

Altium Designer Workshop Part 2 Guide

Emily Marshall

Spring 2022

Contents

1	Introduction	4
2	Preparing the PCB	5
2.1	Adding New PCB to Project	5
2.2	Setting the Origin	6
2.3	Editing Board Shape	7
2.4	Tasks	8
3	Beginning Your PCB Layout	9
3.1	Moving from Schematic to PCB Layout	9
3.1.1	Understanding the Engineering Change Order	9
3.2	Initial Layout	12
3.2.1	Helpful Shortcuts and Tips	12
3.3	Fine-Tuning the Layout	13
3.3.1	A Bit of Theory	13
3.3.2	Some More Tips	14
3.4	Tasks	18
4	Routing	19
4.1	Setting Rules	19
4.2	Routing Differential Traces	22
4.3	How to Route a Single Trace	24
4.4	Polygons	24
4.5	Helpful Shortcuts and Tips	27
4.6	Tasks	29
5	Silkscreen	31
5.1	Adding Useful Silkscreen	31
5.2	Tasks	33
6	Additional Details	34
6.1	Adding Test Points	34
6.2	Adding Mounting Holes	36
6.3	Tasks	37
7	Bonus Material: Multilayer PCB Techniques	38
7.1	The Layer Stack	38
7.2	Adjusting the Stackup	38
7.3	Using Multiple Layers	40
7.3.1	Top Layer	41
7.3.2	Bottom Layer	41
7.3.3	Ground Layer	41
7.3.4	Power Layer	42
7.4	Using Vias	42

7.4.1	Adding Vias	42
7.4.2	Adding Via Stitching	44
8	Conclusion	46

1 Introduction

Welcome back! This is Part 2 of the Altium Designer workshop! If you are reading this, I assume you have already completed your CAN transceiver schematic from Part 1 and are ready to start the PCB layout. By the end of Part 2, you will have a completed PCB design for a CAN transceiver board! Your final product should look something like this:

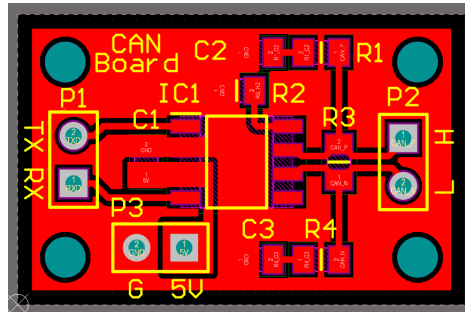


Figure 1: Sample 2D PCB Layout of the CAN Transceiver

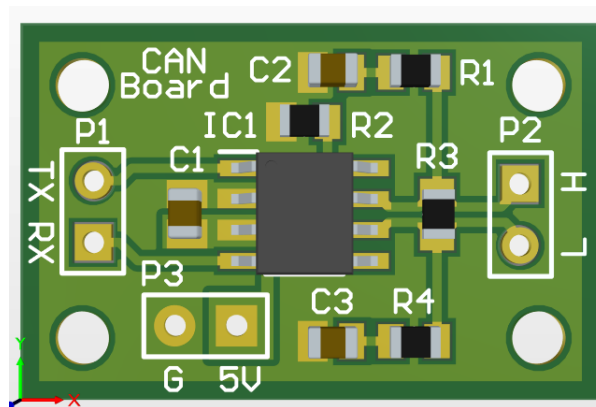


Figure 2: Sample 3D PCB Layout of the CAN Transceiver

If you have any problems after the workshop and would like help with Altium, please feel free to reach out to me at emarshall33@gatech.edu. But for now, let's get started!

2 Preparing the PCB

Sometimes the hardest part is just getting started! In this section, you will learn how to set up your PCB file.

2.1 Adding New PCB to Project

When you first start making a new PCB, you need to right click on your .PrjPcb file on the left panel and select **Add New to Project** → **PCB**, as shown in Figure 3.

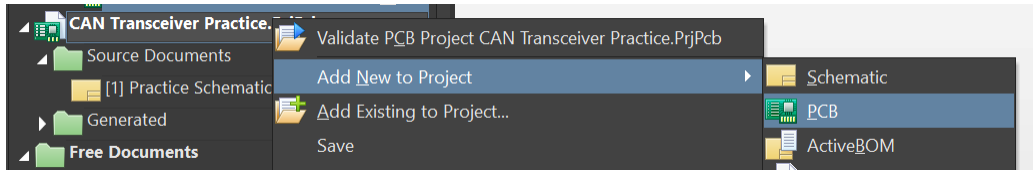


Figure 3: Screenshot of steps to add a new PCB to the .PrjPcb project

Make sure that your schematic (.SchDoc) and PCB (.PcbDoc) are under the same .PrjPcb file. If they are not, then you will not be able to move your schematic over to your PCB in a later step. If you accidentally get them separated, all you need to do is instead of selecting Add New to Project → PCB, just right click your .PrjPcb file and select **Add Existing to Project** so you can add the existing file to your project. If your setup was successful, you should now see a large black rectangle on the main part of your Altium screen. You need to save your PCB file before you go any further, so simply right click the .PcbDoc file, select **Save As**, and give it a meaningful name, as shown in Figure 4.

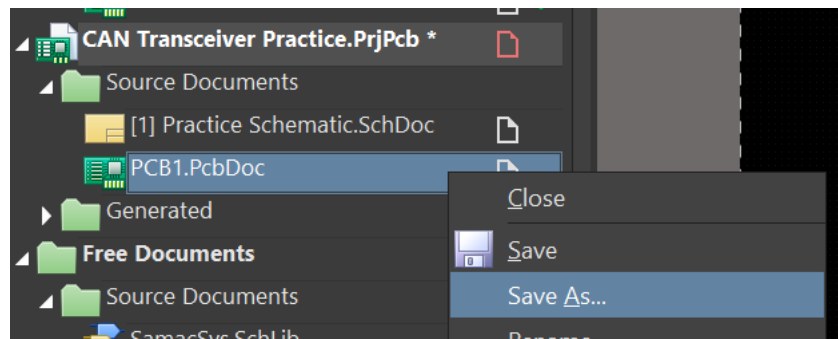


Figure 4: Screenshot of how to save .PcbDoc file

2.2 Setting the Origin

Your PCB needs to have an origin to reference, so let's set one! Go into the upper left corner to **Edit** → **Origin** → **Set** as shown in Figure 5.

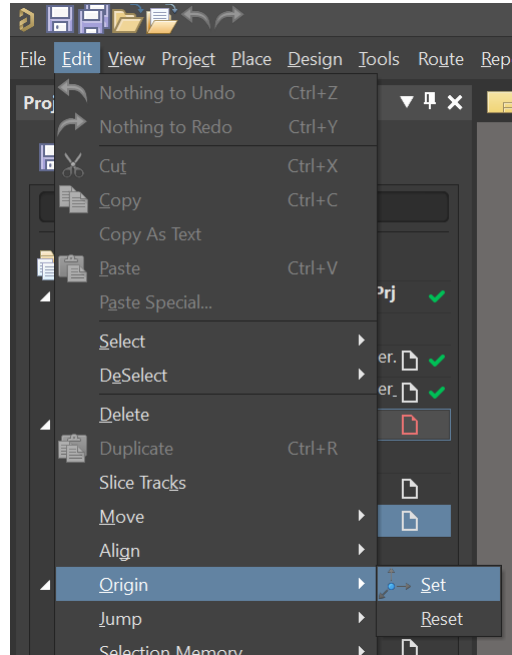


Figure 5: Screenshot of menu navigation needed to set origin

Next, you need to select the bottom left corner of the black rectangle on the screen. This will be our $(0, 0)$ (just like in Cartesian coordinates). You can zoom in a bit to be more precise, but once you are happy with your placement, click the spot to lock in the origin. The origin will be at the center of the white circle with a cross that appears. (Note that if you make a mistake in placing the origin, just go back to the menu to set the origin again). See Figures 6 and 7 for reference on placing the origin.

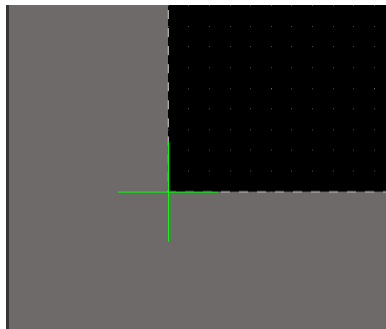


Figure 6: Screenshot of the green cross-hairs that mark where you will be placing the origin

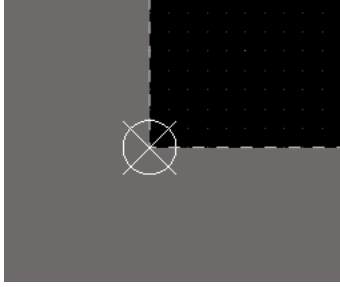


Figure 7: Screenshot of the white circle with the cross that marks the location of the origin once it is set

2.3 Editing Board Shape

While you are working with a .PcbDoc file, you have three different modes you can view. By pressing **1** on your keyboard, you can access **Board Planning Mode**. By pressing **2** on your keyboard, you can access **2D Layout Mode**. By pressing **3** on your keyboard, you can access **3D Layout Mode**. By default, the .PcbDoc opens in 2D Layout Mode when it is first created. That is the mode we are in right now.

In order to edit the board shape, you want to be in Board Planning Mode, so press 1 on your keyboard. Once you are in Board Planning Mode, you should see the black rectangle on your screen is now green. In the upper left of your screen, go to **Design** → **Edit Board Shape**, as shown in Figure 8.

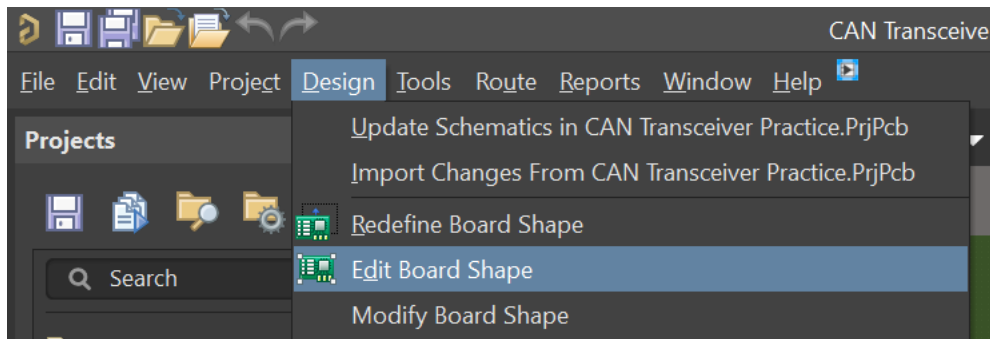


Figure 8: Screenshot of steps to edit board shape from the menu

You should see a white border appear around the green rectangle. You can now drag the edges of the board to make it the size that you want, as shown in Figure 9. Note that you need to respect your origin, so make sure you are only adjusting the right and top edges of the board. If you touch the other sides of the board, you will need to reset your origin. As a note, if you need to set your board to a specific size, it helps to look at the bottom left corner of your screen where you can find the coordinates of your cursor. For example, if you wanted to make a board that was 800 mil x 500 mil, you could drag the right edge inward until you reach an x value of 800 and drag the top edge downward until you reach a y value of 500.

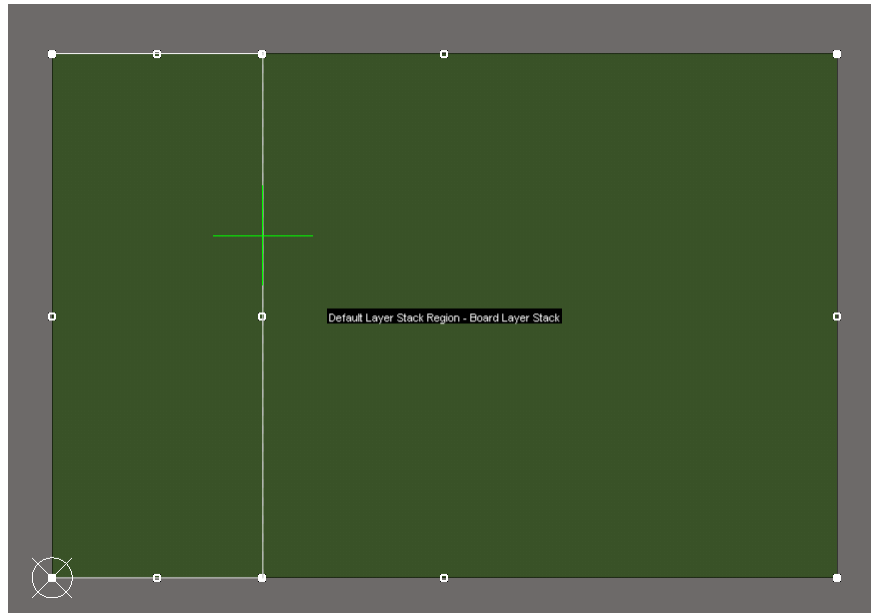


Figure 9: Screenshot showing how to edit board shape by dragging board edges

2.4 Tasks

In this section, you are going to prepare a PCB file for your project. Using the techniques discussed above, complete the following steps:

1. Add a new PCB file to your .PrjPcb file for your CAN transceiver project.
2. Set the origin to the bottom left corner of the black rectangle on the screen.
3. Press 1 on your keyboard to enter Board Planning Mode.
4. Change the dimensions of your board to be **940 mil horizontally and 610 mil vertically**. This is not a standard size, but is just what we will be using for this practice project. (Hint: Look in the bottom left corner for your coordinates!). **Note: If for some reason your coordinates are in mm instead of mils, simply press 'q' on your keyboard to switch units.**
5. Once you are done adjusting the dimensions of your board, press 2 on your keyboard to return to 2D layout mode.

3 Beginning Your PCB Layout

Yay! You have successfully set up your .PcbDoc file and are ready to move everything over from your schematic to your new PCB! In this section, you will learn how to import from your schematic and create a PCB layout. For this section, it is very important to make sure that you have your schematic and PCB under the same .PrjPcb file, otherwise importing won't work.

3.1 Moving from Schematic to PCB Layout

To begin, you should be in 2D Layout Mode (press 2 on your keyboard if you are not in 2D Layout Mode already). Go up to the upper left of your screen and select **Design** → **Import Changes From [Your Project Name]**, as shown in Figure 10.

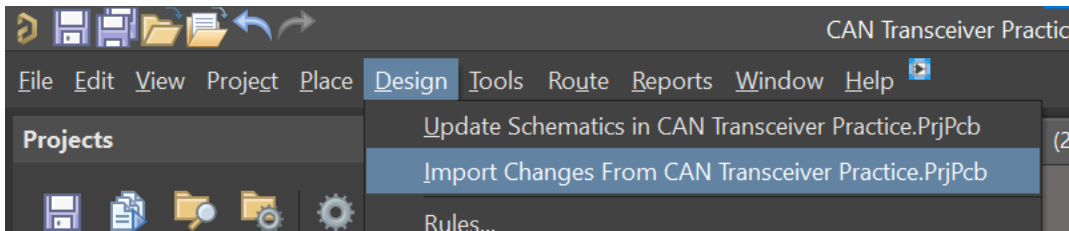


Figure 10: Screenshot of steps to import changes to PCB

Another window will pop up that is called the Engineering Change Order or ECO. Think of it sort of like someone reading you back your order. The ECO just wants to make sure it is going to be doing everything you would like before you go through with it.

3.1.1 Understanding the Engineering Change Order

Below are some example images (Figures 11 - 15) from an Engineering Change Order. Note that you may have different sections in yours.

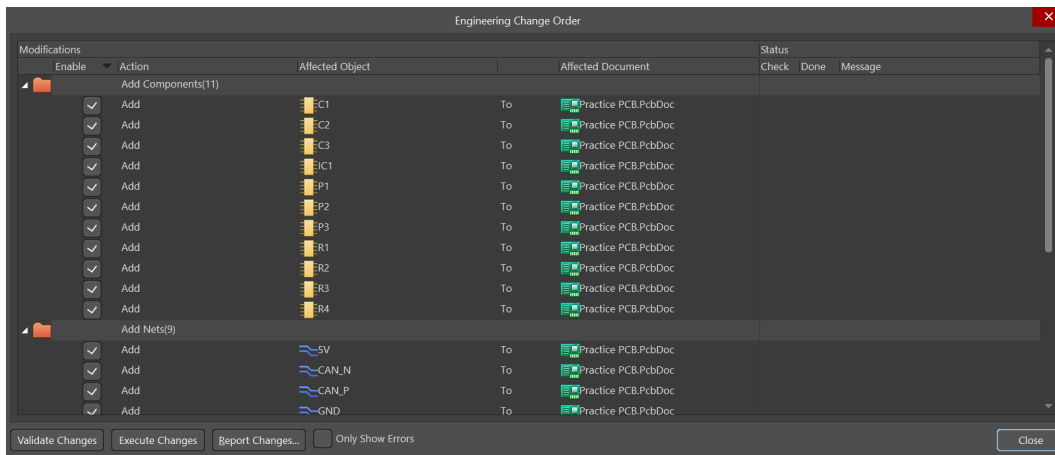


Figure 11: Screenshot of Engineering Change Order window

In Figure 11 we see part of the ECO that pops up. It may seem overwhelming at first, but we'll break it down piece by piece.

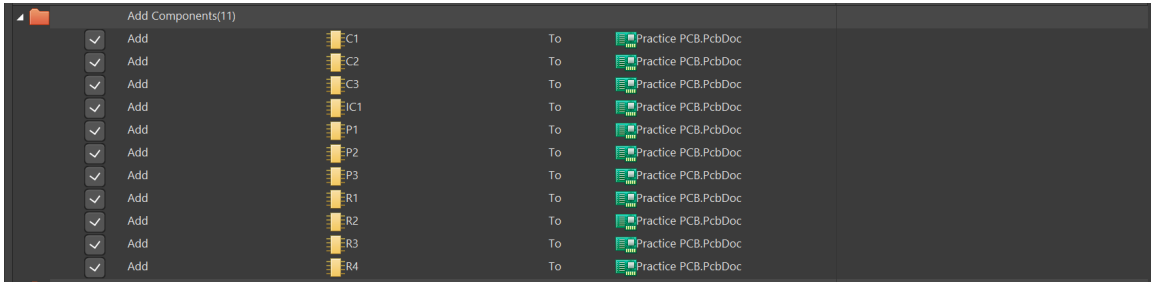


Figure 12: Screenshot of ECO Adding Components

In Figure 12, the ECO is showing all the components that it is going to add to your schematic. It's basically saying, "Hey, I noticed that all these components on your schematic are missing from your PCB. I'll add them for you." If this is okay (which it usually is), leave the boxes checked.

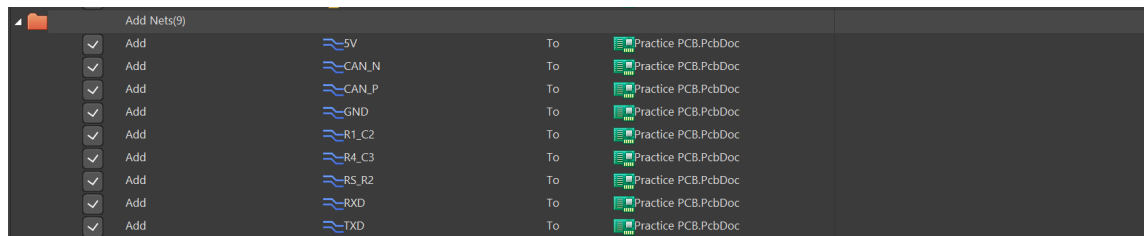


Figure 13: Screenshot of ECO Adding Nets

In Figure 13, the ECO is showing all the nets that it is going to add to your schematic. Like with the components, the ECO is trying to tell you, "Hey, I'm going to add these nets that I noticed are missing." This is normal for your first time importing from the schematic in a project. Leave the boxes checked.

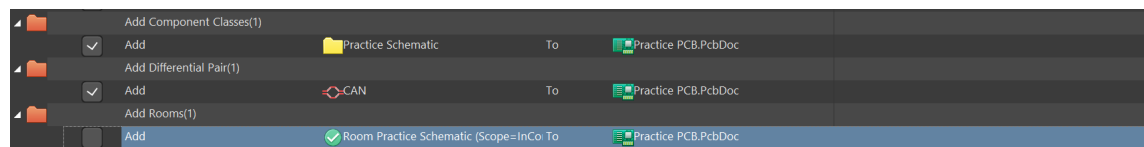


Figure 14: Screenshot of Removing Rooms from ECO

In Figure 14 we see three different sections of the ECO. One is called "Add Component Classes." To be honest, I'm not sure exactly what that one does. I'd just leave it. In general, if you don't know what an ECO section is doing, it is usually okay to just leave it. The only ones I really uncheck (other than the rooms which I will mention in a moment) are the ones where it is trying to change names of components I have already renamed in the PCB. This is not shown in the above example, but I am just letting you know you might see it. The second section shown in the figure is called "Add Differential Pairs."

This one is definitely important because you want to make sure your differential pairs are routed correctly. If you are making a board that you expect to have differential pairs, but you don't see the "Add Differential Pairs" section in the initial ECO, go back to your schematic and make sure you marked and labeled the differential pair nets correctly. The last section in the figure is called "Add Rooms." I almost always uncheck the box for this one. Rooms basically collect all of your components for a particular schematic and import them in a little box. It's annoying for a board coming from one schematic because then you just have to delete the room, but they are helpful for designs that involve multiple schematics coming into one board because it improves organization.

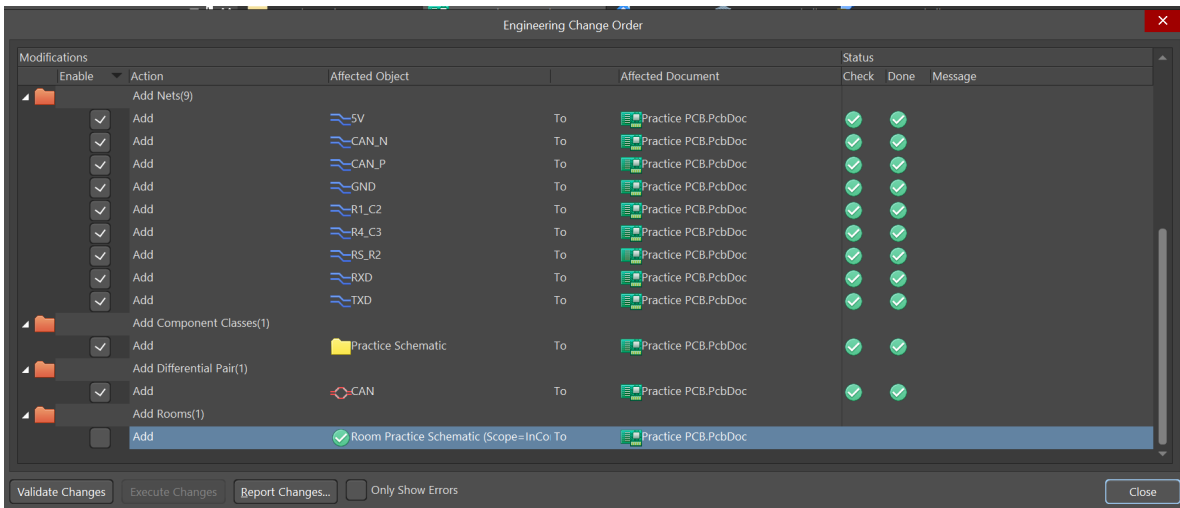


Figure 15: Screenshot of a Successful ECO

In Figure 15, we see there are two columns of green check marks next to each row of the ECO. When you see the ECO, you should scroll through and uncheck anything you don't need. Then click the **Validate Changes** in the bottom left corner of the window. The first column of check marks should appear. If you see a red x instead of a green check, that means that there is a problem and you should consider cancelling the ECO and going back to the schematic to see what's wrong. However, sometimes a red x will appear on the first time you run the ECO for a project because nothing is on the board yet. For example, you will likely see a red x next to "Add Differential Pairs" because it doesn't think those nets have been added yet, but once you execute the changes, it'll realize it's okay. When you are satisfied with the ECO and have validated the changes, click **Execute Changes**, which is the button in the bottom left that is directly next to the Validate Changes button. If you see all green check marks, you're good to go. If you see any more x's, no worries. Just Ctrl+Z it to get rid of what you just did and go back and fix whatever upset the ECO. Sometimes it's an iterative process. Note that anytime you update the schematic (even if you already have things placed on the PCB), you need to import your changes and run through a new ECO. Sometimes it will even ask you to match nets if you have recently replaced components or changed net names. I usually like to do this manually so that I can see what it is pairing up.

3.2 Initial Layout

Once the ECO is complete, you should now see your components all gathered off to the right of your board, as shown in Figure 16.

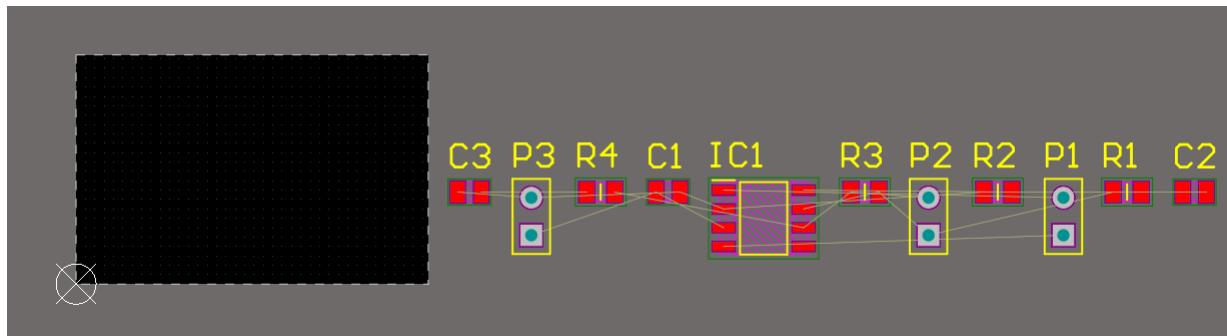


Figure 16: Screenshot showing components generated after ECO

The first thing you are going to want to do is to make a rough initial layout. This is where you take a look at what you have on your schematic and you place components in the general area they'll need to be in. This is the time to think logically about how your board is going to be connecting to other boards in the system. For this board, think about where the power should come in, where the signals should come in, and where the signals should leave. In this example, I recommend having the TXD/RXD signals coming in on the left, the CAN signals leaving on the right, and power coming in near the bottom, just for this example. You should also begin rotating components so that they are in a good orientation for connection. Line up pads that are on the same net so that the traces connecting them can be shorter. You will see faint lines connecting the pads before you route them, and these are called airwires. In general, you want to try to minimize the number of crossing airwires because this will make it easier to route things later, but don't stress about uncrossing all of them, because that's basically impossible in a lot of cases.

3.2.1 Helpful Shortcuts and Tips

1. It doesn't have to be perfect yet! Don't forget that this is just a rough draft of the board layout.
2. Clicking and holding a component and pressing space at the same time will rotate it just like it does in the schematic.
3. Right click and hold allows you to grab and move your PCB space.
4. Left clicking and dragging from right to left to select objects allows you to select an entire object by only touching part of it.
5. Left clicking and dragging a from left to right to select objects allows you to select an entire object only by dragging over the entire object. This is useful when there are components spaced tightly together and you want to move only one or two of them.

6. Clicking the mouse wheel and moving the mouse up/down lets you zoom in/out.
7. Scrolling up and down scrolls your view up and down. Scrolling up and down while holding the Shift key scrolls your view left and right.

3.3 Fine-Tuning the Layout

Once you have completed your initial layout to your satisfaction, you are ready to fine-tune it and make it ready for routing.

3.3.1 A Bit of Theory

- There are certain components on a board that are more important than others. For example, if this board had a microcontroller on it, it would likely have a crystal oscillator. This crystal oscillator or clock resonates at a certain frequency and helps provide the clocking the microcontroller needs to perform all of its tasks. Without it, the entire circuit would malfunction. Because of this, special care needs to be taken to make sure that clocks are placed (and routed) first so that they get priority. When you are designing a layout with a clock, place the clock and its supporting components (usually two capacitors) close to the IC, as shown in the "Best Practices for Oscillator Layout" document from Atmel (http://ww1.microchip.com/downloads/en/DeviceDoc/Atmel-8128-Best-Practices-for-the-PCB-Layout-of-Oscillators_Application_Note_AVR186.pdf).
- Another important component is anything that is analog. Analog signals are very sensitive and components that are receiving analog signals should be placed far away from digital signals and components that have significant switching that could interfere with the analog signals. This doesn't apply to this board, but it is good to know.
- Components connected to differential pairs and other timing-sensitive signals are also important. With differential pairs and timing-sensitive signals, you need to make sure the traces are close to the same length or else one signal will get there slightly before the others, causing the data to become garbage. To minimize this possibility, plan to make the traces approximately the same length (and preferably a short length) by placing their corresponding components near each other.
- Perhaps one of the most important component types to lay out carefully is bypass or decoupling capacitors. These capacitors connect between power and ground and help keep voltage in the circuit steady. When you have a lot of switching in the IC, voltage is bound to drop sometimes, but when this happens, the bypass capacitor is there acting as a sort of reservoir that will push the system to the right level to keep things steady. The only problem here is that nothing is ideal. In reality, there is parasitic inductance in PCB traces that can cancel out some of the benefit of these capacitors. To minimize the parasitic inductance, all you have to do is place the bypass capacitors as close as possible to their corresponding IC power and ground pins. See the example layout at the beginning of this document for an example.

3.3.2 Some More Tips

In general, whenever you are making a board, you want it to be something that you are proud of. Making the layout clean and as symmetric as possible helps you put your best foot forward (especially if you are going to be showing off your board to potential employers). Below are some tips to help fine-tune your layout.

- A useful thing to do when fine-tuning your layout is to align components so that they aren't placed haphazardly. There are a few ways to do this. You can either double click a component and change its X/Y location in its properties (as shown in Figure 17) so it is at the same X or Y position as the component you are aligning with. Or, the easier option is to use the align tool. Simply select the components you would like to align, right click one of them, and select **Align** then whichever type of alignment you would like. In the example shown in Figure 18, we want the components to be at the same vertical height, so we can select **Align Vertical Centers**. If you want them to be at the same horizontal position, you can select **Align Horizontal Centers**. Once you select your alignment type, click the component you would like the alignment to be referenced to, then the other components should fall into place automatically.

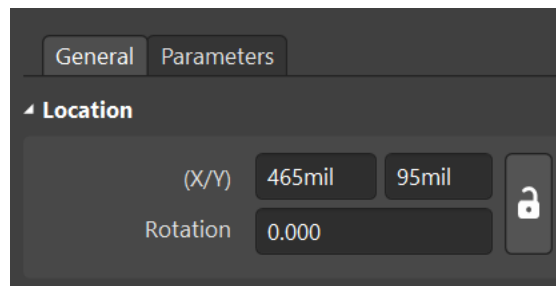


Figure 17: Screenshot of X/Y location settings in Properties panel

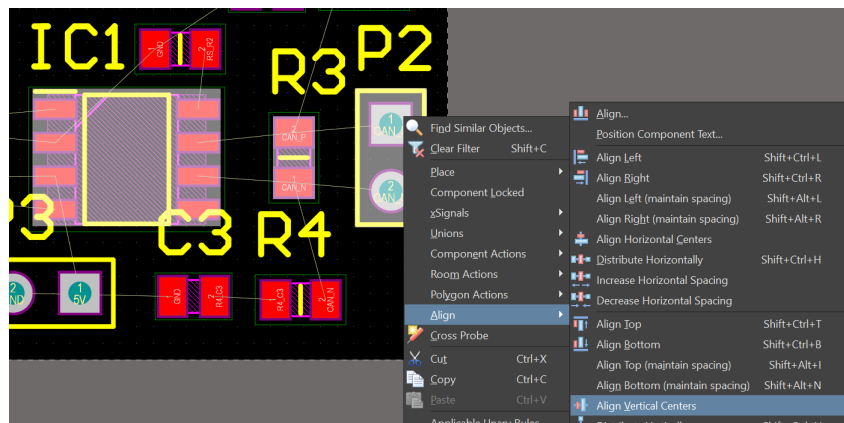


Figure 18: Example of using the Align Vertical Centers alignment type

- Another type of alignment that is helpful is distributing components vertically or horizontally. Especially when you have many of the same component that you would

like to have in a nice row or column, you can select them, right click on one of the components, then select **Align** and whichever alignment type you would like. In Figure 19, we are distributing the components horizontally, which will space them out evenly horizontally. Sometimes this doesn't work as expected though because components can have different origins which makes it tricky to figure out how to align them. Ctrl+Z is your friend if anything goes wrong. No worries, just try again!

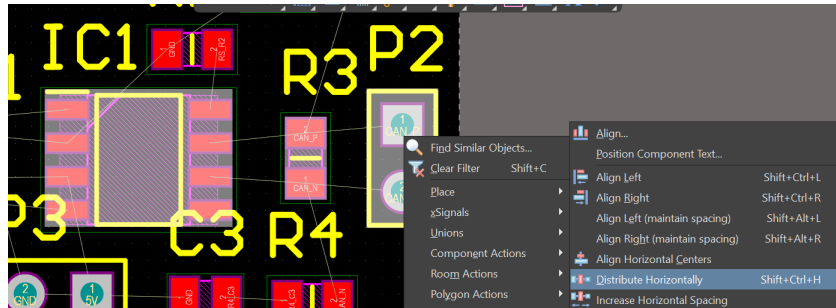


Figure 19: Example of using the Distribute Horizontally alignment type

- Another useful trick is to use the selection filter. Let's say you only want to change the size of the text (because maybe it is distracting and large, which it often is by default). Instead of having to go around selecting every piece of text, you can simply use the selection filter to select only text. Go into the bottom right corner of your Altium screen and select **Panels** then **Properties**. At the top, you should see something like in Figure 20, which has every object type in blue showing that they could all be selected. Select the **All-On** button in the Selection Filter to turn all of the objects grey (meaning that none of them can be selected on the board). Click on your desired objects (in this case, Texts) and they should turn blue again. This means you can select them on the board. Use Ctrl+A to easily select all of the text now. Figure 21 shows how to set the Selection Filter for texts only. With all of the text selected, you can even change the size of all of the text at once, as shown in Figure 22. I like to set my text size to a **text height of 36 mil** and a **stroke width of 6 mil**. This works for JLCPCB, which is the company we usually order boards from.

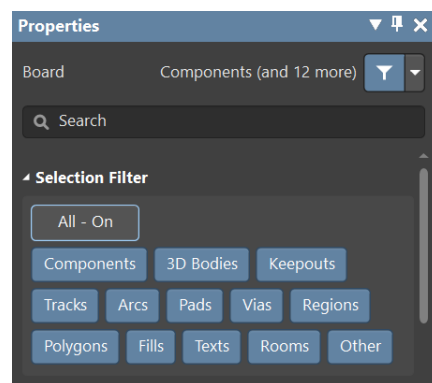


Figure 20: Screenshot of default selection filter

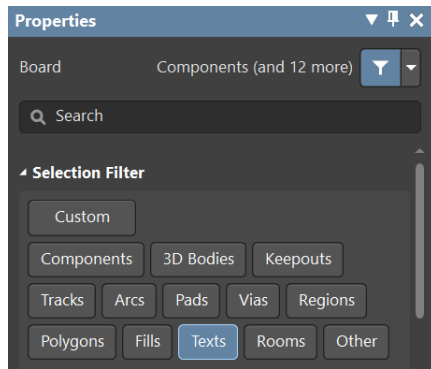


Figure 21: Screenshot of selection filter set for texts only

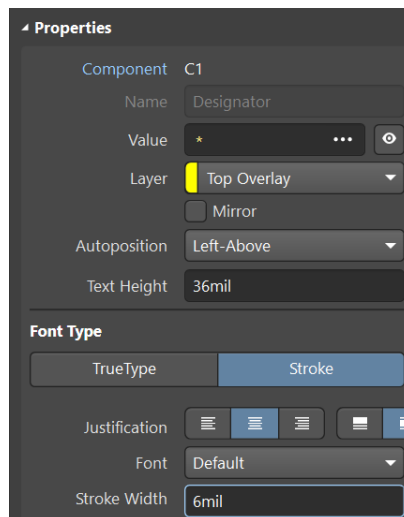


Figure 22: Screenshot showing text adjustment in properties

- **AN IMPORTANT NOTE ABOUT THE SELECTION FILTER:** Make sure you go back and make it so that all the objects are blue in the selection filter again. Selecting the "Custom" button at the top of the selection filter should do this. If you forget to set it back to letting you select all objects, you may find that you suddenly can't move components, traces, etc. When in doubt, check the selection filter!

3.4 Tasks

In this section, you are going to create a PCB layout for your project. Using the techniques discussed above, complete the following steps:

1. Import changes from your schematic to your PCB.
2. When all your components are imported, create an initial layout (remember this is just a rough draft! The components don't have to be perfectly aligned just yet). It should look something like this:

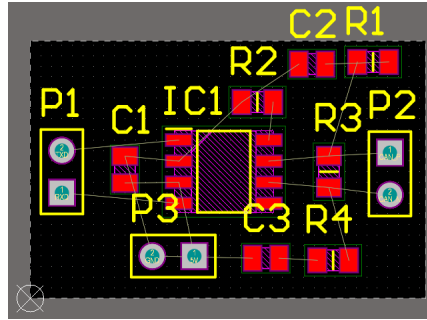


Figure 23: Initial layout example for CAN Transceiver board

3. Once you are happy with your initial layout, fine-tune it so that components are better aligned for routing (and for aesthetics). A decent fine-tuned board layout is shown in Figure 24.

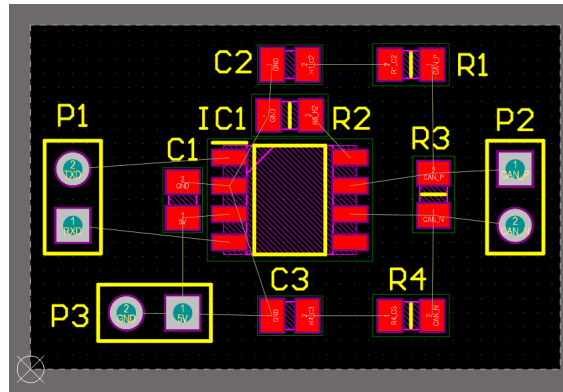


Figure 24: Example of a decent fine-tuned PCB layout ready for routing

4 Routing

Once you are done with the PCB layout, you are ready to route all of your traces! Note that it is still totally okay to readjust your layout as you go to make routing easier (I do it all the time because often you'll place things in an area that makes sense but once you go to route it, you realize there's no space!). Be patient, be kind to yourself, and think of PCB layout and routing as a sort of puzzle. Honestly, one of my biggest pieces of advice if you ever get stuck and feel like there's no way to route something is to take a step back and ask yourself, "Why is this component in this position?" See if you can figure out new ways the component could be placed so that you can route more efficiently. Think outside of the box. I like to think of it like entering your apartment on move-in day. Most of the time, the furniture is not exactly where you'd like for it to be. Maybe the desk is facing the wrong way or the bed isn't at the height you like. You have to take a look at the room and reimagine how the furniture could be arranged so that it is functional and aesthetically pleasing. This is essentially exactly what you are doing when designing a PCB.

4.1 Setting Rules

Before we begin routing, it is important to make sure that you don't get too far without setting some rules for your design. JLCPCB, a company we can order our PCBs from, has certain requirements for boards you submit. If your boards don't fit their requirements, their machines can't make it. We won't go over all the nitty-gritty details of their rules in this document (more details can be found on their website, or on the website of any fab house you choose to send boards to). For this practice project, you will only need to set a few rules to get started. To get to the rules, you need to go to **Design** → **Rules** as shown in Figure 25. Note that usually, design rules should be set before starting the PCB, but we are doing them now because I didn't want you to worry about them when you were learning the basics of creating a layout.

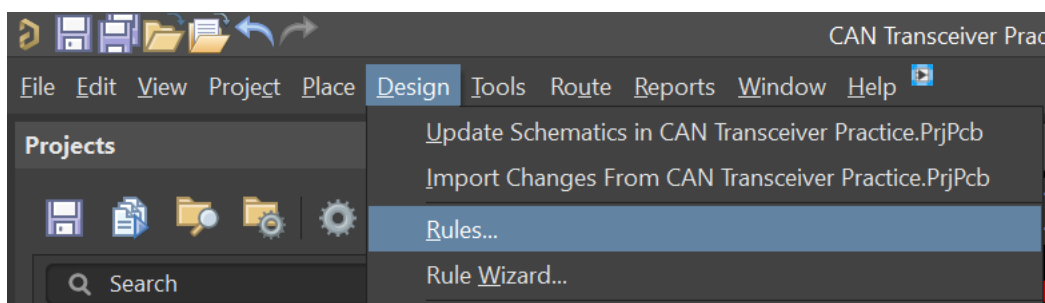


Figure 25: Screenshot of steps to get to the rules menu

The following are three rules you will want to update.

1. The default trace width often does not give you the full range of trace widths that you will want. When you are in the rules menu, go to **Routing** → **Width** as shown in Figure 26 and set the minimum trace width to 5 mil and the maximum trace width to 20 mil for each layer.

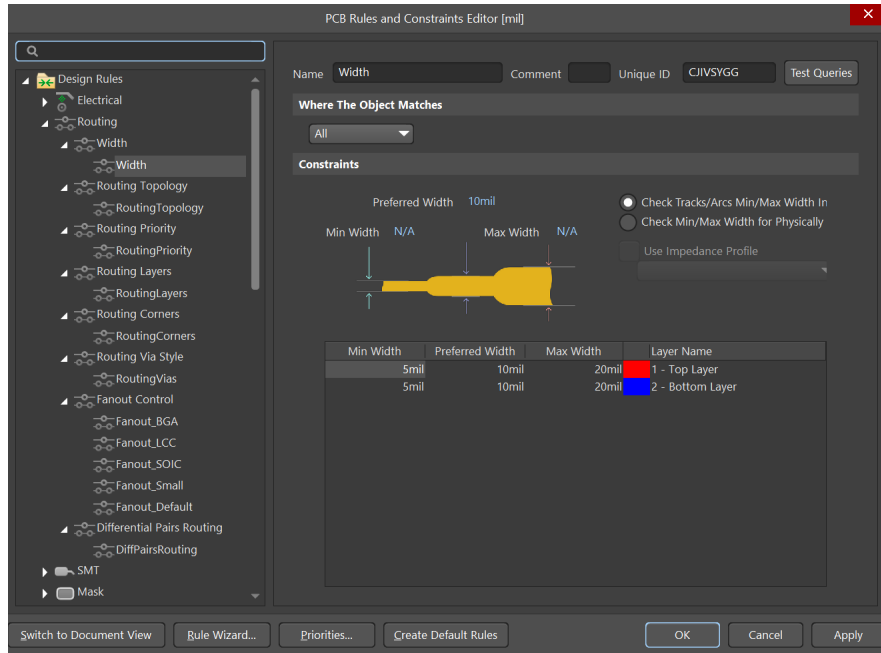


Figure 26: Screenshot of trace width rule settings

2. The default polygon connect style is with thermal reliefs, which look like little tabs that attach to the pads when there is a polygon connected. In general, it is nice to have thermal reliefs when you have a very large board with a ground plane because when you solder, the ground plane will take heat away from your soldering iron and make it impossible to melt the solder if you don't have thermal reliefs. However, for small boards like this one, it's not a big deal (and I think it helps to make a more reliable connection to the pad for current to pass through, but whether or not you use thermal reliefs is up to the requirements of your design). To change the polygon connect style, go into the rules menu then go to **Plane** → **Polygon Connect Style** → **PolygonConnect** (or make a new PolygonConnect rule if there isn't one called PolygonConnect already). Using the Connect Style dropdown, select **Direct Connect**. Figure 27 shows how to set the polygon connect style to direct connect.
3. Soon we will be adding polygon pours to your PCB. It is best practice to make sure that the polygons are not directly against the edge of the boards, where they risk damage. I like to have mine at least 30 mil away from each edge. To set this in the rules, go into the rules menu then scroll down on the left side of the menu to **Board Outline Clearance** and make a new rule (by clicking New Rule at the bottom), as shown in Figure 28. Double click the new rule to enter its settings. As shown in Figure 29, set the polygon outline edge clearance to 30 while leaving everything else at its default.

Make sure that you hit **Apply** and **OK** before closing out of the rules menu.

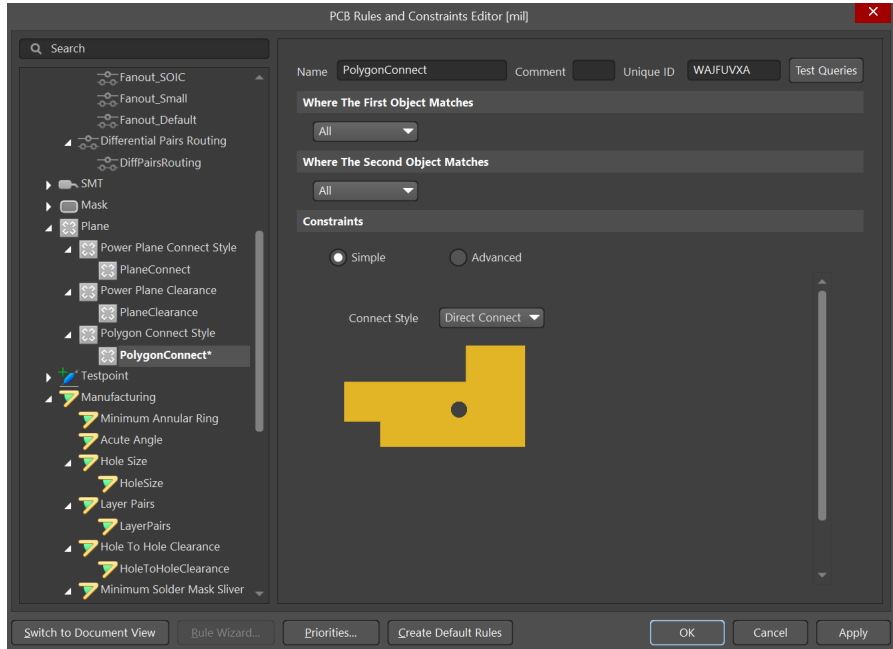


Figure 27: Screenshot of polygon connect rule settings

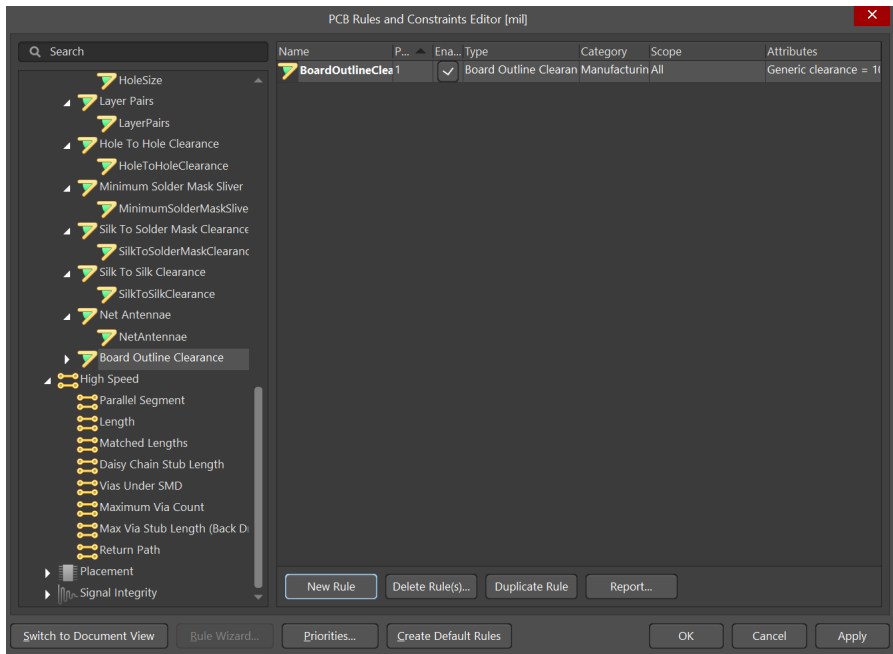


Figure 28: Screenshot of the creation of a new Board Outline Clearance rule

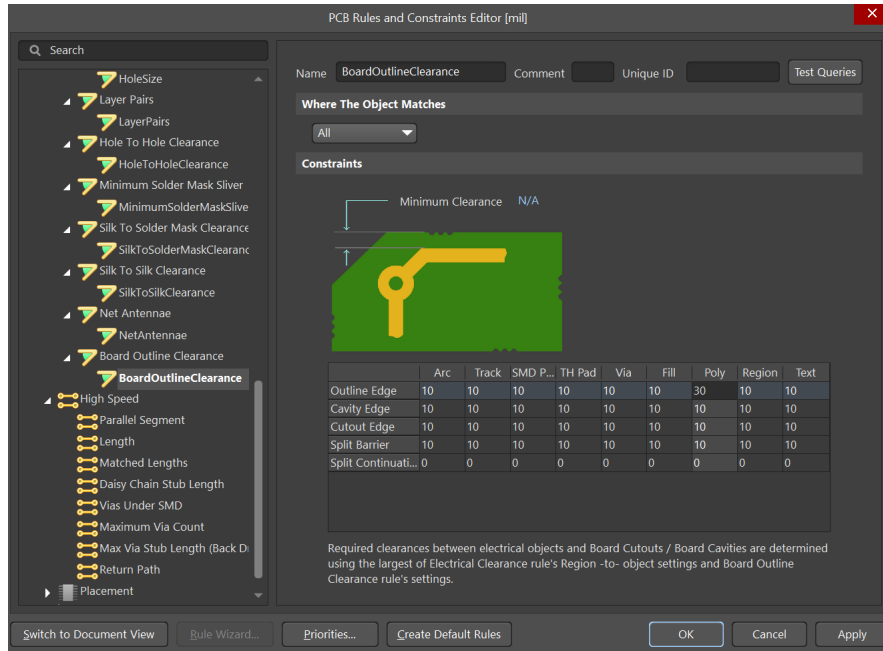


Figure 29: Screenshot of the board outline clearance rule settings

4.2 Routing Differential Traces

Finally we are ready to start routing! To begin, let's start with our trickiest and most important traces: the differential pair! To route a differential pair, all you have to do is go to the upper toolbar and right click the interactive routing tool to show other options. Then select the interactive differential pair routing tool, as shown in Figure 30.

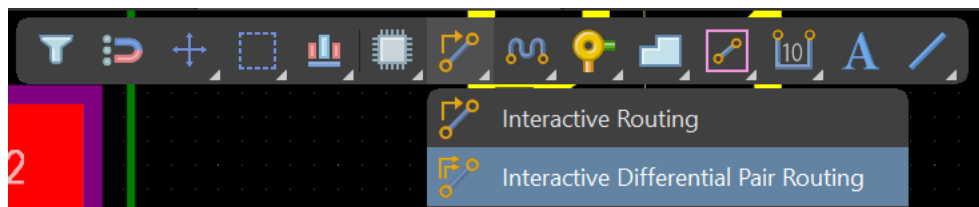


Figure 30: Screenshot of toolbar showing interactive differential pair routing tool

Once you have selected the right tool, go to any pad that is connected to the differential pair, and click the center of the pad. Begin moving your cursor over to where you would like to connect to another pad. Because this is a differential pair, the second trace should be automatically routing for you and should hopefully be approximately the same length (remember that length matching as closely as possible is important for differential pairs). Click on the center of the pad you would like to end on. Figure 31 shows you how to begin a differential pair trace. When routing differential pairs, try your best to make them symmetrical so that they are approximately the same length. Figure 32 is an example of what NOT to do. The problem is that the traces are very wiggly and are likely not length-matched as well as they would be if they were straight for more of the distance.

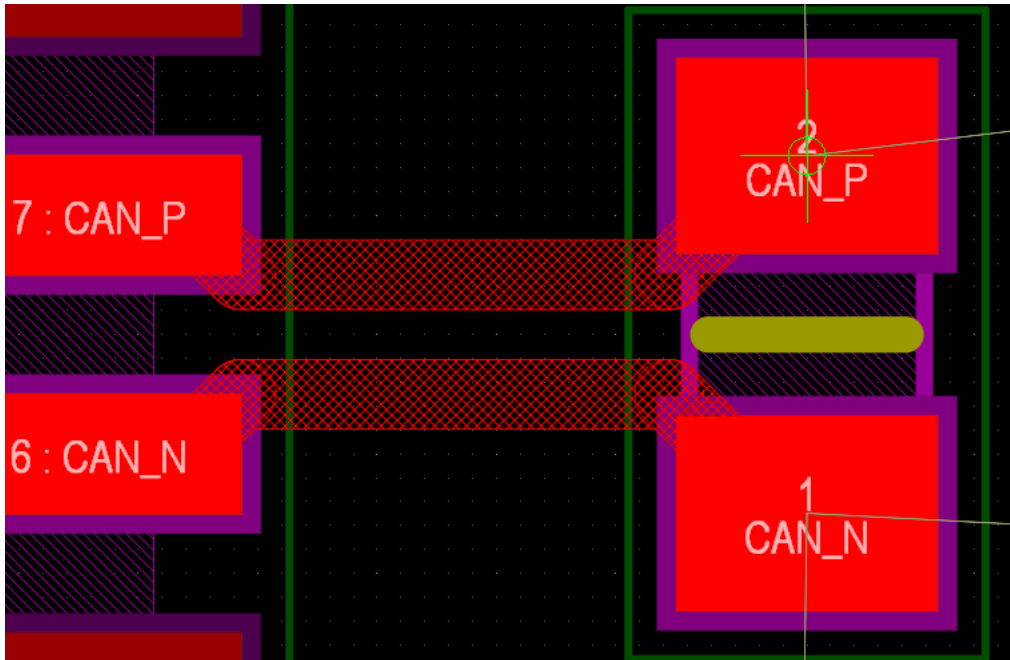


Figure 31: Screenshot showing the beginnings of routing a differential pair

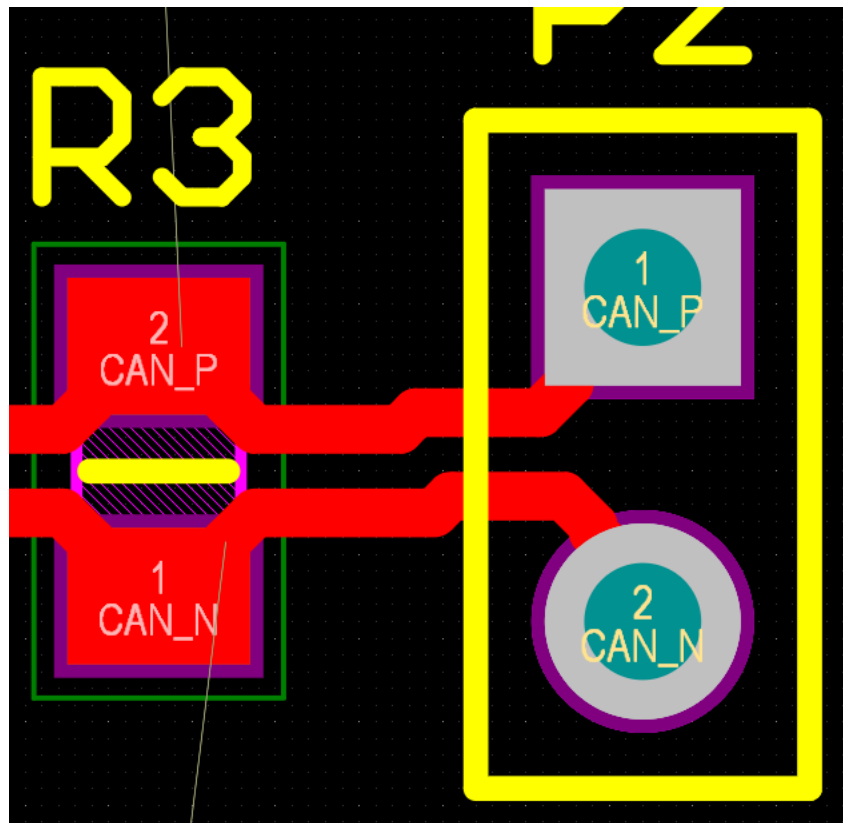


Figure 32: Screenshot showing what NOT to do when routing a differential pair

4.3 How to Route a Single Trace

After you have routed your differential traces, you can go ahead and route your regular old traces everywhere else (except for power and ground nets). Make sure you still do more important traces first (if you have clock traces, analog traces, etc.). To route a single trace, you go up to the same tool as before for the interactive differential pair routing, except you click the button for regular interactive routing, as shown in Figure 33. As an aside, you'll probably hear people talk about using an "auto-router" sometimes to do the routing for them. I am not going to teach you to use the auto-router because at the end of the day, it is a computer and you are a human. You know better than it does where your important traces need to be. Trust your own routing!



Figure 33: Screenshot showing the interactive routing tool

Just like the differential routing, to start a trace, simply click the center of a pad and then move your cursor to where you would like the end of the trace to go. Note that you can always place a trace and then drag it around to adjust its shape later. Figure 34 shows what it looks like to begin a trace that is not differential.

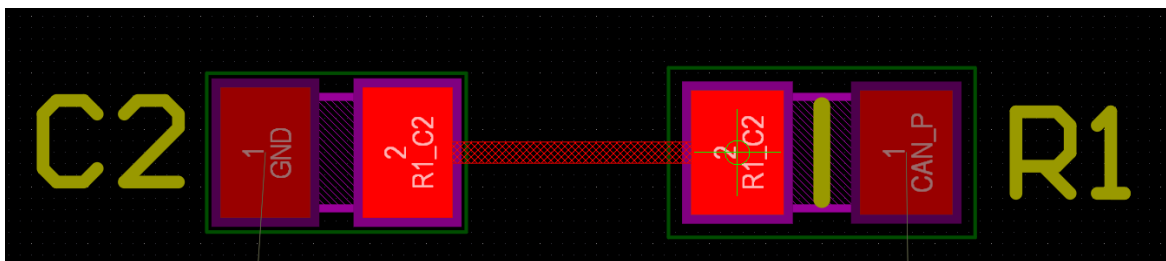


Figure 34: Screenshot of what it looks like to begin a regular trace

You may be wondering why I keep saying that you should click the centers of pads when you are doing your routing. It is best practice to have your traces connect to the centers of pads, so it is a good habit to get into. Another important best practice is to make sure your traces have only straight pieces or straight pieces connected with 45 degree angles. You never want to have a 90 degree angle in your trace if you can help it. When a signal is propagating through the trace, it has a much harder time when you have a 90 degree angle that makes it suddenly change direction as opposed to changing direction a bit more smoothly with a series of 45 degree angles.

4.4 Polygons

Once you have routed all traces (except for power and ground), you are ready to make your polygons. What is a polygon, you ask? A polygon (or polygon pour or copper pour) is basically a large region of copper that connects multiple pads on the board at once. It

can be used on pads that are nearby and on the same net other than power and ground, but power and ground polygons are probably used most often. All you have to do to make a polygon is go up to the top toolbar and select the polygon tool, as shown in Figure 35.



Figure 35: Screenshot showing polygon tool

Once you select the polygon tool, you can begin drawing your polygon around the pads you want to be covered. Click each vertex to place the corner. You can choose to make it a rectangle across the area that contains the pads (in which case copper won't be poured around other nets inside that don't match your target net) or you can make a more "abstract" shape that only covers the pads you want. It is up to you, but if you choose to be a bit more abstract, try to make the edges at least look nice. I find that the polygon tool often likes to make my polygon corners look really weird (like in Figure 36), so once you are done making the shape, feel free to go back and drag those corners into the place you'd actually like.

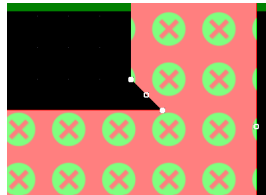


Figure 36: Screenshot of problematic polygon corner

Once you have drawn your polygon, double click it to open up its properties tab on the right panel. There are a few important settings to make sure you set up. First, you want to set the polygon's net, as shown in Figure 37. For example, if you are making a polygon for 5V pads, you will want the net to be 5V. Also verify that the polygon is on the layer you want. For this project, everything will be on the top layer, but in future projects you may have bottom layer polygons, in which case you would simply select bottom layer from the layer dropdown.

Next, if you scroll down in the polygon settings a bit, you will see something that looks like the image in Figure 38.

You should see four check boxes and a dropdown. The Remove Islands check box basically says, "If there is a little area of copper that is less than what is specified, I will just not pour copper there." This is fine to leave checked, but sometimes on really small pours you need to uncheck it so that the polygon actually shows up. The Remove Necks check box basically says, "If there is a little copper peninsula that is less than what is specified, I will just not pour copper there." Again, this is usually fine to leave checked, but if your pour is really small, you may want to uncheck it just to make sure the polygon actually shows up. Check the box below that says "Remove Dead Copper." This is useful because it does sort of what the other boxes do and keeps you from pouring copper that doesn't really do anything. Finally, the last box says "Optimal Void Rotation." I really don't know what

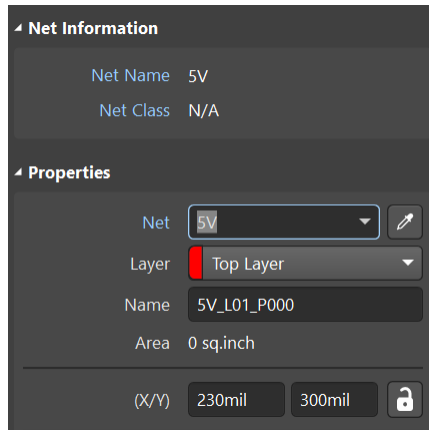


Figure 37: Screenshot showing where to adjust the polygon net and layer

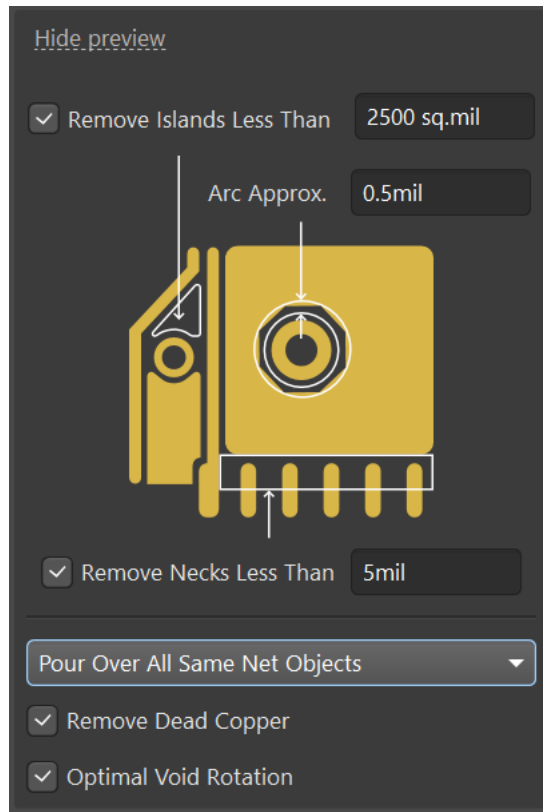


Figure 38: Screenshot of more polygon settings

that one does, so I just always leave it checked. In the dropdown, make sure you select **Pour Over All Same Net Objects**. This should be sufficient for most of your polygon needs.

Once you are finished in the settings, click **Repour** at the top of the polygon settings, or right click the polygon and select **Polygon Actions** → **Repour All** as shown in Figure 39. The polygon should then cover the space that you specified.

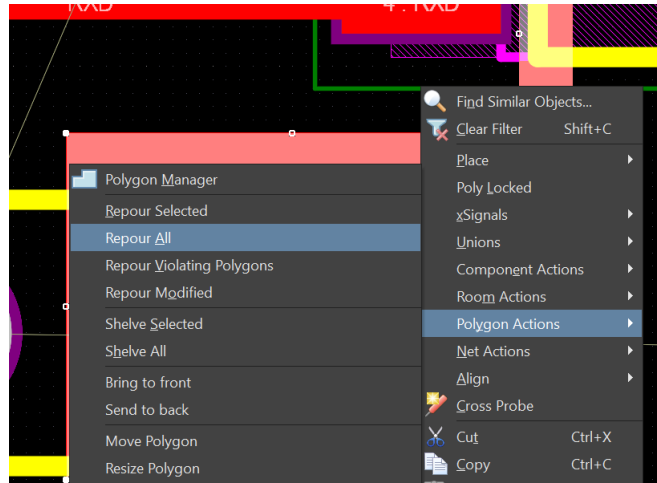


Figure 39: Screenshot showing how to repour polygon

4.5 Helpful Shortcuts and Tips

- While you are routing, if you press the Tab key on your keyboard, you pause the routing and open up the trace settings. From there, you can adjust the trace width (in the bottom right under "Width") if you so choose, and simply press the pause button in the center of the screen to resume routing with your new trace width. This process is shown in Figure 40.

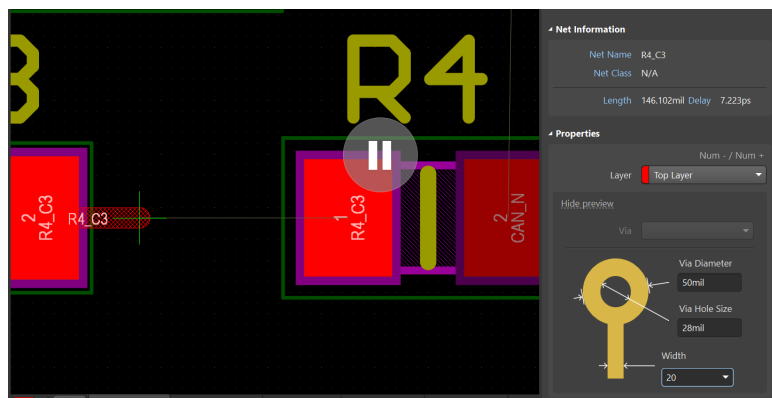


Figure 40: Screenshot showing how to adjust trace width while routing

- Feel free to get creative with your routing! Sometimes it is possible to even route under components, as shown in Figure 41. As you can see, we are still using our best practices of ending the traces in the centers of the pads and avoiding 90 degree angles, we just found additional routing space under the IC. Note that you can't do this with all ICs because sometimes they have pads beneath them, but if not, you can certainly use this technique. You can also route beneath resistors and capacitors if there is space. However, NEVER route beneath a crystal oscillator unless you are on a different layer (we'll see this later).

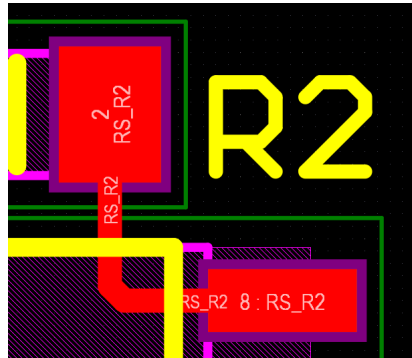


Figure 41: Screenshot showing some clever routing beneath an IC

- If you are ever trying to adjust traces but a polygon you poured is getting in the way, don't delete the polygon, just shelve it! To shelve a polygon (putting it away for later), simply right click, select **Polygon Actions** → **Shelve Selected**, as shown in Figure 42. This removes your polygon and holds it in the polygon manager until you are ready to restore it.

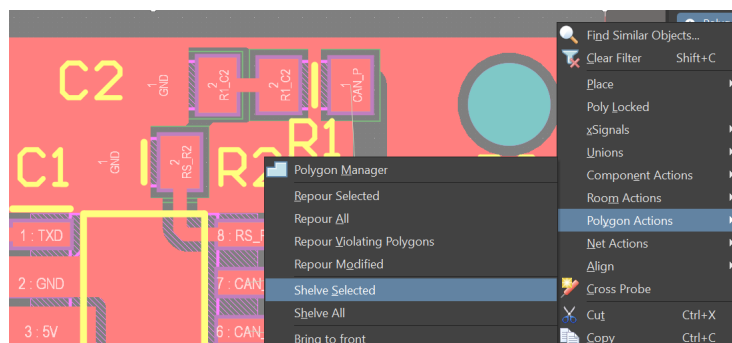


Figure 42: Screenshot showing how to shelve a polygon

To get the polygon back when you're ready, you can either go to **Tools** → **Polygon Pours** → **Polygon Manager** or I would recommend simply going to **Tools** → **Polygon Pours** → **Restore Shelved Polygons**, as shown in Figure 43. This will put your polygons back. Just remember to repour them! Figure 44 shows how to access the Polygon Manager if you choose the harder method.

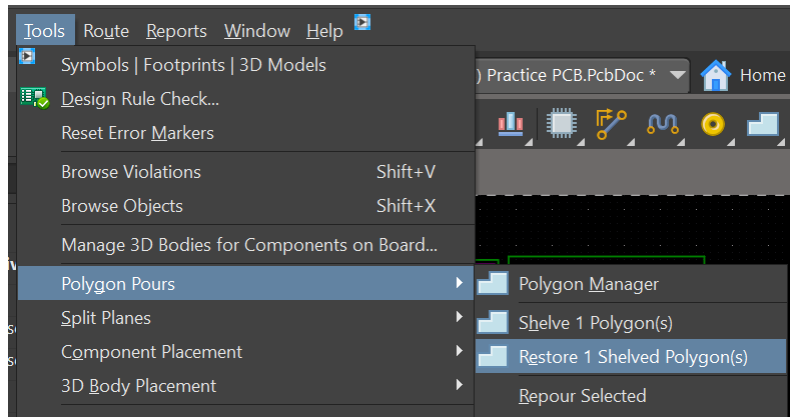


Figure 43: Screenshot showing how to restore polygon

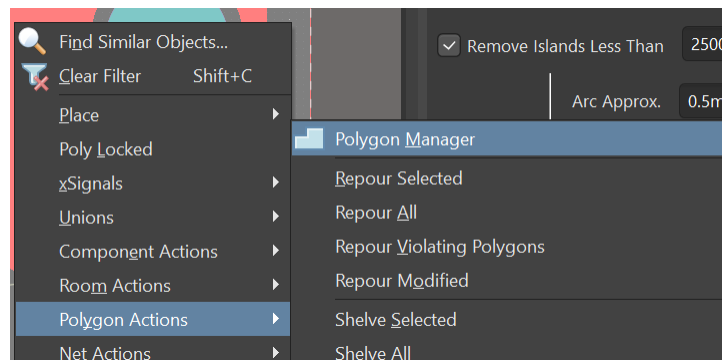


Figure 44: Screenshot showing how to access the polygon manager (if desired)

- You may notice that when you are placing polygons, it is important to have matching pads close to each other. This is an important layout consideration. For example, if you want to connect all the 5V pads to one polygon, you will want them to all be in relatively the same area. This is especially important to do with power and ground pads on two-layer boards like the one in this project.

4.6 Tasks

In this section, you are going to route your PCB. Using the techniques discussed above, complete the following steps. NOTE: PLEASE USE 15 MIL TRACE WIDTH FOR ALL TRACES ON THIS BOARD. This is not a set standard, but will work for this board. You may need to go into the rules and adjust the minimum trace width settings.

1. Update the trace width, polygon connect style, and board outline clearance rules as discussed in the above section.
2. Route the differential traces first. Remember that we route differential traces, clock traces, and other sensitive and time-dependent traces first because they are most important!

3. Route all other traces besides 5V and GND. Remember to try to get them to meet the components in the center of the pads and without snaking around at right angles!
4. Create a 5V polygon pour to cover the 5V pin of the IC, the 5V pad of the bypass capacitor, and the 5V pad of the power header. You may need to go into the rules and adjust the polygon connect style to direct connect (which is desired for this project).
5. Create a GND polygon pour across the rest of the board. Make sure that you have set the board outline clearance in your rules so that the polygon will end 30 mils from the edges of the board.

5 Silkscreen

Finally, you're done with your layout and routing! You're almost done! All you need to do now is adjust your silkscreen and add some finishing touches! What is silkscreen? Silkscreen (on the Top Overlay layer) appears as yellow text in 2D layout mode and white text on the actual board. It's the little letters and shapes that help you label things on the board.

5.1 Adding Useful Silkscreen

Whenever you are making a board, think about the end user. Are you the only one using this board? Probably not. You're an expert on how this board works, but another user may not be, so make sure that your silkscreen makes it clear how to use the board. Examples include labeling the power connector with a "5V" and "G" to show which pin is for power and which is for ground. You can also add a label naming the board (maybe call this one something like "CAN Board") so that other users don't have to struggle to figure out what the device is. The first thing I would suggest doing is cleaning up the silkscreen you already have for each component. You can select multiple texts and align or evenly distribute them (using the alignment tool from earlier), and you can rotate them to make them fit better on the board. **NOTE: Be consistent in your silkscreen placement!** It is okay to have some text horizontal and some vertical, but be consistent in what direction they face. Figure 45 shows an example of what good silkscreen orientation looks like. Figure 46 shows an example of messy silkscreen you should NOT have.

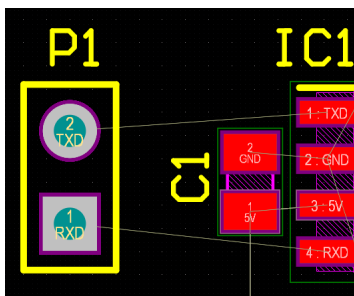


Figure 45: Screenshot of nice and consistent silkscreen orientations

There is one case where having silkscreen in multiple different orientations may be helpful. If you know which way the user will be looking at the board while plugging things in, you will want to have the text on the connectors facing in such a way that the text will be right-side up when it is being read. See the sample 3D model of the board at the beginning of this document for an example.

Once you have adjusted the silkscreen from the components (also, make sure that it isn't overlapping any pads or going off the edges of the boards, because then you won't be able to see it on your final product!), you can add the additional silkscreen we talked about earlier (the labels that will help the end user know what's going on, such as 5V and GND labels on power connectors and a board title naming what the board is). To add new text, go up to the upper toolbar and select the text tool, as shown in Figure 47.

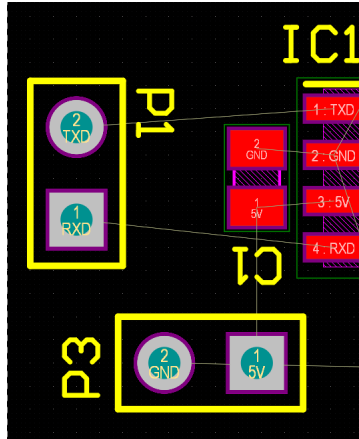


Figure 46: Screenshot of messy silkscreen orientations that you should AVOID



Figure 47: Screenshot showing the location of the text tool

Once you have selected the text tool, click anywhere on or around the board to place your text. You will notice that by default, the word "String" will appear in red. This is NOT what you want to leave it as! Double click the text to bring up the text properties in the right panel, as shown in Figure 48. In the field labeled "Text," replace the default text with the text you actually want. In the layer dropdown, make sure you choose the yellow **Top Overlay** layer. This step is EXTREMELY important because it makes sure that the text is actually a part of the silkscreen. If you leave the text in its default red form, that means that it will be on the top layer, and you will essentially be writing with copper traces! To change the text size, simply adjust the values in the text height and stroke width fields. I prefer to have my text with a **Text Height of 36 mil** and a **Stroke Width of 6 mil** because that fits the standards of JLCPCB.

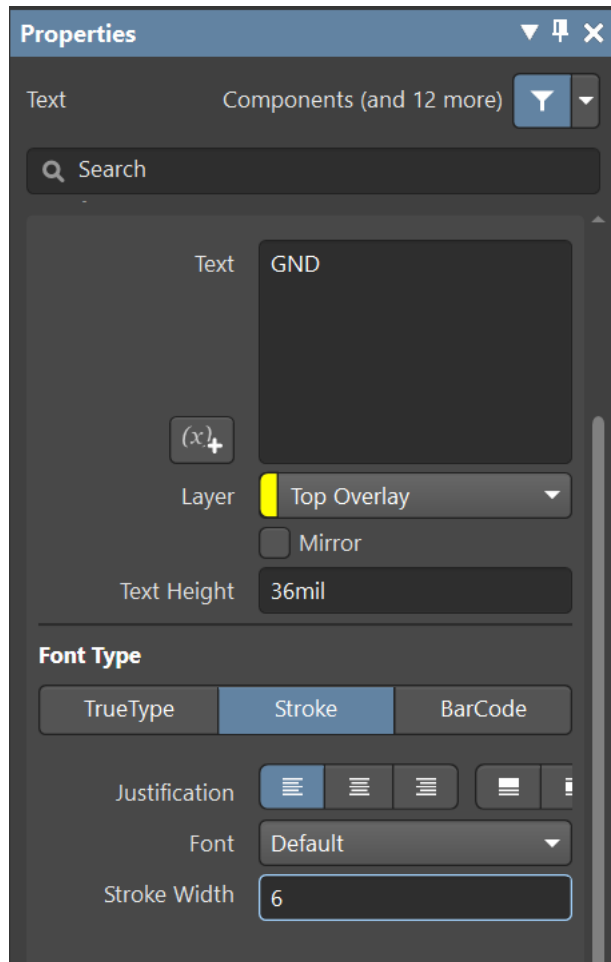


Figure 48: Screenshot showing the text settings

5.2 Tasks

In this section, you are going to add to the silkscreen on your PCB. Using the techniques discussed above, complete the following steps:

1. Add silkscreen labels for 'TX' and 'RX' on the left connector.
2. Add silkscreen labels for 'G' (or 'GND') and '5V' on the bottom connector.
3. Add silkscreen labels for 'H' (or 'CANH') and 'L' (or 'CANL') on the right connector that goes to the differential pair.
4. Add a board title of some sort (e.g. 'CAN Board') somewhere on the board.
5. Clean the silkscreen as you see fit (align it, make sure they are facing in logical directions, etc.).

6 Additional Details

At this point, you're basically done! There are just a few finishing touches that you can add to make your board even better. While there are lots of options here, we will only go over two of them for now. Later, we will discuss more options such as via stitching. Additionally, usually you will end a board design by running the DRC (design rule check), but that is not in the scope of this workshop or document. The purpose of this document is just to teach you the skills you need to make a PCB layout in Altium.

6.1 Adding Test Points

While you can often probe pins on the board while you are testing the physical board itself, sometimes it is useful to have little test pads/points that you can place your probe on. To make a simple test point, all you need to do is go up to the upper toolbar, right click the via tool to see more options, and select the pad tool. This process is shown in Figure 49.



Figure 49: Screenshot showing how to access the pad tool from the upper toolbar

Once you have selected the pad tool, click the location on the board where you would like to place the pad. You can attach pads to traces with smaller branching traces, or you can do what I would recommend and place it on a polygon somewhere. For example, you could create a ground test point by placing the pad on a ground pour. Double click on the test pad you have placed to bring up the pad settings. In Figure 50, we see the first few pad settings and how to select the pad layer. Make sure that the layer shown on the Layer dropdown matches the layer you would like the pad to live on. Don't leave it on multilayer. Next, in the Net field, shown in Figure 51, set it to the name of the net you are adding the pad to. For example, if you are putting a test point on a ground polygon, your net will be GND. The rest of the settings shown in Figures 50 and 51 can be left as their defaults.

If you scroll down a little farther in the pad settings, you will see where you can adjust the size, as shown in Figure 52. The X/Y sets the diameter. If you would like a circular pad, set both to the same value. Leave all other settings shown in Figure 52 as their defaults. When you are done adjusting the pad, be sure to repour all polygons.

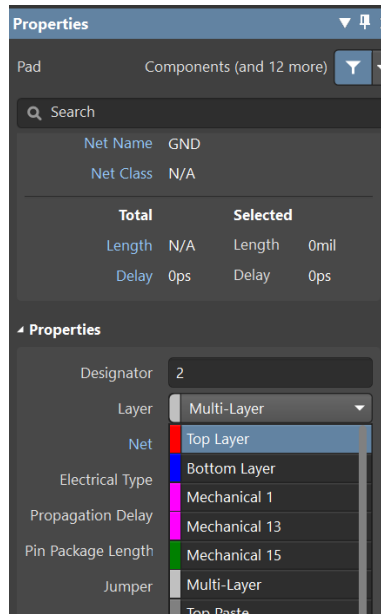


Figure 50: Screenshot showing pad layer settings

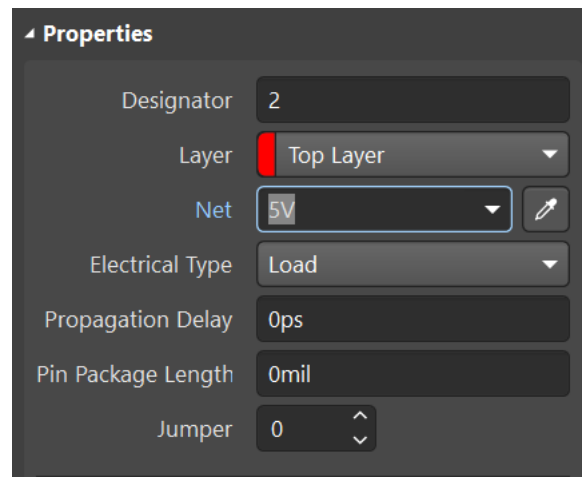


Figure 51: Settings showing how to set pad net

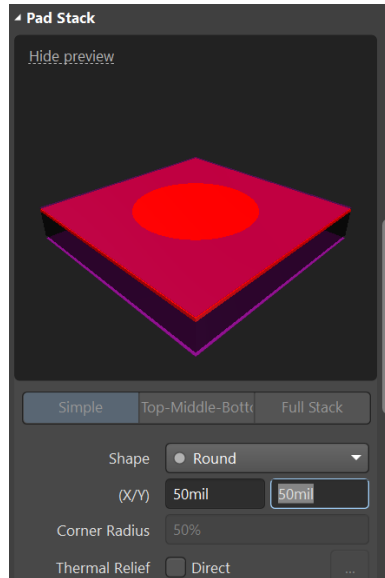


Figure 52: Screenshot showing how to adjust pad size settings

6.2 Adding Mounting Holes

Often, you will want to add mounting holes to your boards. You can use them to attach standoffs that help your board stand steady when you place it down on a lab bench, or you can use them for screws to attach to something like a robot. There are many standard sizes for mounting holes, but in this practice project, we will be using M2 size holes, which have an 86.6 mil hole size. Adding a mounting hole isn't any harder than adding a pad. To add a mounting hole, click the pad tool again, then place your pad where you want a mounting hole (usually in one of the corners of the board). Double click the pad to open its settings. You can set the pad's location to a specific spot by changing the X/Y location in its properties, as shown at the bottom of Figure 53. Note that this is NOT the same as the X/Y size of the pad, which is shown lower in the settings. I usually like to have my mounting holes 100 mil from the edges. So if I had a board that was 600 mil x 500 mil, the center of my upper right mounting hole would be at 500 mil horizontal and 400 mil vertical.

Once your mounting hole is in the location you want, make sure that the layer is set to **Multilayer**. We didn't want this before with the test points because we wanted those pads to be on the top layer for us to probe. However, for mounting holes, we want them to go through all the layers, so leave the layer setting on multilayer. Also, make sure that your net for the mounting hole is set to **No Net**. This is sufficient for our purposes. Finally, scroll down in the settings until you reach the section about pad/mounting hole size, as shown in Figure 54. What really matters here is the hole size. Set the X/Y size to something small, like 10 mil in each direction, so that the hole is much larger than the rest of the pad (we don't want to see an actual pad). In the **Hole Size** field, type in the hole size you would like for your mounting hole. For M2, the value is 86.6 mil. You should now have a nice mounting hole! You can take a look at the sample 3D layout at the beginning of this document for an example of what mounting holes should look like on a board.

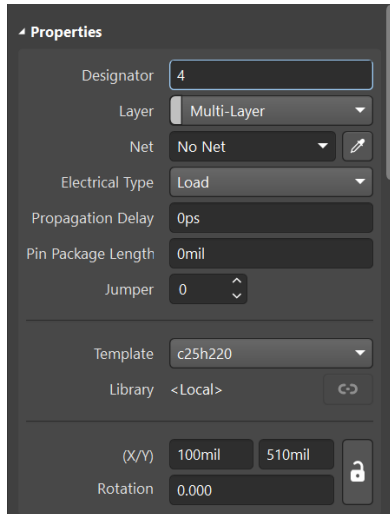


Figure 53: Screenshot showing how to access pad location settings

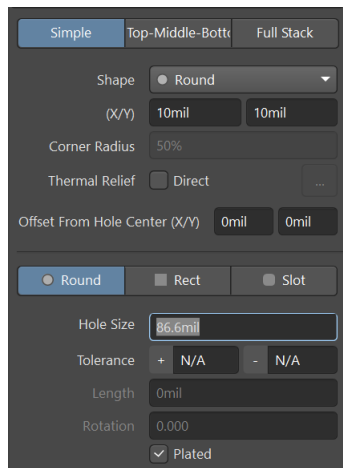


Figure 54: Screenshot showing how to adjust mounting hole size in the pad settings

6.3 Tasks

In this section, you are going to practice adding test points and mounting holes to your board. Using the techniques discussed above, complete the following steps:

1. Add a test point to GND somewhere on the board. Make the pad 50 mil in diameter and make sure it is labeled.
2. Add four M2 mounting holes (86.6 mil hole size), one to each corner of the board. Try to make them about 100 mil from each edge in the corner (e.g. the upper right hole will be 100 mil below the top and 100 mil left of the side, so it will be at the coordinates 840, 510).

7 Bonus Material: Multilayer PCB Techniques

Now that you have two-layer PCB design down, I want to teach you some techniques that you can apply to multilayer PCB design!

7.1 The Layer Stack

When making a one- or two-layer board, both layers are signal (routing) layers, meaning you can route traces on the top or bottom of the board. With a multilayer board, we add additional copper layers into the board so that you have more room to route (not just on the top and bottom) and/or so you can connect to power and ground more effectively. There are many different layer configurations that are used for different purposes, and boards can have anywhere from 1 layer to dozens of layers and beyond. Adding more layers greatly increases the cost though, so for most hobby applications, I recommend sticking with either 2-layer or 4-layer PCBs. Otherwise things would get way too expensive.

7.2 Adjusting the Stackup

The stackup that we will be using is (from top to bottom): Signal, Ground, Power, Signal. This means that you will be able to route on the top and bottom layers and then connect to the second layer for ground and the third layer for power. This is the stackup that has been used in all 4-layer boards I have made before, but it is not the only option. How do we tell Altium this is the stackup we want? We need to go into the Layer Stack Manager! Go to the Layer Stack Manager by going to **Design** → **Layer Stack Manager** as shown in Figure 55.

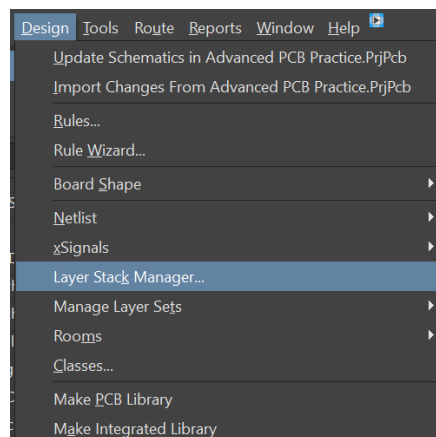


Figure 55: Screenshot showing how to access the Layer Stack Manager

Once the Layer Stack Manager opens, you should see the default 2-layer stackup (2-layer as in 2 copper layers—solder mask, prepreg, etc. doesn't count), as shown in Figure 56. We need to add two new layers. We can do this in one step because Altium will mirror the layers for us by default. Right click the first copper layer where it says "Top Layer"

#	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 1	FR-4	Dielectric		12.6mil	4.8	
2	Bottom Layer		Signal	1oz	1.4mil		
	Bottom Solder	Solder Resist	Solder Mask		0.4mil	3.5	
	Bottom Overlay		Overlay				

Figure 56: Default Layer Stack

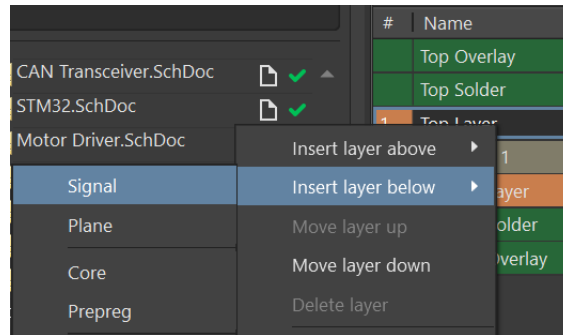


Figure 57: Screenshot showing how to add an additional layer (or layers) to the layer stack

and select **Insert Layer Below** → **Signal**, as shown in Figure 57. This creates new signal layers. When this is complete, you should see something like what is shown in Figure 58. Now, rename Layer 1 "GND" and Layer 2 "Power" (click the box that says the layer name and then type in what you want). Your final stackup should match Figure 59.

#	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	Layer 1	CF-004	Plane	1oz	1.378mil		
	Dielectric 1	FR-4	Dielectric		12.6mil	4.8	
3	Layer 2	CF-004	Plane	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				

Figure 58: Layer stack updated to have four layers

#	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				

Figure 59: Final stackup for our multilayer board

You may notice that there are a ton of extra layers and values that we haven't touched. (Also note that the copper layers are the only ones we count when describing how many layers a PCB is). The defaults are fine in this case, so we don't have to change anything else. The green layers represent the top and bottom overlay and solder mask. The solder mask is the material (that is often green) that goes on the outside of the board to repel solder (so the solder goes only on the pads). Top/Bottom Overlay is the layer name for silkscreen layers on the top and/or bottom of the board. In between the copper layers are non-conducting materials such as FR-4 (a common PCB material). You often don't have to change the values of the dielectric or thickness for these layers because the places that can fabricate the boards have their own stock of the materials, so you basically have to go with whatever they have.

7.3 Using Multiple Layers

Alright, now that we have set up the layer stack (make sure you save the stackup when you're done adjusting it!), what do we do with it? Basically the same thing as always, but with just a few adjustments!

7.3.1 Top Layer

The top layer will still be where you can do a significant portion of your routing, and will often be where most of your components are. You can change layers in the component properties (double click on a component then go to the Layer field in the properties window that pops up) to put your component on the bottom side of the board if you want. While this is totally valid and common in PCB design as a whole, I recommend not adding components to the bottom side of the board for projects you are soldering by hand or reflow oven because it will make it significantly more difficult to solder in house (unless it's like one or two easy things on the bottom). Typically we would like to try to use a reflow oven to solder tiny surface mount components to the board, but if the board has components on both sides, one side will have to be soldered by hand so the reflow oven doesn't desolder one side while trying to solder the other.

7.3.2 Bottom Layer

Like the top layer, the bottom layer is a great place to route traces (and place components if you have to). Feeling trapped on the top layer? Bring the trace to the bottom layer to get around any "traffic" then head back up to the top layer to complete the trace! To switch layers on the board, you will need to place something called a via. A via is basically just a little plated hole through the board that acts as a tunnel for the trace. We only use through hole vias (that connect from top to bottom and have little conductive pads to connect to internal layers as needed) on our boards here, but it is also possible to have blind vias that connect from the top or bottom to an internal layer or buried vias that connect internal layers (this is just not necessary for our application and JLCPCB, the company that I usually use to make my boards, can't do blind or buried vias). I'll talk about vias in more detail later in this document. Once you place the via, go to the bottom of the screen and select the layer you would like (Top, Bottom, etc.), as shown in Figure 60. This will switch you over to the new layer so you can route and make polygons as usual. You'll probably notice that your traces will be a different color. This is totally normal and just lets you see what is a part of a different layer. If you start getting distracted by all the colors on the board across all the layers, you can use the shortcut **Shift+S** to have Altium focus on one layer at a time. Hitting Shift+S two more times will return you to the original layer view.

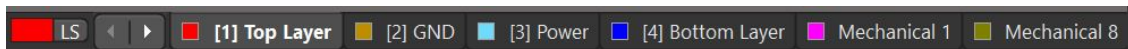


Figure 60: Available layers shown at the bottom of the Altium window. Selecting a different layer takes you to that layer.

7.3.3 Ground Layer

The ground layer in our stackup is directly below the top layer and will basically be a full layer of copper. I recommend just putting a GND polygon across the entire ground layer. If this gets too distracting toward the beginning of your routing (because you'll have

to keep repouring), you can either add the ground polygon at the end (all the GND vias should connect to it automatically), or you can shelve it until you're ready to add it back in. While you're routing, anything that is connected to ground will connect to this ground layer through a via somewhere. I would recommend not connecting anything to ground through a trace if you can because it will look better if you use polygons. What you can do is create a ground pour (polygon) on the top of the board, on the bottom of the board, and on the ground layer. Then you can add something called via stitching (basically just an array of vias) to connect all these ground pours to each other. We'll talk about via stitching in a later section of this document. If you end up not adding a ground pour to the top and bottom of the board, you don't need via stitching. Instead, you can just connect GND pins using traces to vias connecting to the GND layer. Note that it depends on your design requirements whether or not you should use a ground pour on top and bottom and add via stitching. Sometimes it serves to reduce electromagnetic interference or noise, but other times it could add harmful parasitics. Another note: remember our clock? I said you should never route traces beneath a crystal oscillator, but it is okay to do so if you have a GND plane in between them. So if you have a crystal oscillator on the top layer, a GND layer below it, and traces on the bottom layer, that is okay. The GND layer provides the necessary shielding.

7.3.4 Power Layer

The power layer is below the GND layer and above the bottom layer in our stackup. The power layer is a lot like the ground layer in that you will just be adding large polygons and connecting vias to it from other layers. However, the power layer will likely have more than one polygon (whereas the ground layer only has one ground polygon). Based on your voltage requirements, you can place power polygons in convenient places on the power layer. Take a look at Figure 61, the power layer of a board I have worked on, for an example. The polygon in the bottom left is 5V, the polygon in the middle is 3V3, and the polygon spanning the left and right sides is 6V. Let's take the 5V polygon as an example. I knew the CAN transceiver was in the bottom left corner and that it needed 5V, so I placed the 5V polygon below it and made sure the screw terminal that was bringing in the 5V would also be right over the 5V polygon. Placing your power polygons wisely allows you to make things a lot easier while you are connecting your components to power. One thing I'll note that is important is that you need to make sure you leave a little bit of space between your power polygons. You don't want any shorts!

7.4 Using Vias

7.4.1 Adding Vias

As I've mentioned before, vias are little holes in the board that allow you to connect to different layers. To add a via, first go to the top toolbar and select the via tool, as shown in Figure 62.

Next, place the via on the board. Normally this will be while you're routing, but you could also place one on a polygon if you want. For example, you could be routing on the top

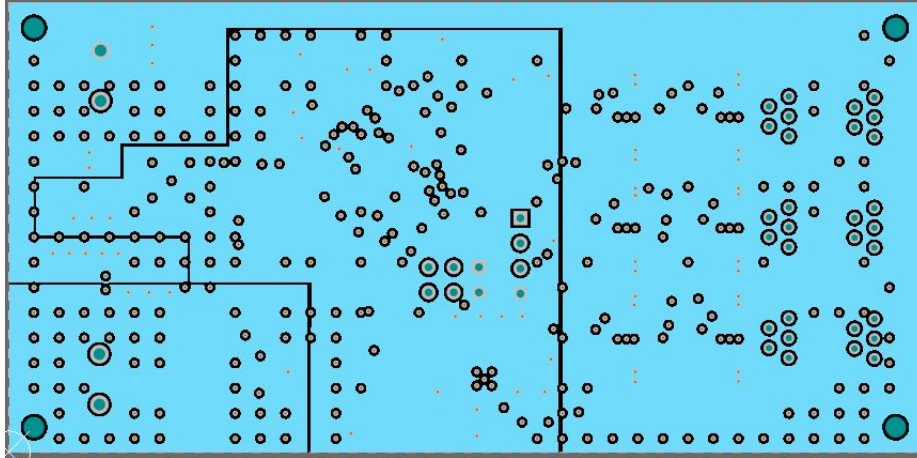


Figure 61: Example of a power layer on a different board with polygons for different voltage levels. The polygon in the bottom left is 5V, the polygon in the middle is 3V3, and the polygon spanning the left and right sides is 6V.



Figure 62: Via tool on the top toolbar

layer when you reach an obstruction (another top layer trace, a large component, etc.). To get around this, you can stop the trace, place a via, go to the bottom layer, route on the bottom layer until you can come back up to the top, place another via, then finish the route on the top layer. When you place the via, it will often be nowhere near the size you want, and sometimes it will even connect to the wrong net. You can double click the via to open up its properties, as shown in Figures 63 and 64. Figure 63 shows the first set of via settings. All you have to change is the Net field. Make sure you set it to the net you are routing (or the GND/PWR net you want to connect to). Figure 64 shows the second set of via settings. There are tons of different sizes of vias you can have, but for this example, please use vias with a diameter of 24 mil and a hole size of 12 mil. This also aligns with JLCPCB's capabilities. Under the Solder Mask Expansion section, set it to manual, then take a look at the boxes next to the word "Tented." If a via is tented, it means that it gets covered up by solder mask. It makes it a lot more aesthetically pleasing, so I recommend you use tenting on the top and bottom wherever you can (so check the tenting boxes). One example of when you **would not** want to use via tenting is when you have a tight layout and want to add test points (maybe around an IC). There often won't be a good amount of space for test pads, but non-tented vias (at least non-tented on the top) can act as test points as long as you know what net it is connected to.

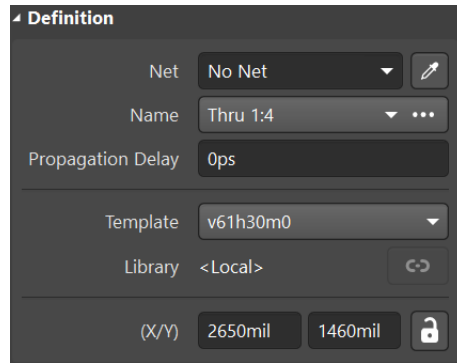


Figure 63: First set of via settings

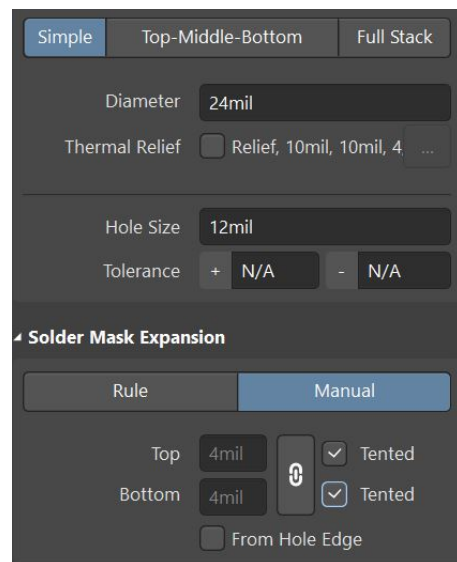


Figure 64: Second set of via settings

7.4.2 Adding Via Stitching

On boards with very high frequency and high noise risks (often these are RF boards), it is sometimes a good idea to add via stitching (again consider the design requirements first). You can add a ground pour on the top and bottom layers and use via stitching to connect them together and to the GND layer. This is great for reducing electromagnetic interference (EMI) and other noise concerns, but is often overkill for most non-RF boards. It's a good thing to know though, so it's good to practice. Make sure that before you attempt to add via stitching, you shelve the power layer polygons. If you don't do this, the via stitching doesn't like to work for whatever reason. To add via stitching (after you have all your ground pours on the top, bottom, and GND layers), go to **Tools** → **Via Stitching/Shielding** → **Add Stitching to Net**. A new settings window will pop up as shown in Figure 65 that allows you to change the via stitching properties. You can leave almost everything at its default and play with the spacing later as needed, but what is most important is the hole size, diameter, net, and tenting. Make sure the hole size is 12

mil and the diameter is 24 mil (or really just match these sizes with the dimensions of the rest of your vias). Make sure you set the net to GND so the vias connect to each GND pour. Finally, force complete tenting on top and bottom (you don't want so many non-tented vias—it would be an eyesore!). Make sure you repour all your polygons (including the restoring and repouring the power polygons) when you're done! If you want to remove the via stitching, you can simply go to **Tools** → **Via Stitching/Shielding** → **Remove Via Stitching Group** then click one of the stitching vias to remove them all.

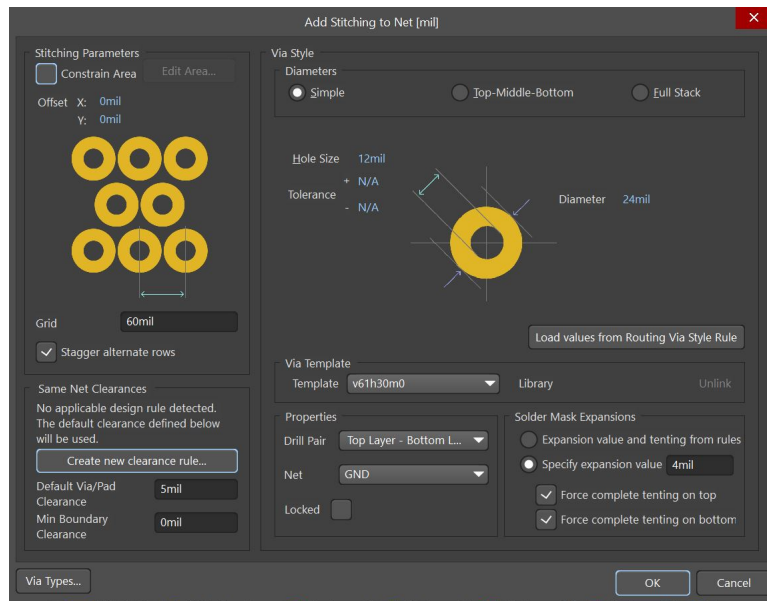


Figure 65: Via stitching settings

8 Conclusion

Congratulations! At this point, you should now have a PCB for your CAN transceiver project as well as some skills you can apply if you want to make two-layer or multilayer PCBs! You should pat yourself on the back, because Altium can be tricky, but you really have come incredibly far. Thank you for attending my workshop, and if you ever have any questions about Altium, don't hesitate to reach out at emarshall33@gatech.edu!