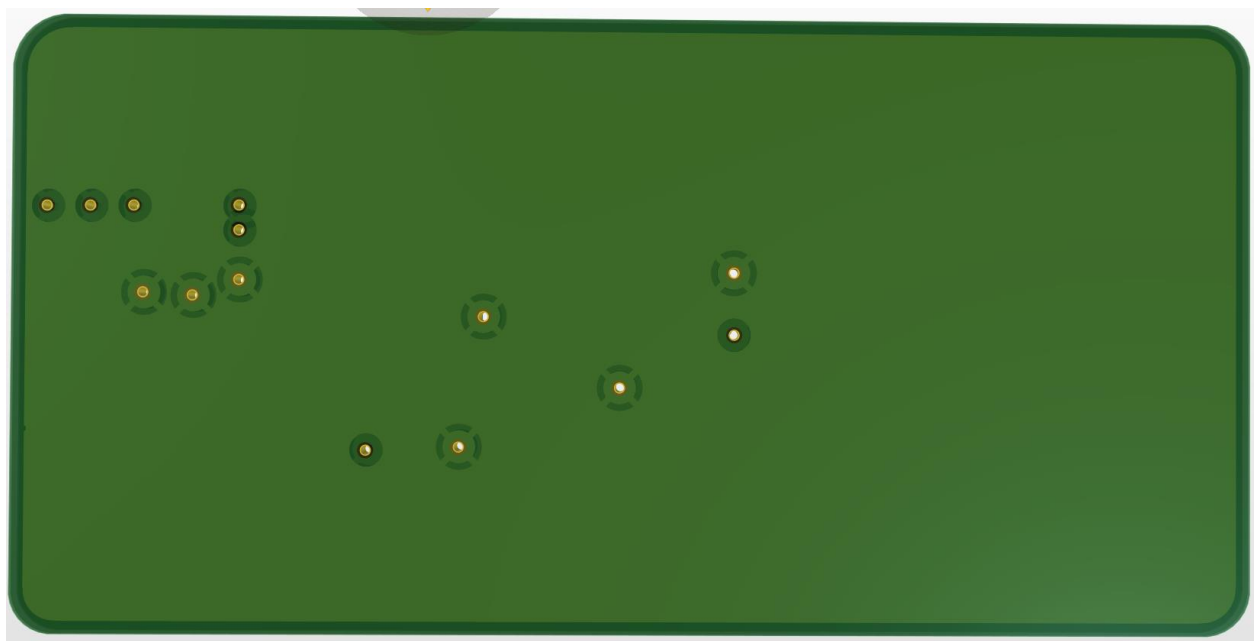


Board is Complete:

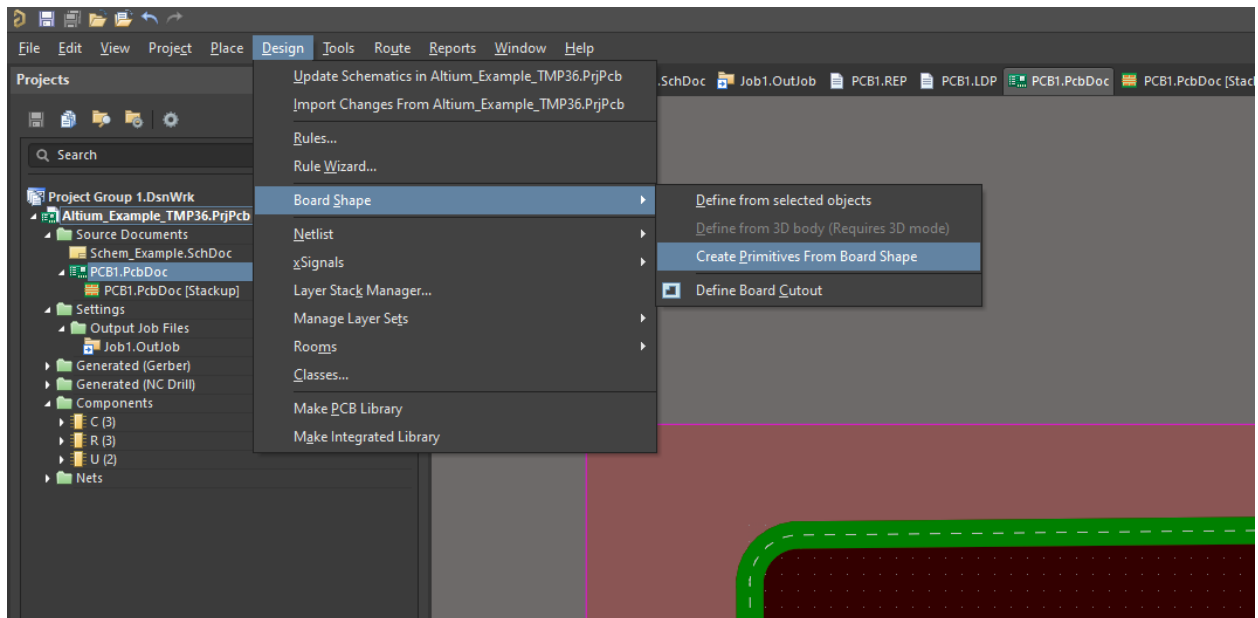




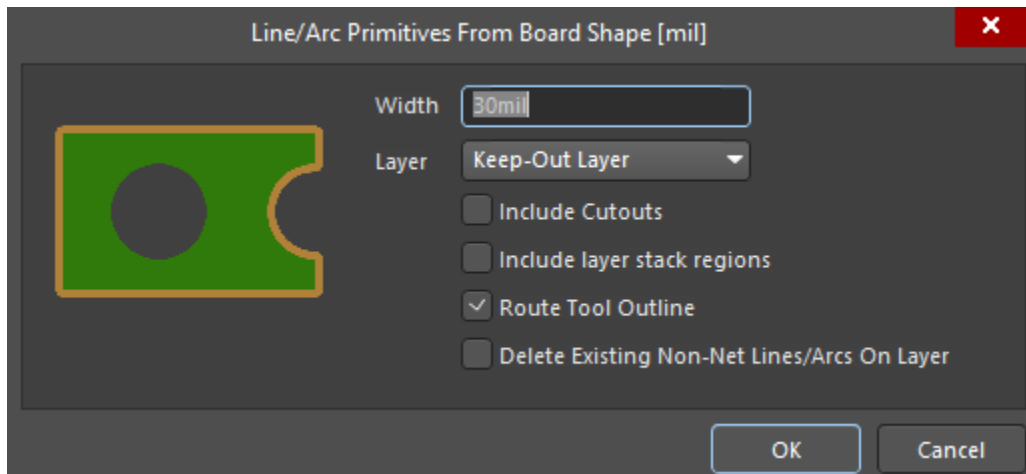
Bottom view:



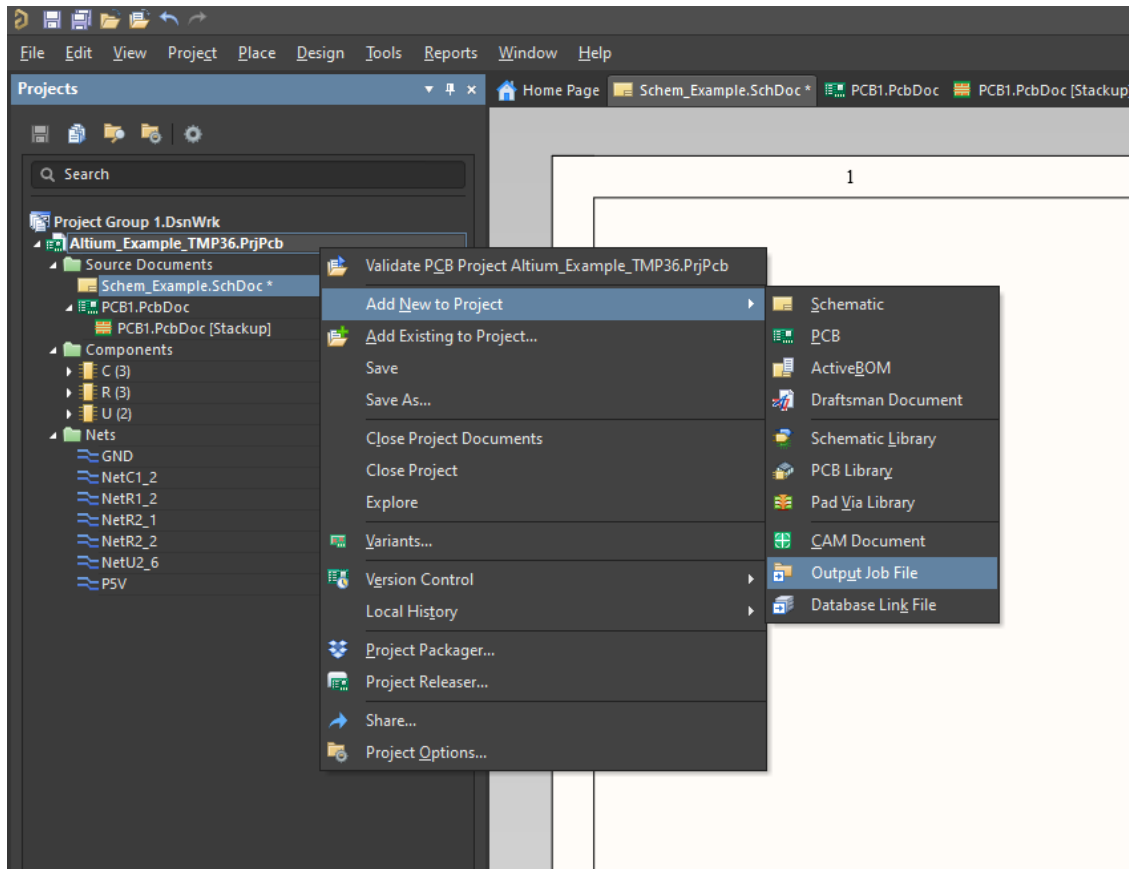
DEFINE BOARD SHAPE:

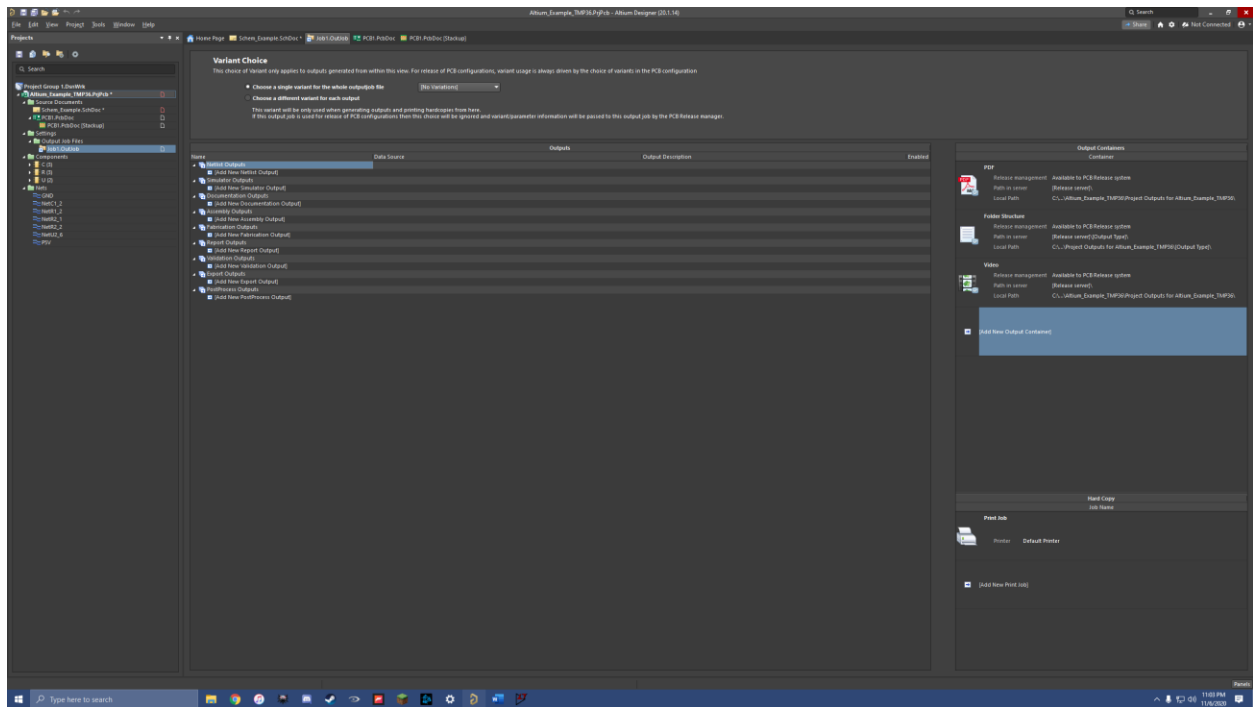


KEEP EXACT SETTINGS AS THE FOLLOWING:

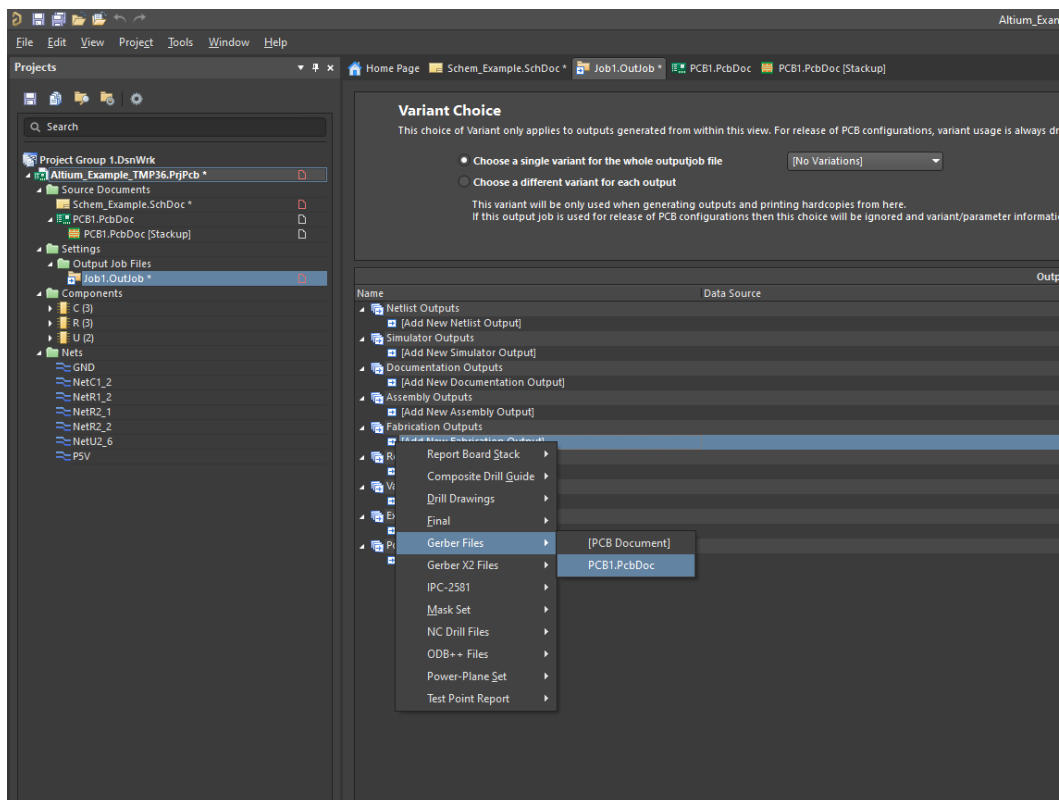


Output Job Guide:

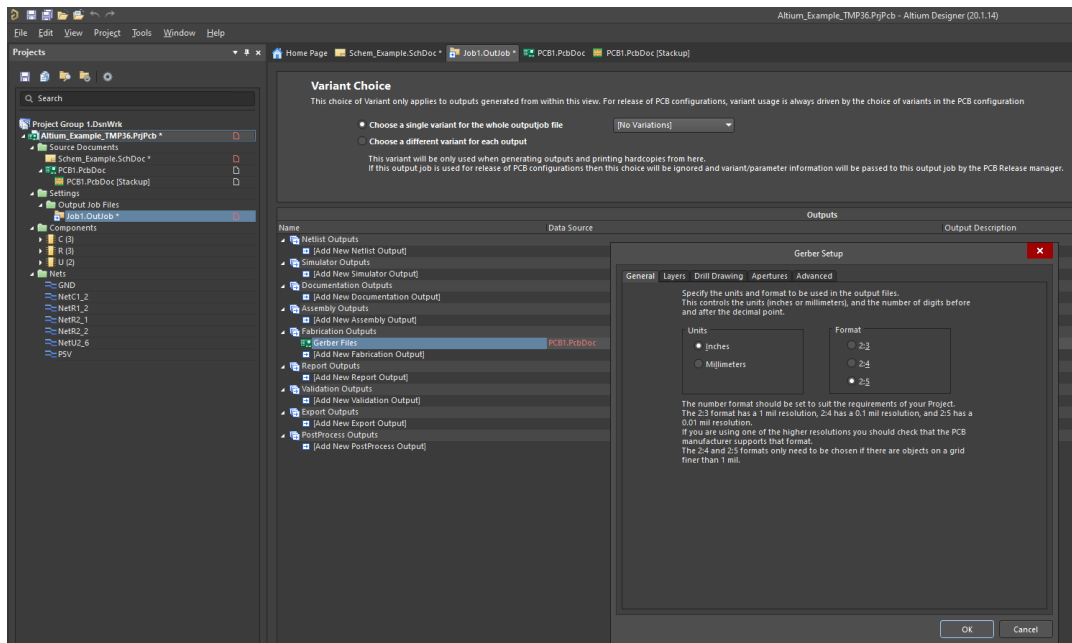




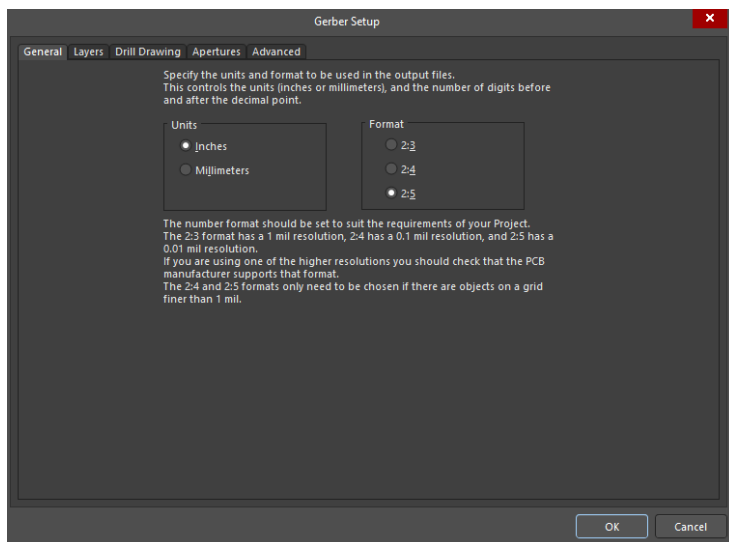
Fabrication files:



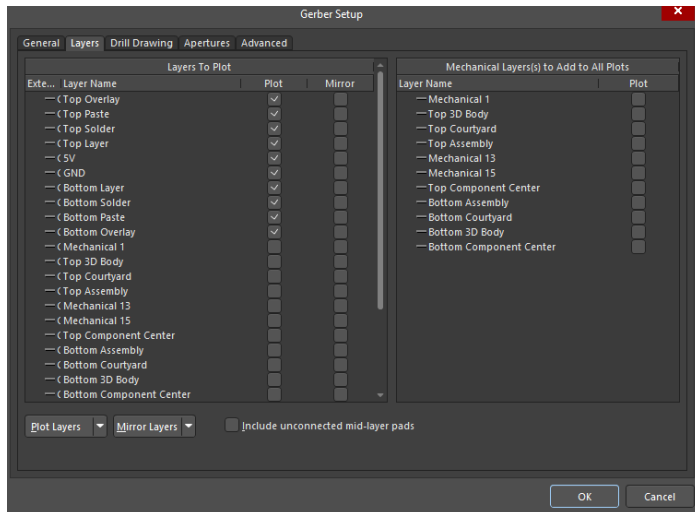
Double click "Gerber Files"



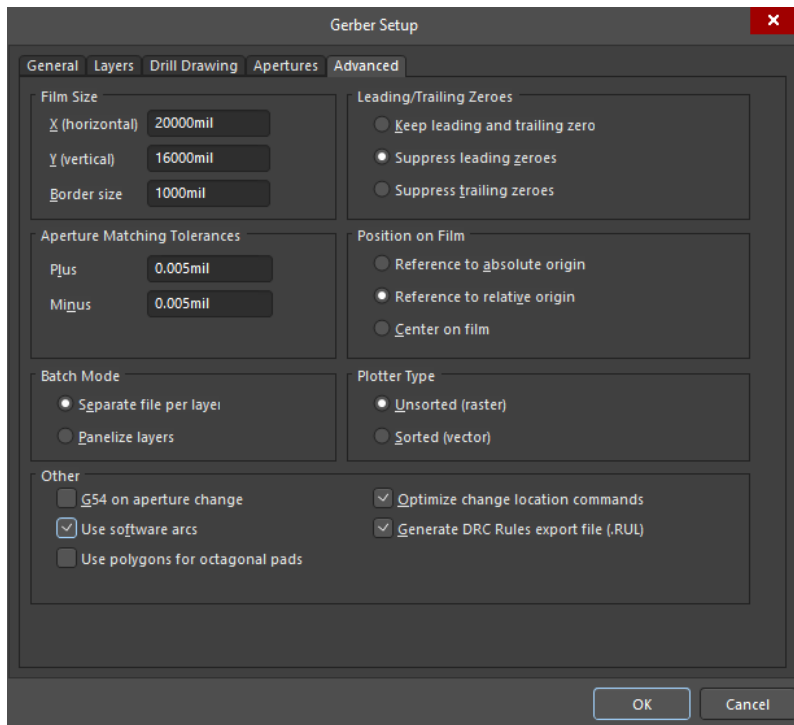
Set these settings up:



Under “Layers” Click the following:

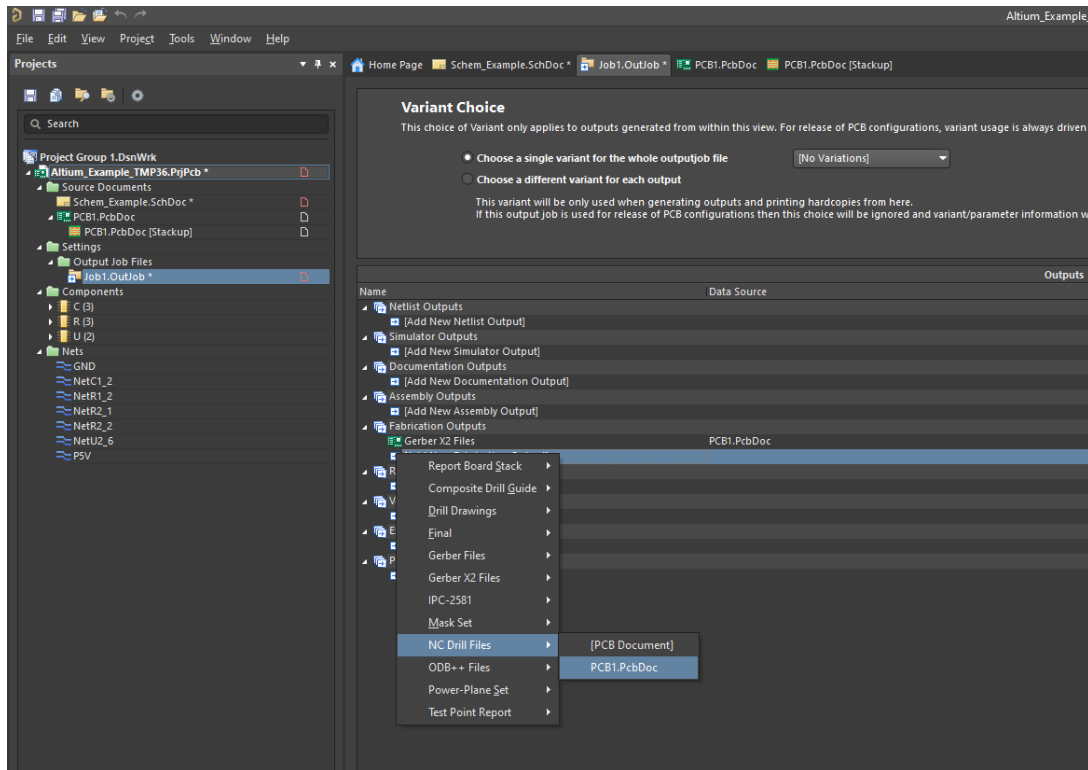


Under Advanced: (click use software Arcs)

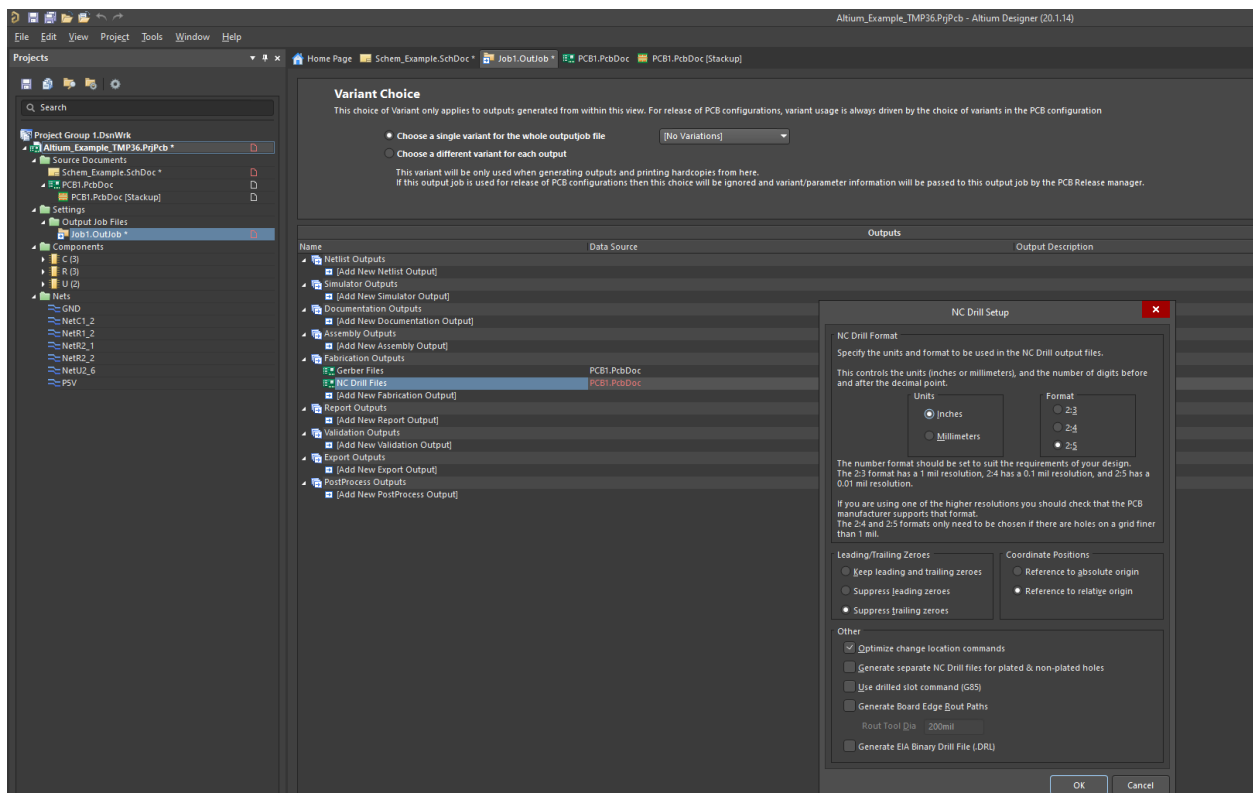


Clic

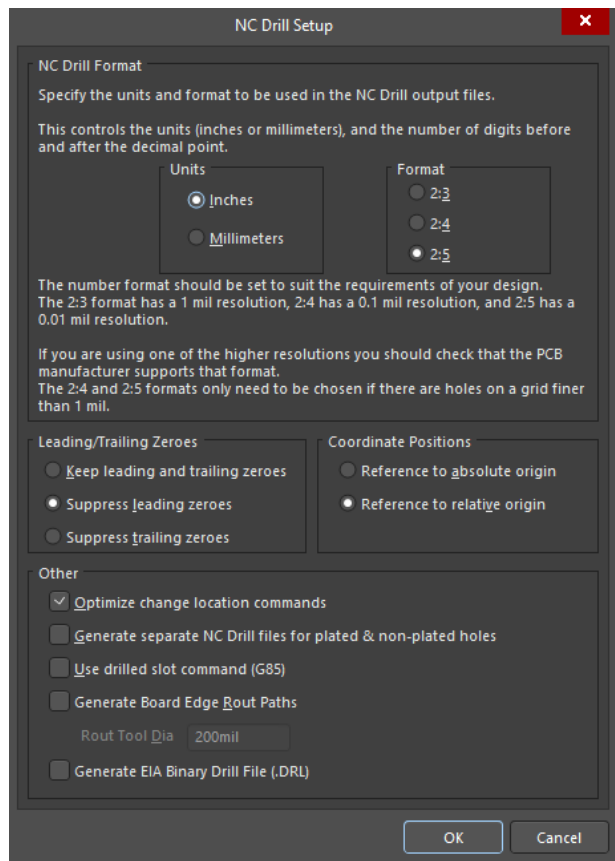
NC Drill Files:



Double click NC Drills:



Change to these settings:



The image shows a 'NC Drill Setup' dialog box with a dark gray background and a red close button in the top right corner. The dialog is organized into several sections. The 'NC Drill Format' section at the top explains that it controls units and decimal format, with sub-sections for 'Units' (Inches selected) and 'Format' (2:5 selected). Below this is explanatory text about resolution and a note about grid requirements. The 'Leading/Trailing Zeroes' section has three radio buttons, with 'Suppress leading zeroes' selected. The 'Coordinate Positions' section has two radio buttons, with 'Reference to relative origin' selected. The 'Other' section contains five checkboxes: 'Optimize change location commands' (checked), 'Generate separate NC Drill files for plated & non-plated holes' (unchecked), 'Use drilled slot command (G85)' (unchecked), 'Generate Board Edge Rout Paths' (unchecked), and 'Generate EIA Binary Drill File (.DRL)' (unchecked). A 'Rout Tool Dia' field is set to '200mil'. At the bottom are 'OK' and 'Cancel' buttons.

NC Drill Setup

NC Drill Format

Specify the units and format to be used in the NC Drill output files.

This controls the units (inches or millimeters), and the number of digits before and after the decimal point.

Units

- ☒ Inches
- ☐ Millimeters

Format

- ☐ 2:3
- ☐ 2:4
- ☒ 2:5

The number format should be set to suit the requirements of your design. The 2:3 format has a 1 mil resolution, 2:4 has a 0.1 mil resolution, and 2:5 has a 0.01 mil resolution.

If you are using one of the higher resolutions you should check that the PCB manufacturer supports that format. The 2:4 and 2:5 formats only need to be chosen if there are holes on a grid finer than 1 mil.

Leading/Trailing Zeroes

- ☐ Keep leading and trailing zeroes
- ☒ Suppress leading zeroes
- ☐ Suppress trailing zeroes

Coordinate Positions

- ☐ Reference to absolute origin
- ☒ Reference to relative origin

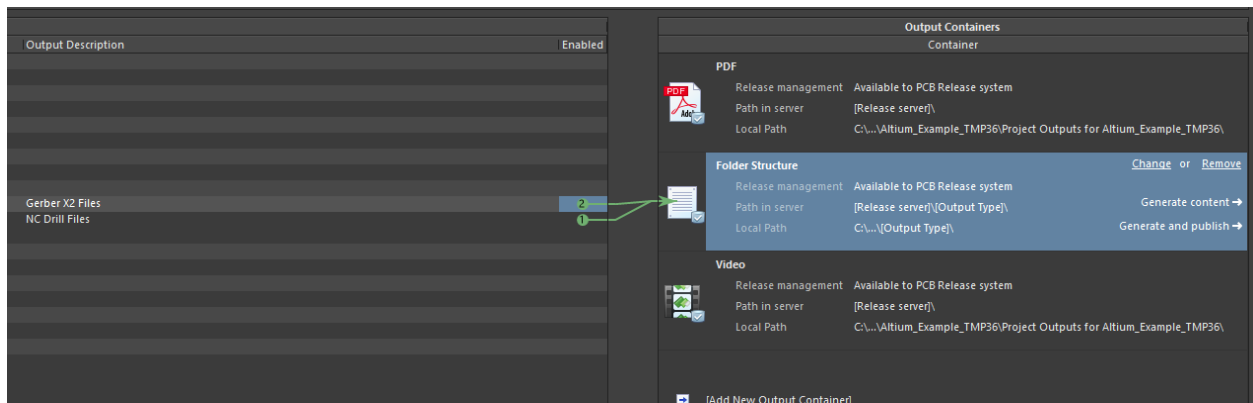
Other

- ☒ Optimize change location commands
- ☐ Generate separate NC Drill files for plated & non-plated holes
- ☐ Use drilled slot command (G85)
- ☐ Generate Board Edge Rout Paths
- ☐ Generate EIA Binary Drill File (.DRL)

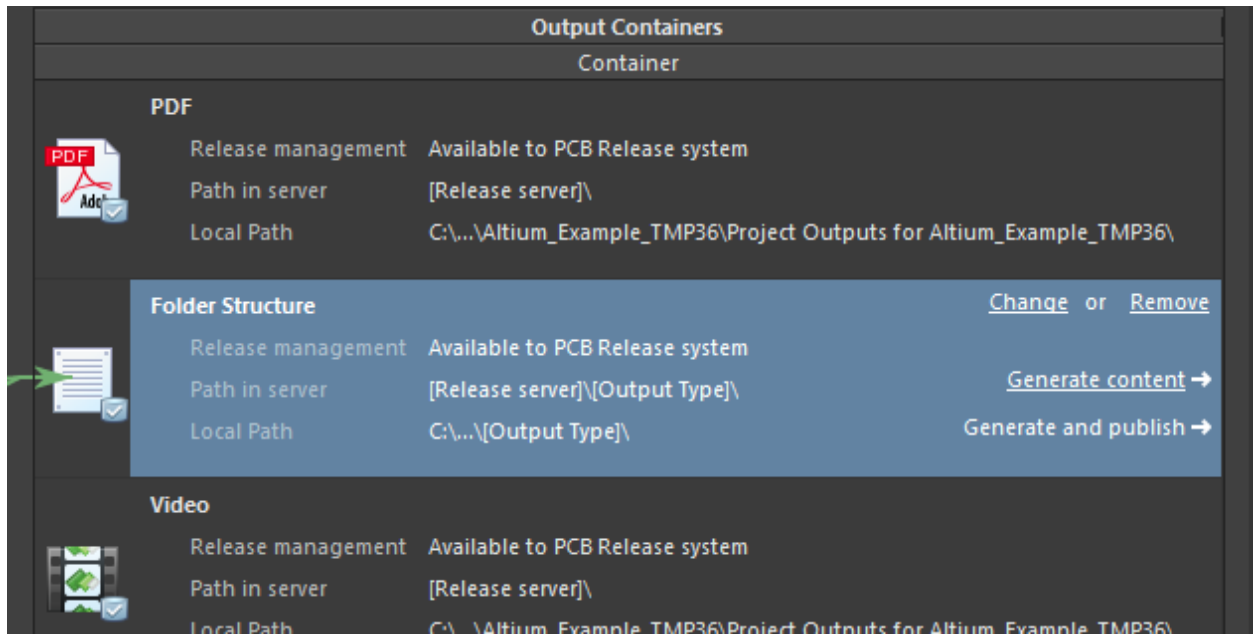
Rout Tool Dia 200mil

OK Cancel

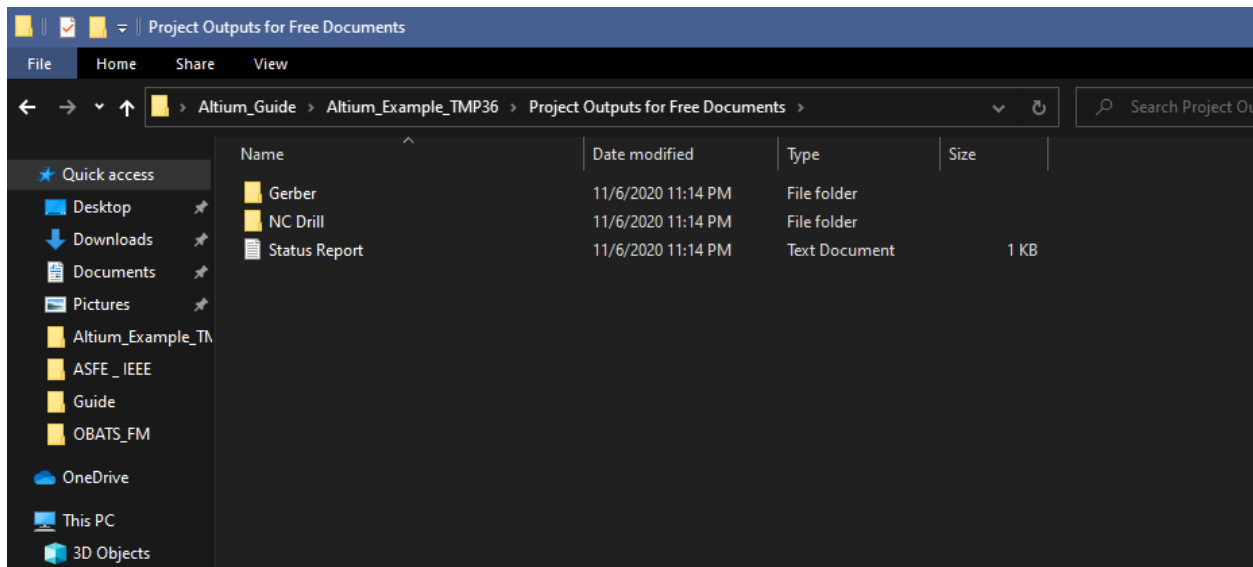
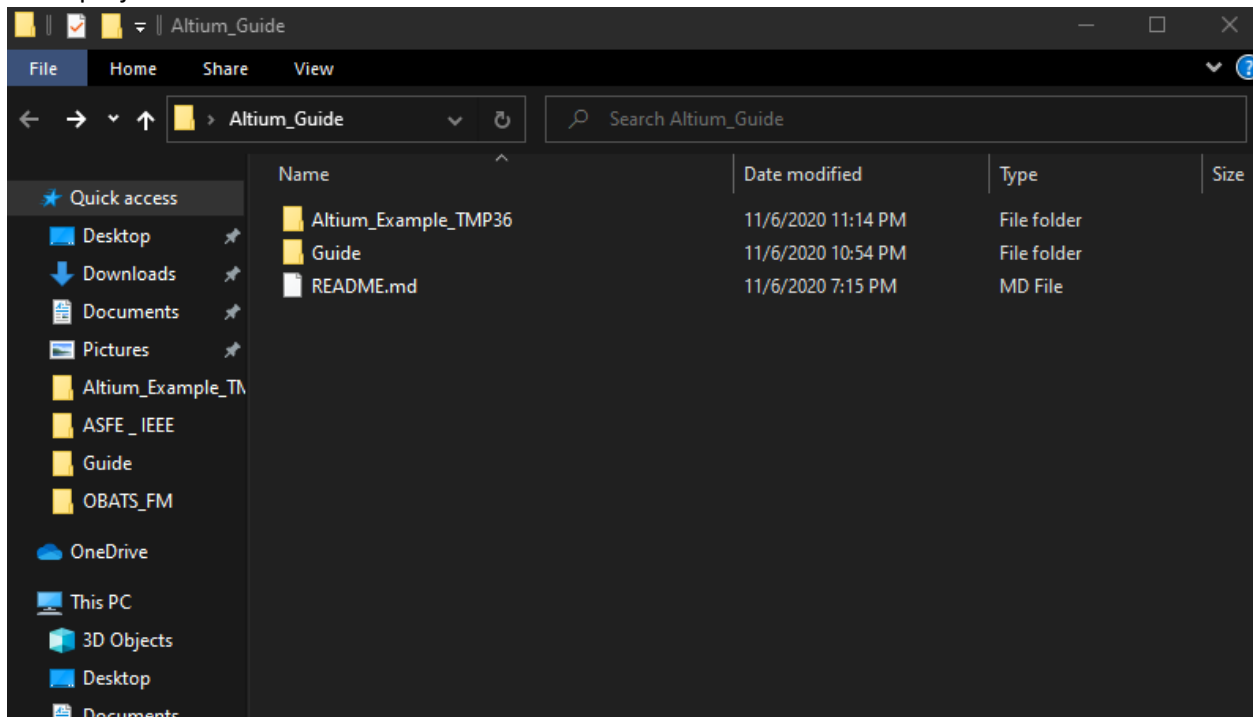
Hit 2 dots near “Folder Structure”



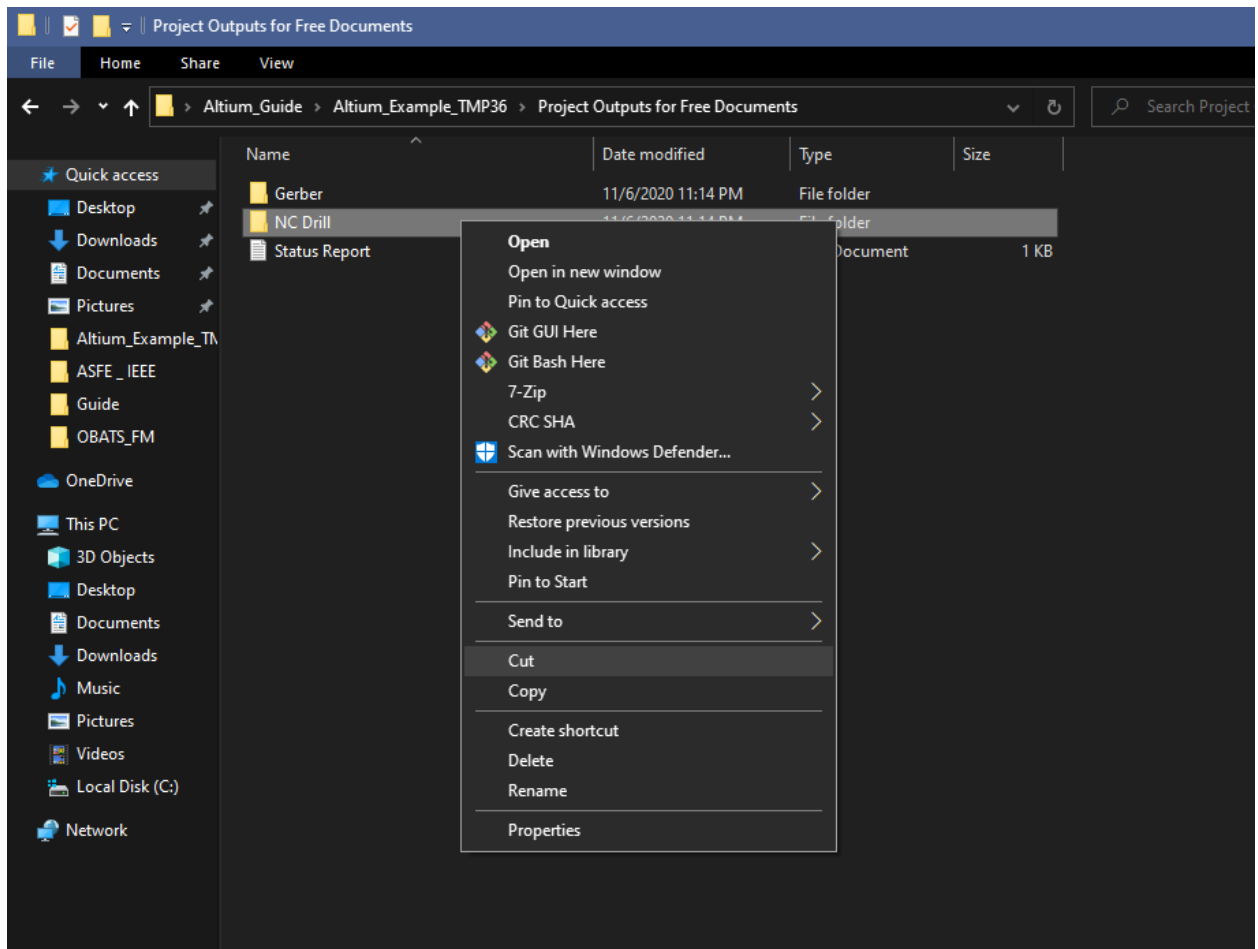
Click “Generate Content”:

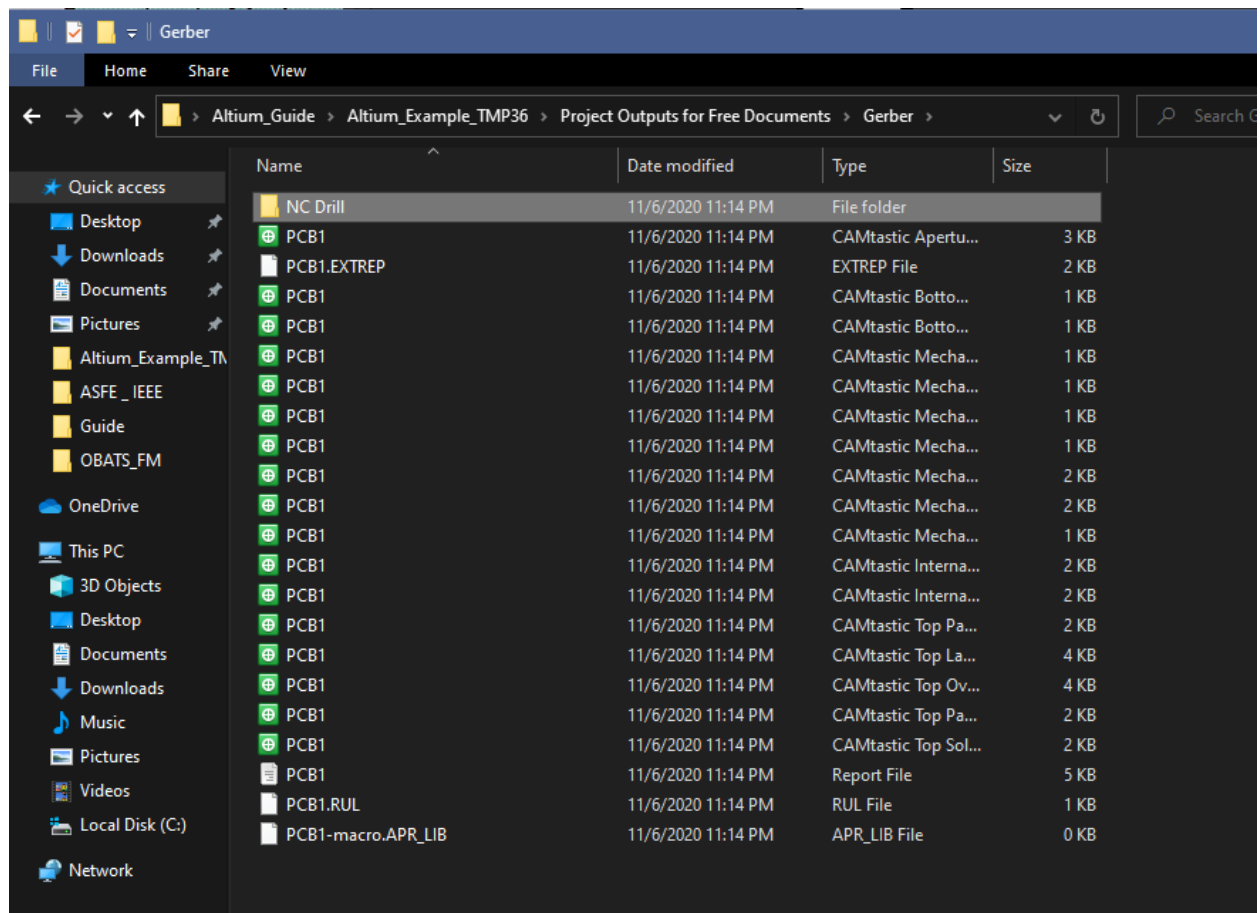


Go to project folder:

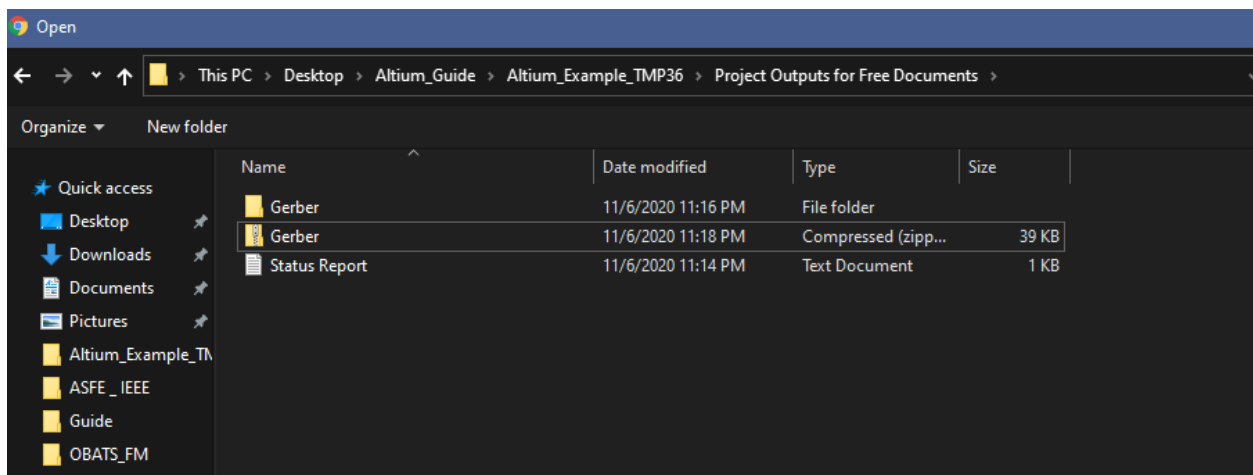
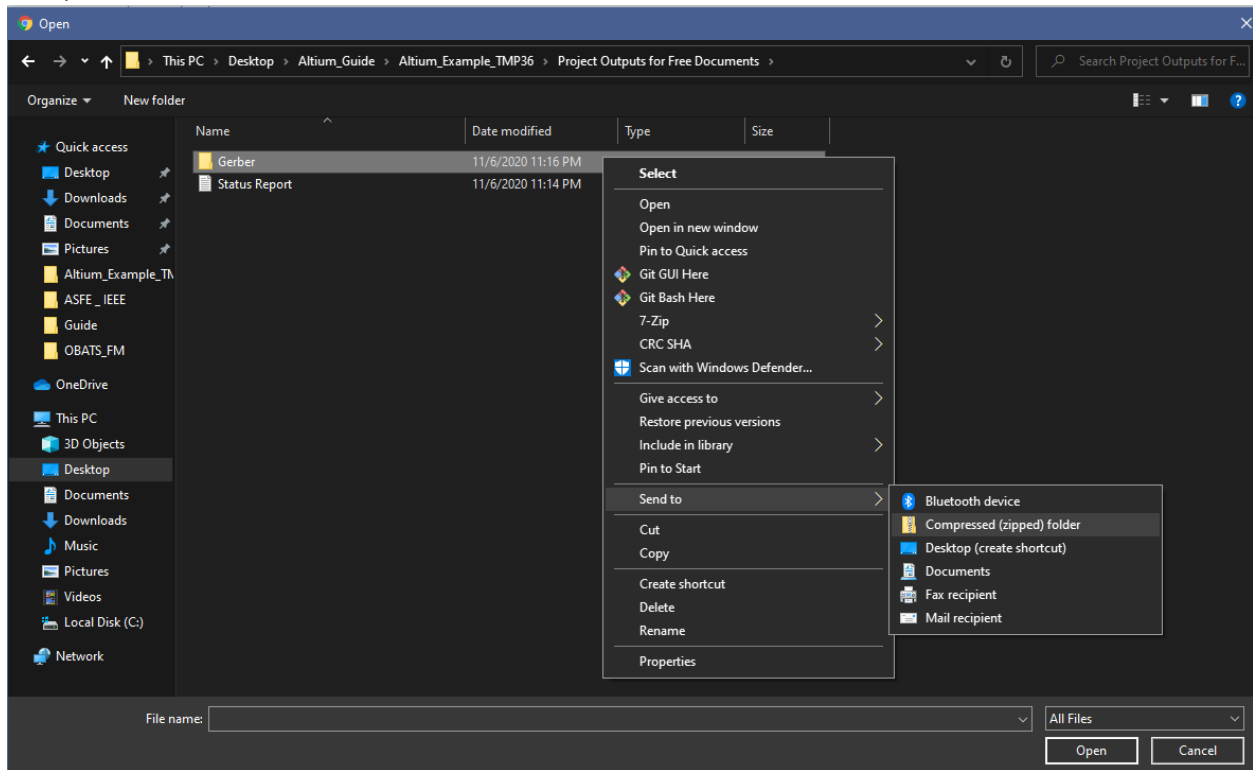


Cut the NC Drills folder and paste into Gerber folder:





Compress the Gerber Folder:



Upload zip folder to OSH PARK:

