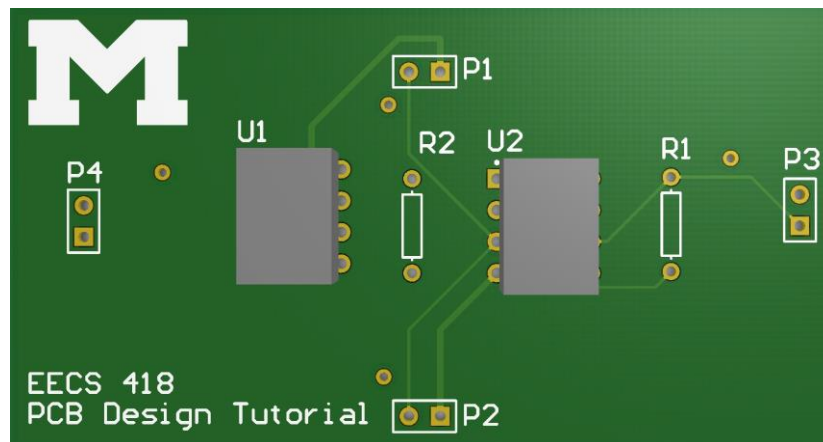


University of Michigan

EECS 418 - Power Electronics
Fall 2018

An Introduction to PCB Design Using Altium Designer



Prepared by Ananth Kumar Mohanram from EECS 418 Class of 2016.

1 Introduction

Altium is one of the most popularly used electrical CAD tools used by engineers and designers in the electronic circuit design field. This tutorial is designed to serve as a primer to PCB design using Altium Designer. The general procedure to design a PCB is as follows:

1. Develop a "Pen and Paper" design of your circuit. This includes identifying a circuit design, necessary components and assigning values to the components. This step is important as you will need to know the form factor of your component; its size and shape to design your PCB.
2. Simulate your circuit using a circuit simulation tool. Identify currents and voltages at each stage of the network. This will help you define the width of the traces (copper wires on your PCB) that carry current.
3. Develop a schematic of the circuit on Altium Designer. This is essentially a software version of the pen and paper circuit. Developing a schematic includes assigning a footprint (a map of how your component would look on the PCB) to components.
4. Using the Schematic, generate a PCB file. Arrange components considering electrical and physical factors (board shape and size). Verify that your PCB design passes basic design rule tests.
5. Generate Gerber files that the PCB manufacturer uses to fabricate the PCB.

2 Getting Started

1. Launch Altium Designer on a CAEN Computer using the S2 hub software launcher.
2. Altium works like a relational database. You must first create a project, and later add your Circuit Schematic and PCB files to this project. To create a new project: File → New → Project → PCB Project.
3. Once the project is created, add Schematic and PCB files to the project. To add Schematic: File → New → Schematic. To add a PCB to the project: File → New → PCB.
4. Save the Project: Right Click on the Project you just created (visible in the "Projects" window), and save your project using the "Save Project As.." option. Save the PCB document (.PcbDoc), the Schematic Document (.SchDoc) and the project (.PrjPcb) with suitable names.

Ensure that your project is saved regularly during the course of your design. (Use the File → Save All option)

3 Creating an Altium Schematic

1. Identify the "Libraries" tab. This is usually on the extreme right hand side of the Altium window. The library contains Miscellaneous devices (resistors, OPAMPS, Integrated Circuits etc.), Connectors and various components that can be added to the Schematic.
2. Find the Component you want to add in the Library Browser, and hit the "Place Component" button in the library window. Alternatively, you can also drag your component from the library window to the schematic. To place the component on the schematic, left click on the desired location on the schematic sheet. Once you place the component, you will find that Altium creates a duplicate of the component (attached to the mouse pointer) that you just placed. This helps to place multiple instances of the same component. To stop placing multiple components, right click or use the Esc key.
3. To rotate your component on the Schematic, click on the component, and use the Space key to rotate. To zoom in and out of the schematic use the Ctrl key along with the scroll bar on the mouse.
4. Continue placing all necessary components on the schematic. Most components, such as Resistors, Capacitors, Diodes, MOSFETS etc. can be found in the Miscellaneous Components library. Components that cannot be found on the Altium library must be added by the user. This will be discussed in a later section.
5. If you have multiple instances of the same component (for example, multiple resistors), you will have to assign a different designator for each of the components. Multiple instances of a particular component which have not been assigned different designators will be underlined in red.

6. To remove these red lines, right click on the component, select "Properties.." from the drop down menu. In the properties window, use the "Designator" field to assign a unique designator for each of the components (for example, if you have multiple resistors, assign designators as R1, R2, R3.....).
7. To change the value of a component, right click on the component, select "Properties.." from the drop down menu. In the properties window, use the "Value" field found in the "Parameters" sub-window to assign a value to the component.
8. To wire components together, use the wire nets button (marked in Figure 1). Once this tool is selected, the mouse pointer will change to a "Plus" sign, with a smaller "cross" sign in center (Shown in Figure 2). Place the pointer at the end of the component, such that the smaller cross sign turns red. Once the smaller cross turns red, left click and drag the wire to the next component you intend to connect to. Right click to exit the wiring tool.
9. To add a ground connection, use the ground button the tool bar (Shown in Figure 1). The procedure to add power and output connections to the schematic is specified in subsequent sections.



Figure 1: Ground Connection, VCC Power Port and Wire Net tools are highlighted in Red, Green and Blue Boxes

4 Adding Power Input and Output Connections to the Schematic

1. In the "Libraries" window, navigate to the "Miscellaneous Connectors" sub-library using the drop down menu and select the "Header 2" Component.
2. It is possible to rename the Header Component by right clicking the component → "Properties..." and changing the "Designator" and "Comment" fields.
3. To add power input to the circuit, select the "VCC Power Port" button from the toolbar. Before you place the "VCC Power Port" on the schematic (and while the VCC Power Port is still attached to the mouse pointer), hit the Tab key. This will allow you to label the VCC Power Port. Under the "NET" field specify a name for the VCC Power Port (for example: +12VDC). Please note that Altium is case-sensitive.
4. Connect the VCC power port to the top lead of the Header and a ground port to the bottom of the header connector. Note that for a particular input voltage, you will have to create only one header.
5. If different parts of the circuit require the same power, simply add another VCC power port and use the same net name. This will ensure that they are connected together to their single common header in the final PCB design. (For example, if 2 different sections of my circuit require a 12V input, I will add 2 VCC power ports and label them both as +12VDC to ensure that they are linked)
6. The header port and connections to the header port are shown in the subsequent figure.

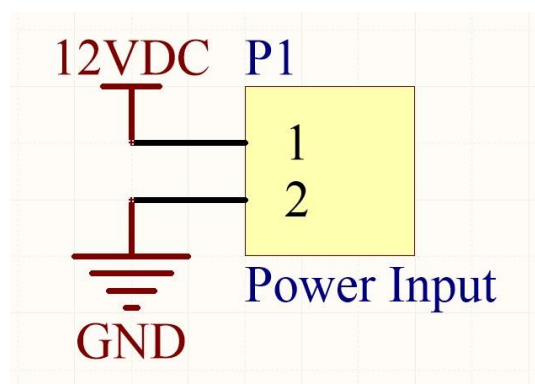


Figure 2: Header configured for an Input Power Connection

- Output ports are also defined using headers. Similar to the input ports, connect the output terminals across the upper and lower terminals of the header.
- Occasionally, it may be necessary to configure an input port that is connected to an external signal whose value may not be well defined. In such a case it is possible to create an input port using the header component connected across the input terminals (without adding a VCC Power Port), similar to the output port. This is also shown in the figure below.

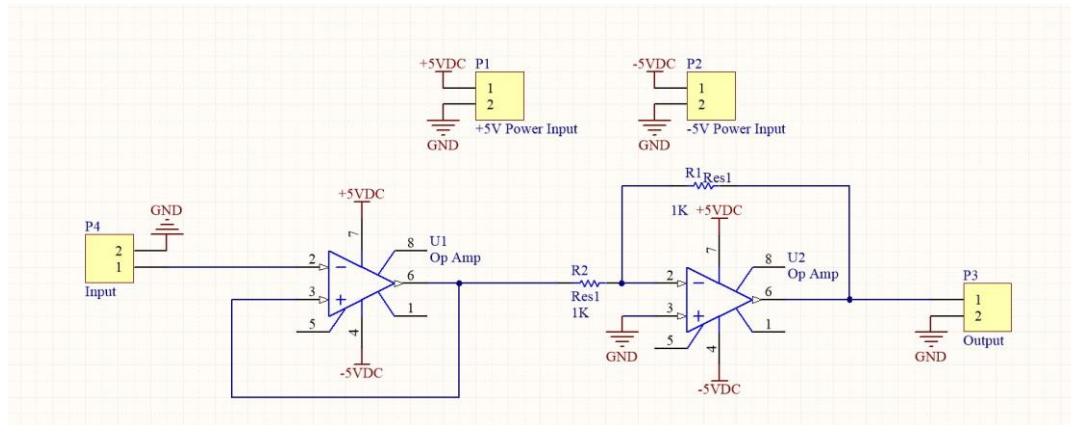


Figure 3: Exemplar Circuit showing Ground, Power Input, Circuit Input and Output Connections

In Figure 3, it can be seen that both OPAMPs require a +5V and -5V power Input. However, it is sufficient to define only 2 headers, one for +5V and another for -5V. The power connections to the OPAMP can be made using VCC power ports having the same net name/label as the VCC power ports connected to the +5V and -5V headers. Having the same net name/label will ensure that they are connected together in the circuit.

Circuit input and Output headers can also be seen.

5 Adding User-Defined Components to the Library

- It is possible that a component used in the circuit may not exist in the pre-defined library. In such cases, the user will have to add the component to the library.
- To add a component, navigate to "Schematic Library" via File → New → Library → Schematic Library. Doing this will open the schematic library window.
- The schematic library allows the user to add any component required. For the purposes of this tutorial, we will consider that the LM555 Integrated Circuit is to be added.
- Before adding a component, it is necessary to be aware of the shape of the component, pin description and mechanical packaging.
- From the datasheet of the LM555, it can be seen that the IC is a rectangular 8 pin Integrated Circuit and is available in the "8 Pin Dual-Inline-Package (DIP)" mechanical package.
- In the Schematic Library Window, navigate to the "SCH Library" pane (on the left of the screen) and click on "Component_1". This is the default name that Altium gives to the new component.
- Once "Component_1" is highlighted, click on the Edit button on the SCH Library Pane (marked in the figure below).

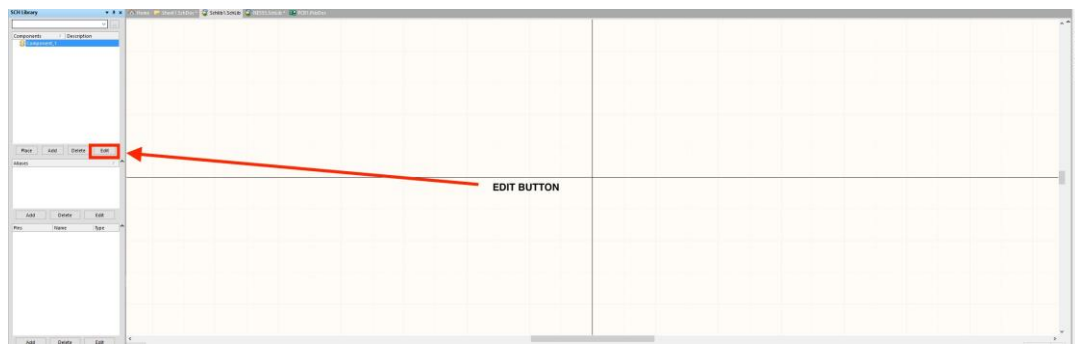


Figure 4: Schematic Library Window

8. Clicking the Edit button will open the Library Component Properties window (shown below). In the "Default Designator" field, enter U1 (most ICs use "U" as a designator). In the "Symbol Reference" field enter "LM555" to identify the IC and click OK.

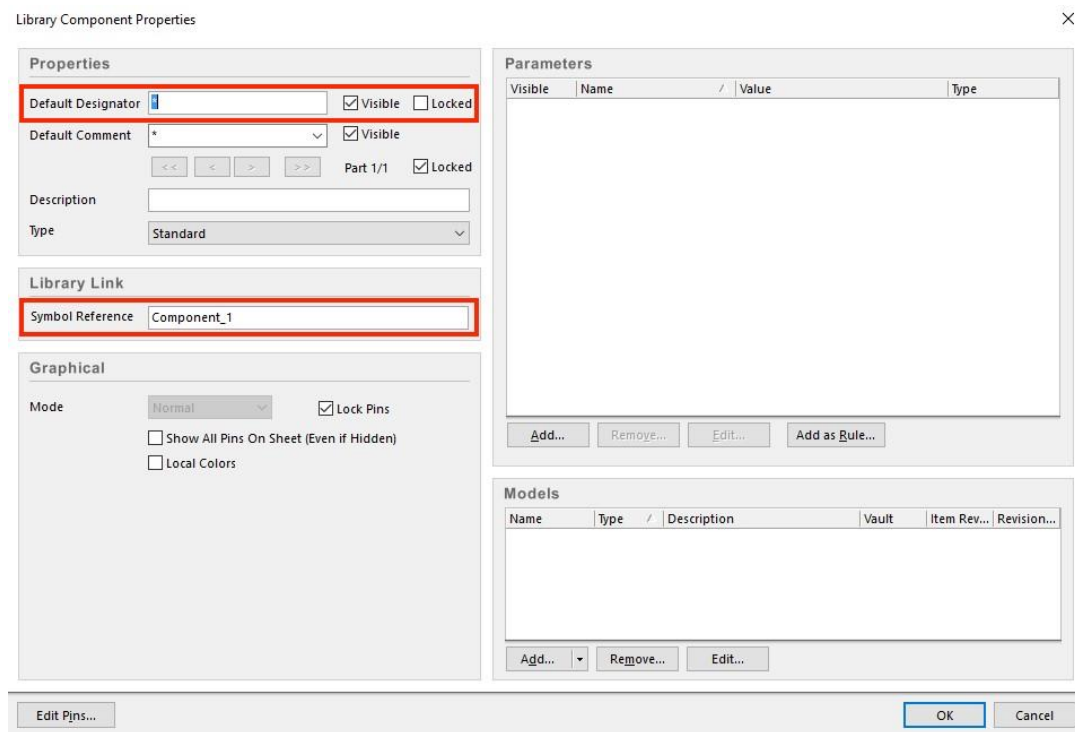


Figure 5: Library Component Properties Window

9. Next, place a rectangular shape on the schematic. This can be done via the menu bar: "Place" → Rectangle. Resize the rectangle as required.
10. Once the rectangular shape is added to the schematic, connecting pins have to be added to it. This can be done via the menu bar: "Place" → "Pin".
11. While adding pins, it is very important to ensure the white connecting pad of the pin is on the outside of the component as shown in the figure.

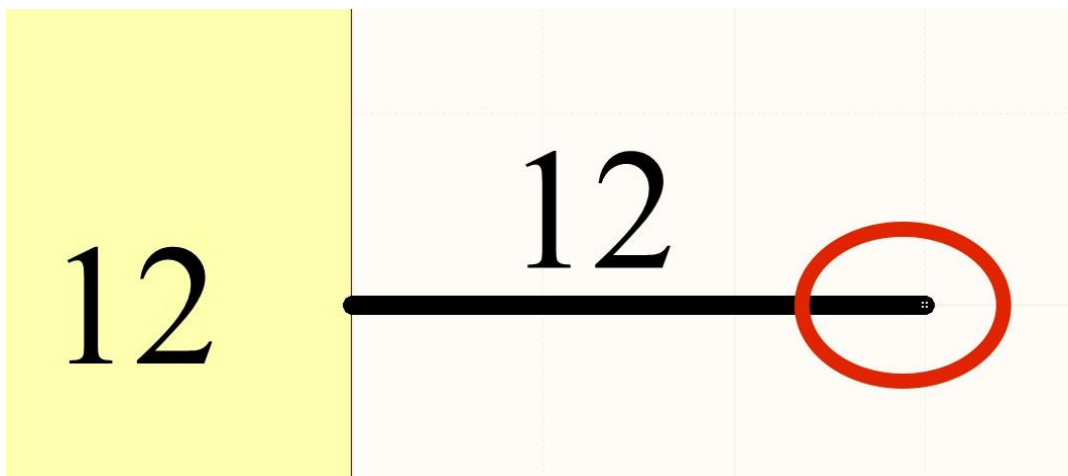


Figure 6: Pin Connected to IC with Connection Pad outside

12. After adding the necessary number of pins, add the pin numbers and labels. This can be done by double clicking the pin and changing the Display Name (for Pin Label; displayed inside) and Designator (for Pin Number; displayed on the outside).
13. For Pins that have active low functionality such as RESET, SHDN, UVLO, the bar over the label can be added by including a backslash over the characters.

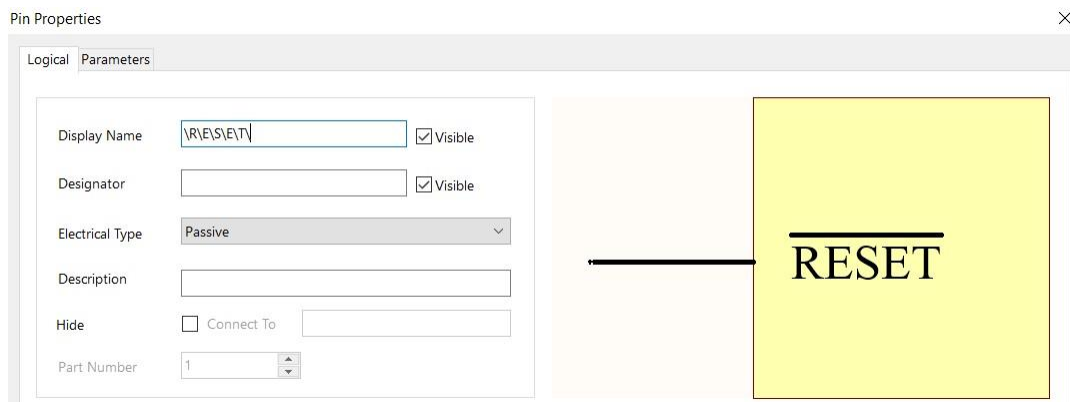


Figure 7: Adding Active Low Labels

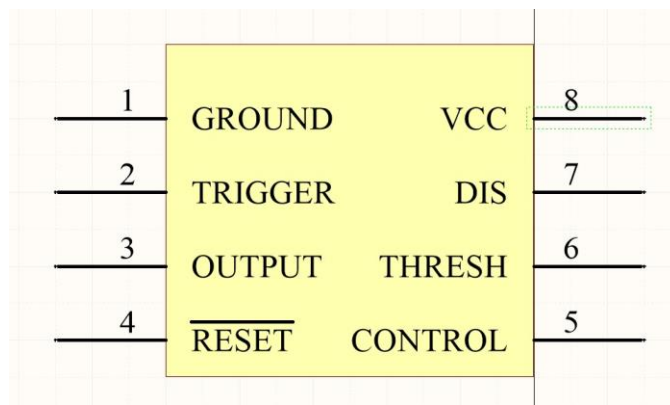


Figure 8: Completed Integrated Circuit Schematic Library

14. Once the schematic for the component is completed, it has to be saved. Navigate to "File" → "Save As...". In the window create a new folder (preferably on the Desktop) and name the folder (For Ex: Added Altium Library). Save the Schematic Library within this folder. In this example., the Schematic Library has been saved as "EECS418_Project_Library.SchLib" within the "Added Altium Library Folder".
15. Multiple components can be added to the same schematic library. To add another component (LT1077 OPAMP, for example) to the "EECS418_Project_Library" schematic library, navigate to "File" → Open → Desktop → "Added Altium Library".
16. In the file explorer window, change the folder extension to ".SchLib" from the dropdown menu to view the Schematic Library that was created. Open the Schematic Library once it is visible.
17. Once the Schematic Library window opens, click the Add button in the SCH Library Pane. In the "New Component Name" window, add the name "LT1077" in the text field.

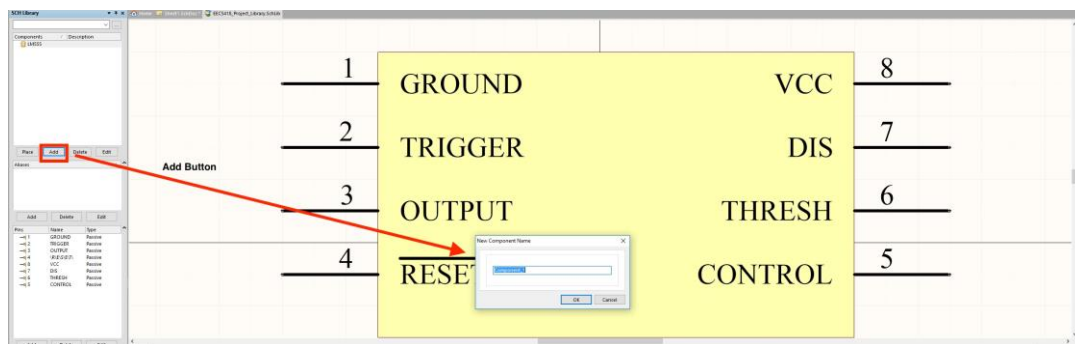


Figure 9: Adding a second Component to the User-Defined Schematic Library

18. Create the schematic for the new component as discussed in steps 9 to 13. Click the Save button once you are done.

6 Using User-Defined Components in the Circuit Schematic

1. Navigate to the circuit schematic and click the Libraries tab and click on the "Libraries..." button to open the "Available Libraries" window.

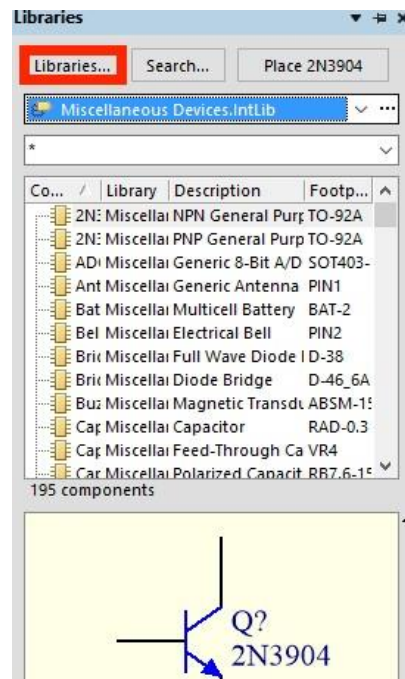


Figure 10: Library Window with the "Libraries..." button marked

2. In the the "Available Libraries" window, select the "Install" button and select "Install from Folder..." from the drop down menu.

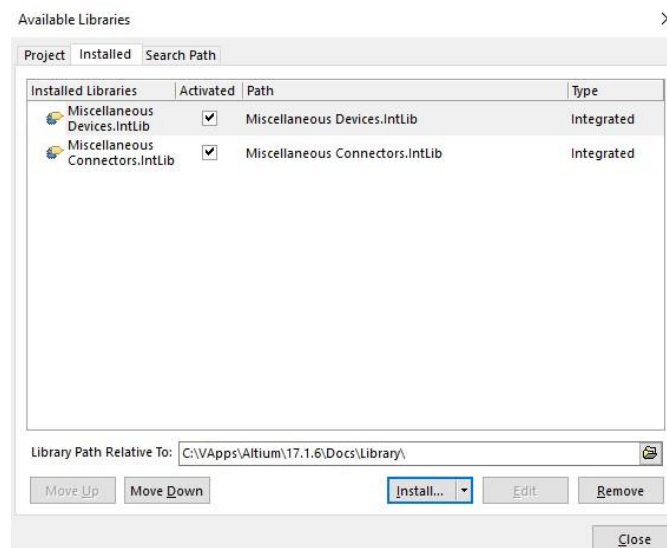


Figure 11: Available Libraries Window

3. Navigate to the "EECS418_Project_Library.SchLib" by selecting the ".SchLib" from the file extension drop-down menu. Double click the "EECS418_Project_Library.SchLib" to install the library. Close the "Available Libraries" window.
4. Once this is done, you should be able to see the "EECS418_Project_Library" with the component that were added along with the Miscellaneous Devices and Miscellaneous Connectors libraries, as shown in the figure below.

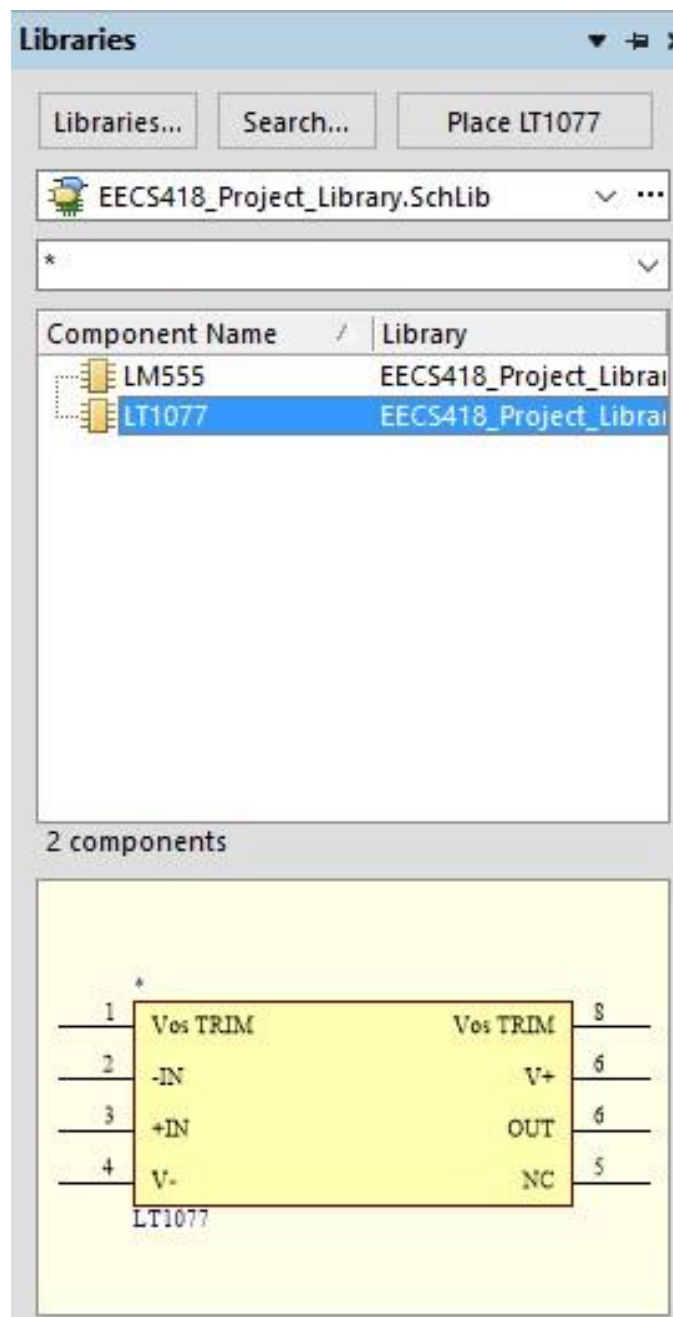


Figure 12: Library Window with User-Defined Library added

5. These components can be added to the schematic by clicking the "Place Component" button or by dragging the component to the schematic.

7 Adding a Footprint to User-Defined Components

1. Once the user-defined component has been added to the schematic, it is important to add a footprint to the component. The footprint defines a map/shape of the component on the PCB layout.
2. Before a footprint is added, check the data sheet of the component to check the component's packaging information. The LM555, used as an example, has a DIP-8 package.
3. Right-click the component on the schematic and select "Properties" from the dropdown menu.

4. In the Properties window, select the Add button and select "Footprint" from the list and click OK.

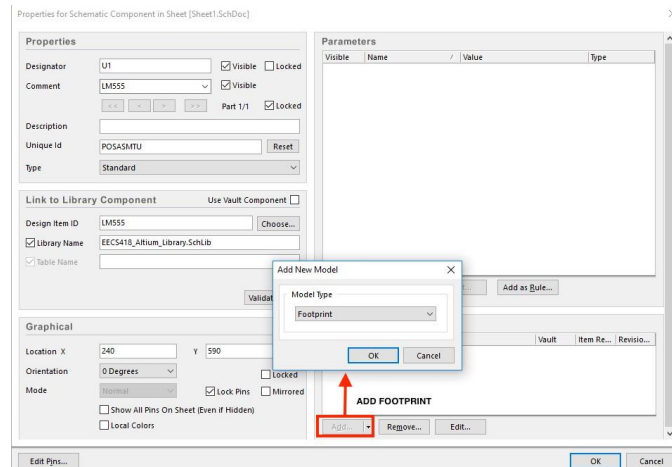


Figure 13: Adding a footprint to a user-defined component

5. In the "PCB Model" window that opens, select the Browse.. button and choose "Miscellaneous Devices.IntLib [Footprint View]" from the dropdown in the Browse Libraries Window.

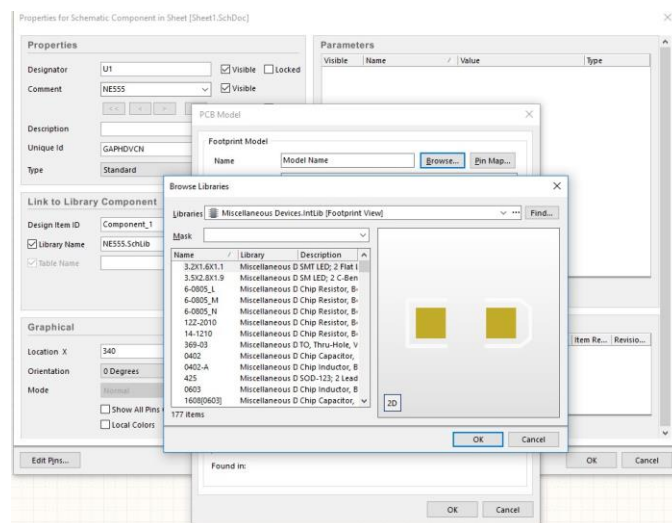


Figure 14: Library Browser Window

6. Scroll down the list of footprints and select the DIP-8 option. Click OK.

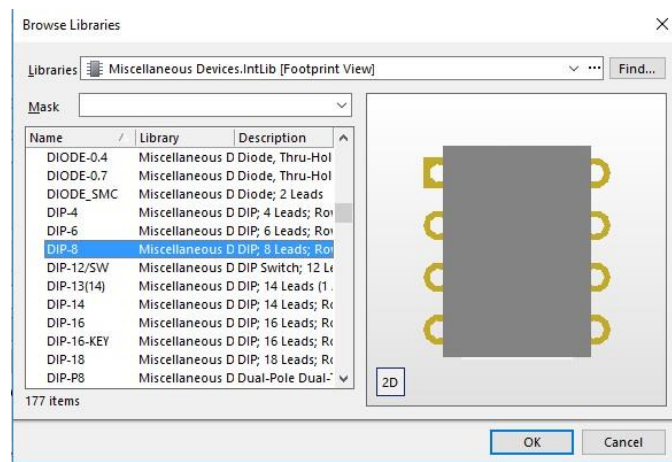


Figure 15: Library Browser Window with DIP-8 footprint selected

7. Once this is done, the DIP-8 footprint will be associated with the LM555 IC.
8. Please remember that this has to be done every time a user-defined IC is added to the schematic, i.e. the same procedure has to be repeated if another LM555 IC is added to the schematic

Complete the schematic using the steps described previously. It is advisable to label all components and associate values to them for easy reference.

8 Verifying Footprints on the Schematic

1. Once the schematic has been completed, it is strongly advisable to verify the footprints of all components in the schematic using each component's datasheet. An incorrect footprint on a fabricated PCB is impossible to fix.
2. Right click the component and select "Properties..." from the dropdown menu. In the Properties window that opens, the Models section will contain information on the component's footprint.
3. In the Models section double-click "Footprint" in the "Type" column. This will open the "PCB Model" window where the component's footprint can be seen. Click on the "2D" button to toggle between 2D and 3D views. Verify that the footprint is correct.

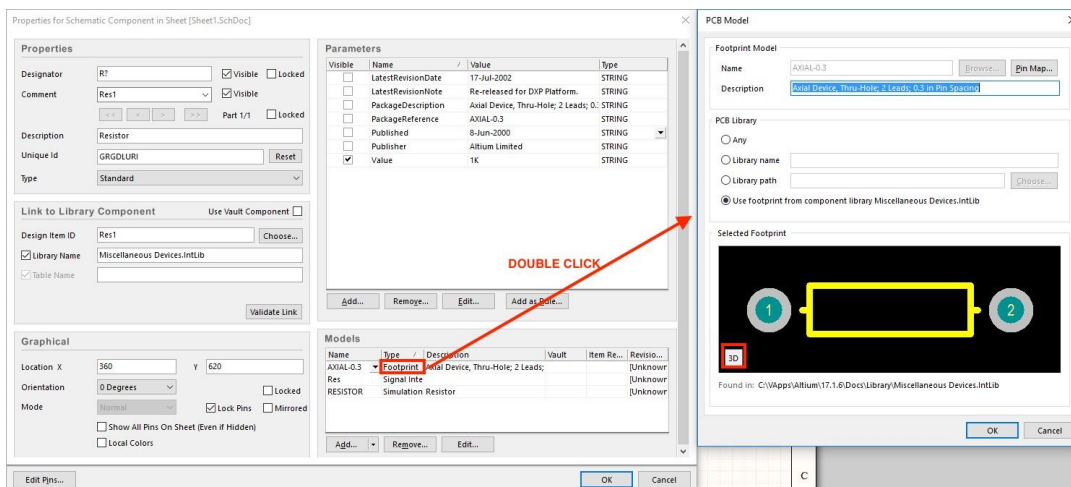


Figure 16: Library Browser Window with DIP-8 footprint selected

4. If the footprint is incorrect, in the **Models** section select "Footprint" and click the "Remove..." button. This will remove the footprint from the **Models** section.
5. Click the "Add.." button in the **Models** section. Follow the procedure outlined in the previous section (Adding a Footprint to User-Defined Components) to add the correct footprint to the component.
6. **General Information on Footprints:** The footprints of components such as resistors, inductors, capacitors, MOSFETS, Integrated Circuits can be found in the "Miscellaneous Devices.IntLib [Footprint View]" footprint library. Footprints of Headers and other connectors can be found in the "Miscellaneous Connectors.IntLib [Footprint View]" footprint library.

9 Creating a PCB Layout

1. Once the schematic has been completed and verified (make sure there are no unwired/unconnected Nets), a PCB layout can be created.
2. To start creating a PCB layout, on the toolbar navigate to "Design" → "Update PCB Document filename.PcbDoc".
3. Selecting this option will open the Engineering Change Order window. This window lists all components and Nets on the PCB Schematic which will be used to create the PCB layout. As we will require all components used on the schematic for the PCB layout, make sure all components and nets are selected. In the Add Rooms sub-section, make sure the "Room filename" is unchecked. A screenshot of the Engineering Change Order window is shown below.

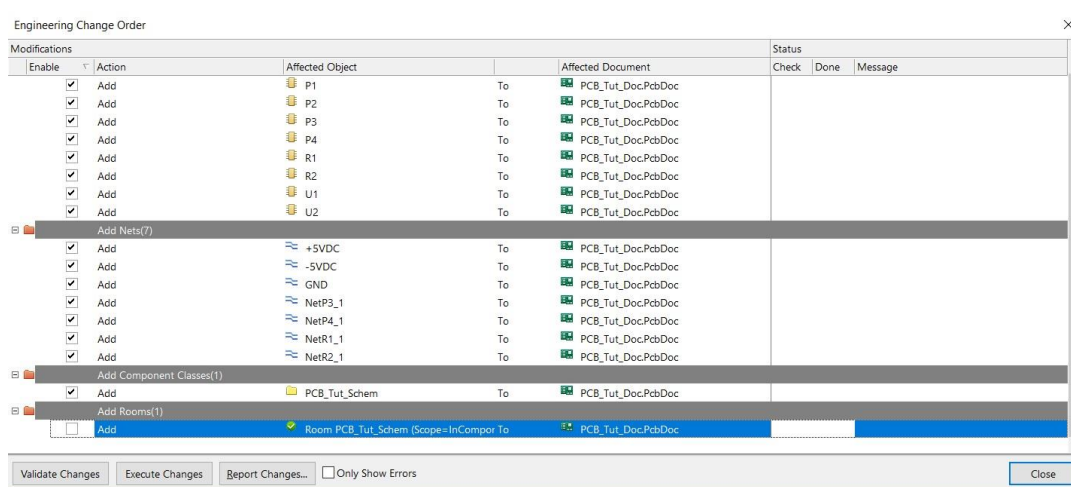


Figure 17: Engineering Change Order Window - Uncheck the "Rooms" option

4. Next click the "Validate Changes" button on the window. A green check mark will appear next to each component and net in the status bar. Then click the "Execute Changes" button.

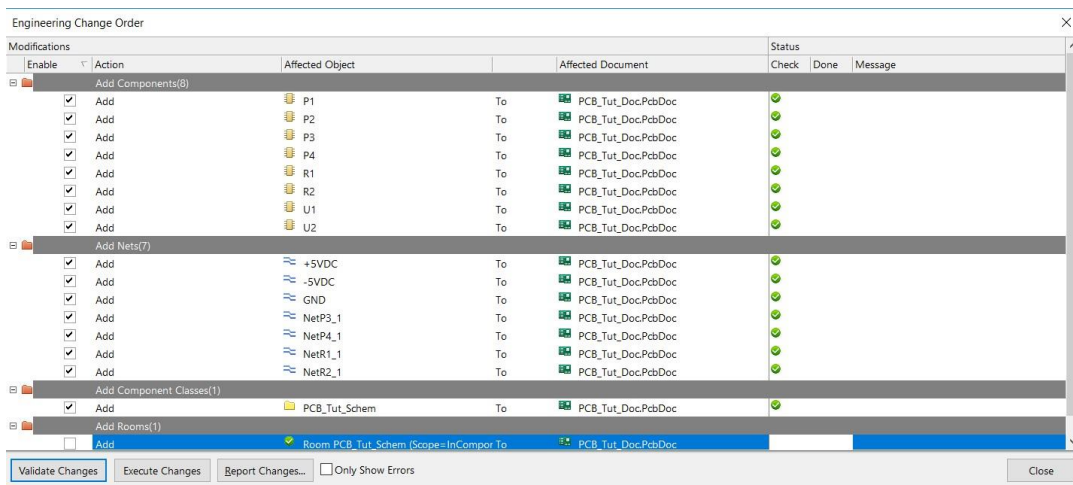


Figure 18: Engineering Change Order Window - Green check marks indicate Validated Changes

- Clicking the "Execute Changes" button will open the PCB layout window, with all the components arranged on the bottom right side of the black box. Hold down the right button on the mouse and drag the mouse to move the PCB layout window.



Figure 19: PCB Layout Window with components in the bottom right corner

- The next step is to define the board size. These dimensions will already be provided to you. To define a board shape of the required dimensions, on the menu bar, navigate to "Place" → "Line". Once this is selected a large "+" sign is attached to the mouse pointer. Using the place line tool define the board shape (length and width) by drawing lines (left click and move mouse). In this example, a rectangular board shape of 2630mil X 1405mil has been defined. (1mil = 0.001 inch).
- VERY IMPORTANT WHILE DRAWING LINES:** Make sure all the lines are continuous, i.e. ensure that there are no breaks in the lines and do not use multiple line sections to form a single line. Also make sure that the rectangle is closed, especially at the corners. This can be verified by zooming in to each line section (Ctrl + Scroll Wheel). This step of ensuring no line breaks and a closed geometrical shape is extremely important, please do not ignore.

8. The dimensions of the rectangle can be checked as and when lines are drawn using the dimensions window.
9. On the dimension window, next to [No Net] Track[10mil X dimension] the length of the line drawn = dimension. Ignore the first parameter, i.e. 10mil as this is the thickness of the line.

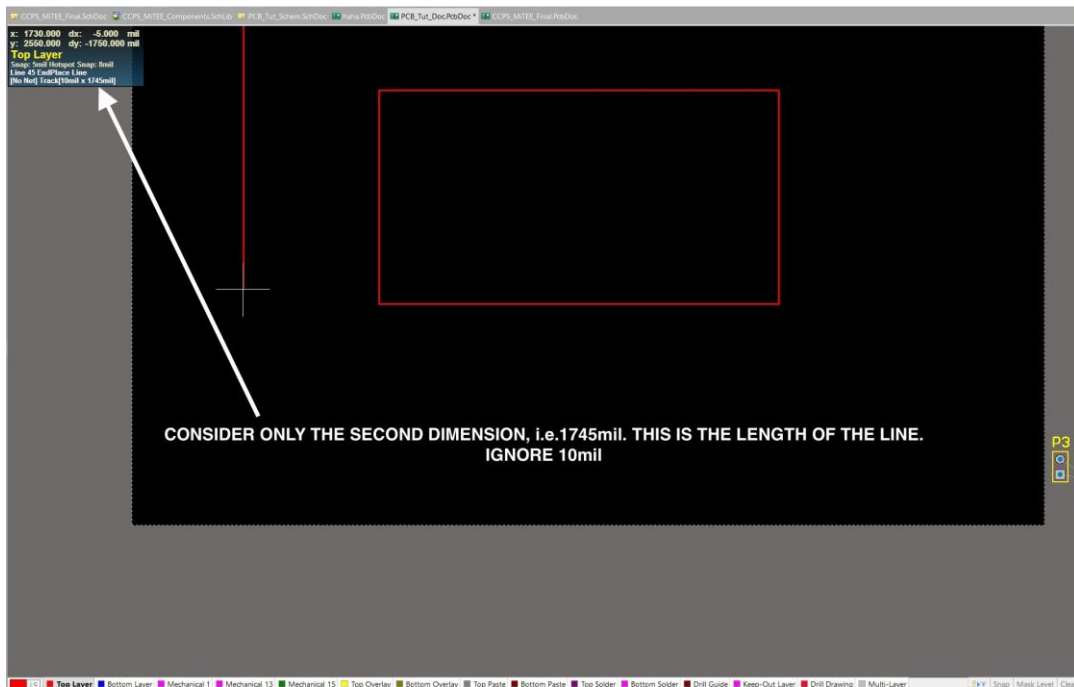


Figure 20: Use the place line tool to draw rectangles

10. Once the rectangle is drawn, select the entire rectangular using the mouse as shown in the subsequent figure to select all the lines used to draw the rectangle. Once this is done, the individual lines will be highlighted in white.

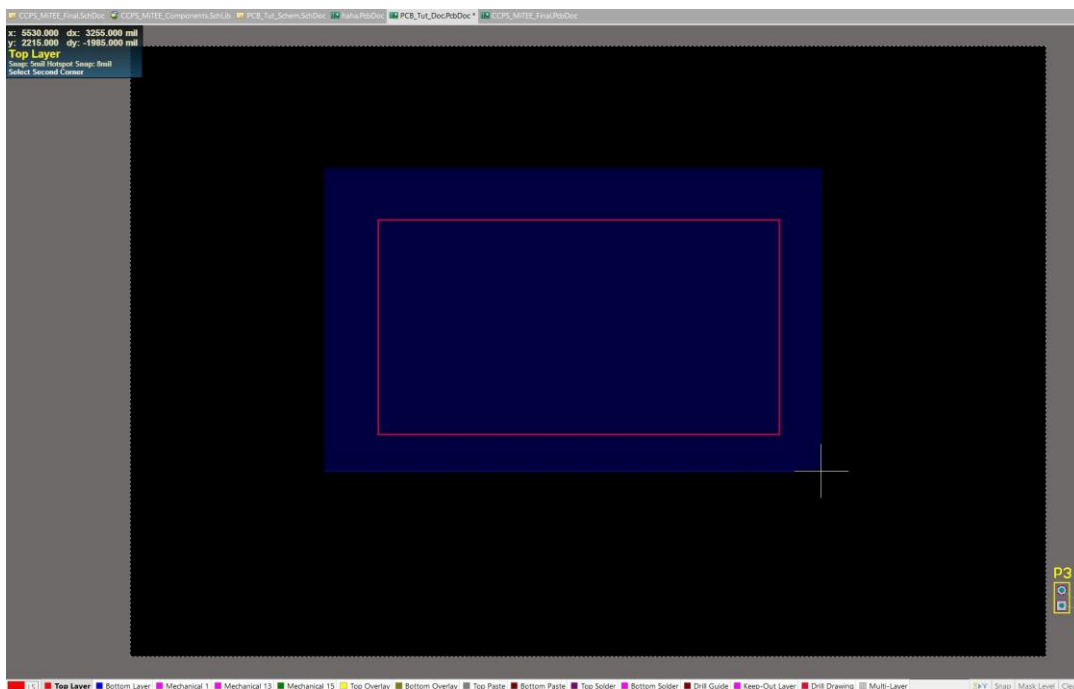


Figure 21: Drag the mouse over the entire rectangular area to select all the constituent lines

11. On the menu bar, navigate to "Design" → "Board Shape" → "Define from selected objects".

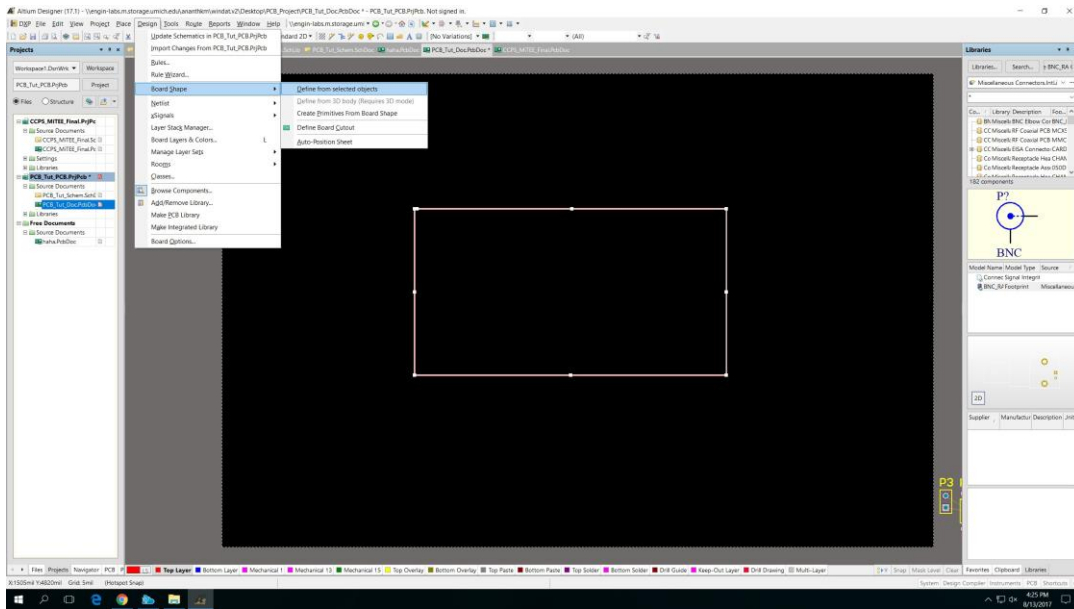


Figure 22: Defining Board Shape from Selected Lines

12. Once this is done, the PCB board shape will be defined as a rectangular area.

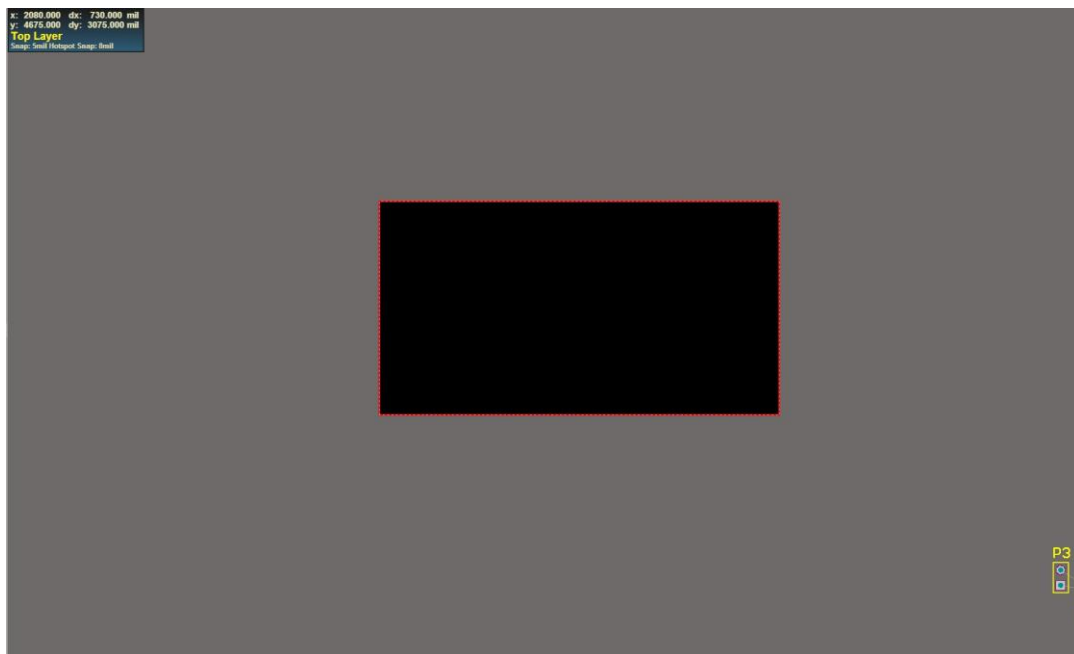


Figure 23: Defined Rectangular Board Shape

13. The last step in specifying the board shape is to select each line (left click). Once the line is selected (highlighted in white), right click and select "Properties" from the dropdown menu to open the Track[mil] window. On this window select the dropdown menu next to the "Layer" section and choose Mechanical 1. Repeat this procedure for all lines. Adding these lines to the mechanical layer helps the board manufacturer identify the right board shape.

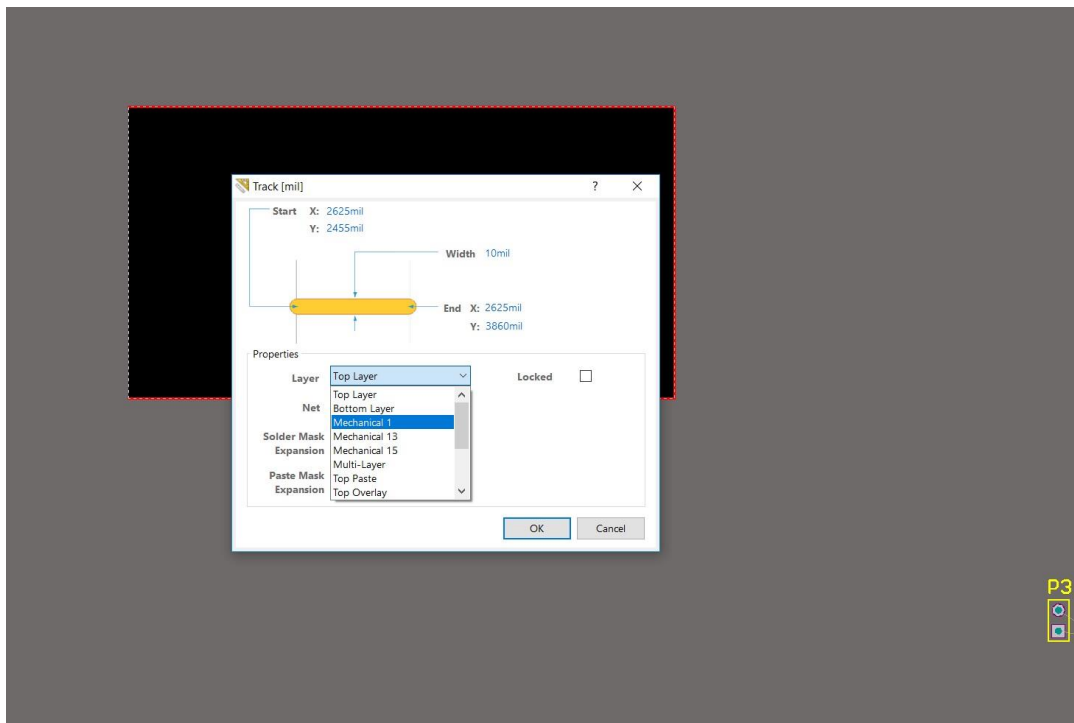


Figure 24: Adding the lines of the Rectangle to the Mechanical 1 Layer

14. Once all the lines have been assigned to the Mechanical 1 layer, they will have the same color. The default color for Mechanical 1 is **Magenta**.
15. The 3D view of the board can be used to check if the board shape has been defined correctly. The numeric key 3 is the shortcut key to activate the 3D view. Once in the 3D view hold down the Shift Key, the right key of the mouse and move the mouse to rotate the PCB 3D model on the screen. Use the numeric key 2 to revert to the 2D view.

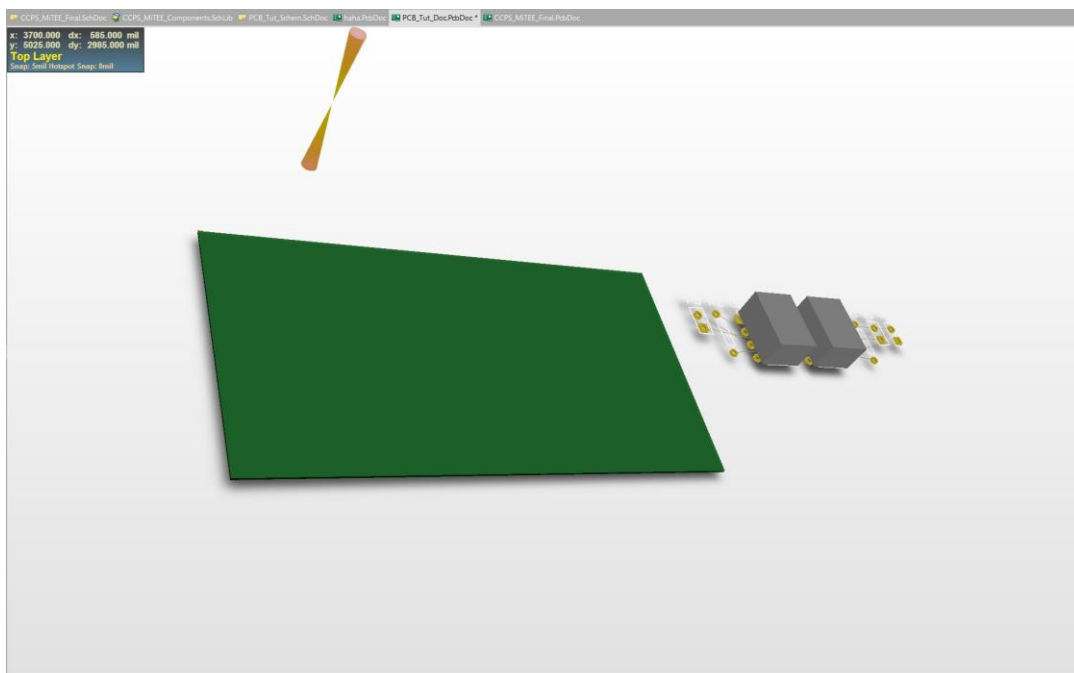


Figure 25: 3D View of the PCB

10 Placing Components on the Board

Once the board shape has been defined and verified, drag all the components (from the bottom right of the window) and place them on the board. The next step is to arrange the components on the board. The following rules can be used as broad guidelines for arranging components on the PCB.

1. It is generally a good idea to arrange components as they are on the schematic, i.e. include all components that form the power stage together, control stage together etc. When the circuit has a large number of components, this method of arranging can be very helpful in testing and debugging.
2. While adding any power input connection, it is recommended that a bypass capacitor is connected across it to remove any inconsistencies in voltage or current input.
3. While placing components such as capacitors, resistors etc. that are connected to ICs, it is recommended that the capacitor/resistor is placed as close as possible to the pins of the IC.
4. For the EECS418 project, while coupling power stages, it is highly recommended to use a coupling circuit between the two power stages. The design of coupling circuits will be discussed in class.
5. While drawing traces eliminate 90 degree turns in the trace. Circuit traces should be as short as possible.

11 Defining Copper Trace Width and Design Rules

1. Trace width, as mentioned previously in this tutorial, is the thickness of the copper wire on the PCB that carries current. In order to achieve desired operation from your PCB, the traces have to be adequately sized to allow the flow of current through various components.
2. The first step to ensure that all copper traces are adequately sized is to run a simulation of the circuit and identify currents through all the components. Typically, components such as power input ports, inductors and flyback transformers draw large currents. Integrated Circuits such as comparators, NAND logic gates, Gate Drivers etc. have a relatively smaller current draw (and current output).
3. Once the maximum current flowing through components such as inductors and transformers is known, the trace width can be computed using the Advanced Circuits PCB trace width calculator tool:
www.4pcb.com/trace-width-calculator.html
4. On the PCB trace width calculator, it is sufficient to enter only the required inputs, i.e. current (in Ampere) and copper thickness (in oz/f²).
5. The default trace width on Altium is 10mil (1mil = 0.001 inch), and is generally very safe to use with components that have a small current draw/current output.

Trace Width Website Calculator

This Javascript web calculator calculates the trace width for printed circuit board conductors for a given current using formulas from IPC-2221 (formerly IPC-D-275).

Inputs:

Current	10	Amps
Thickness	1	oz/ft ²

Optional Inputs:

Temperature Rise	10	Deg C
Ambient Temperature	25	Deg C
Trace Length	1	inch

Results for Internal Layers:

Required Trace Width	737	mil
Resistance	0.000685	Ohms
Voltage Drop	0.00685	Volts
Power Loss	0.0685	Watts

Results for External Layers in Air:

Required Trace Width	283	mil
Resistance	0.00178	Ohms
Voltage Drop	0.0178	Volts
Power Loss	0.178	Watts

Annotations:

- A red box highlights the **Inputs** section with the text "FILL IN THESE VALUES".
- A red box highlights the **Required Trace Width** for External Layers in Air with the text "REQUIRED TRACE WIDTH".

Figure 26: PCB Trace Width Calculator

- In order to allow smaller and larger trace widths, appropriate changes have to be made on the PCB's Design Rules page.
- To access this page, on the menu bar, navigate to "Design" → "Rules..." to open the "PCB Rules and Constraints Editor".

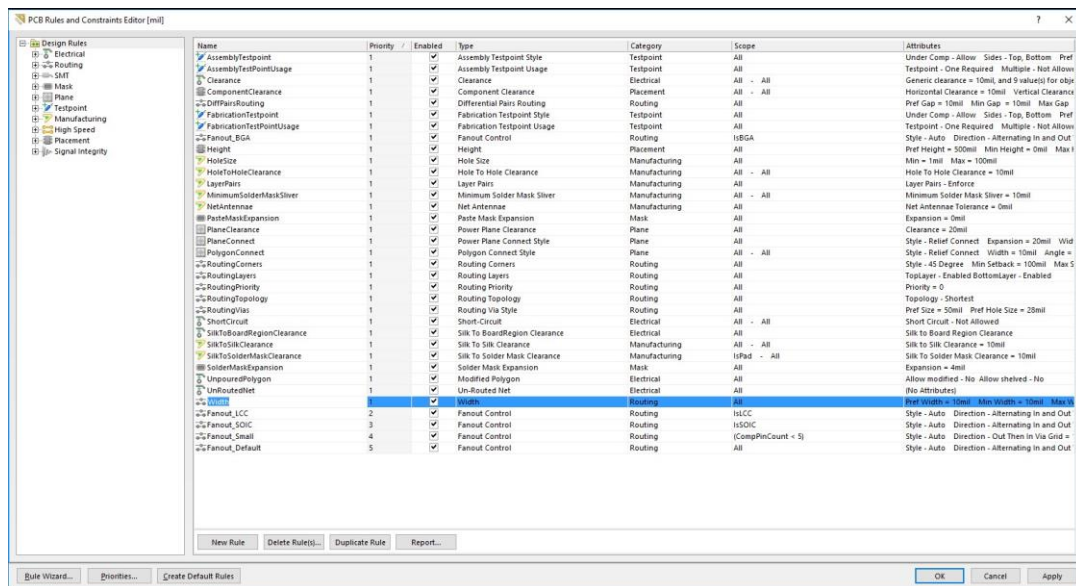


Figure 27: PCB Rules and Constraints Editor Window

8. On this window, select "Design Rules" on the left pane. Then double click the "Width" option on the right pane. In the width window, select and click "Max Width" Parameter (shown in the image below) to edit the maximum allowed trace width. As a general rule of thumb, it is safe to add 5mil to the maximum calculated trace width. For example if the largest calculated trace width (using the PCB Trace Width Calculator) is 20mil, set the "Max Width" parameter to 25mil.

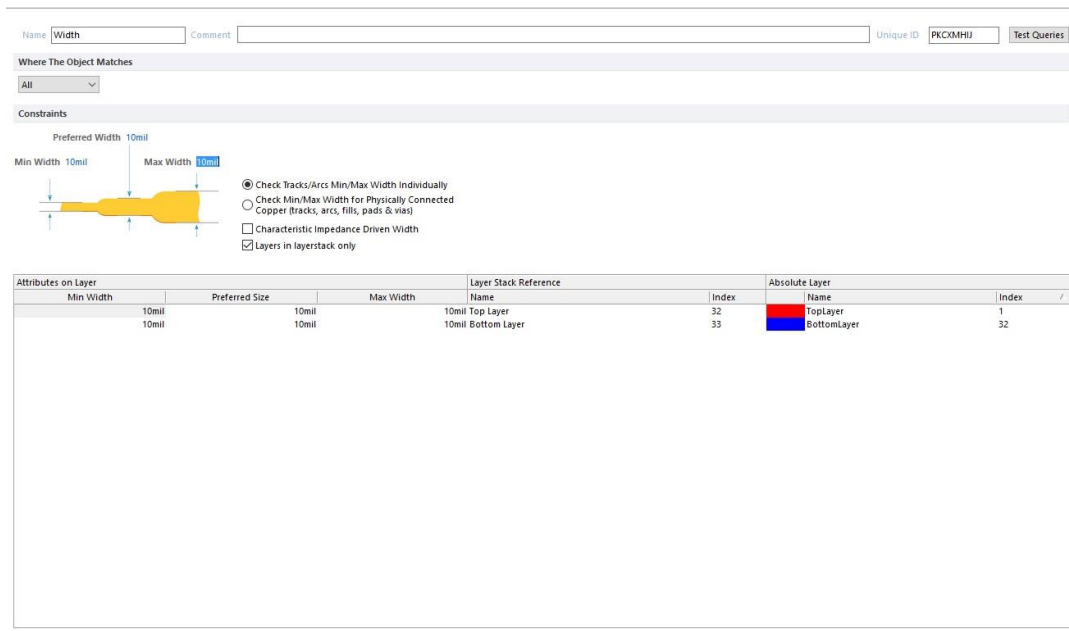


Figure 28: Specifying Max. and Min. Trace Width

9. The steps above tell the Altium Compiler that traces as wide as the value specified in the "Max Width" field may be used in the circuit.
10. Similar to adjusting the trace width design rule, other design rules can also be adjusted on the "PCB Rules and Constraints Editor". A detailed explanation of various Design Rules and constraints can be found here: [http://techdocs.altium.com/display/ADRR/PCB\(\(Design+Rules+Reference\)\)](http://techdocs.altium.com/display/ADRR/PCB((Design+Rules+Reference))).
11. As design rules will depend on a board's layout and design, the user is required to modify design rules in accordance with his/her board layout and design.
12. Sometimes, design rules can be provided (as a .RUL file). In such cases, it is possible to import the design rules from the .RUL file.
13. To import a design rule file, open the PCB Rules and Constraints Editor window. Right-click any where on the left side panel (which has a tree diagram of design rules) and select "Import Rules..." from the dropdown menu.
14. The Choose Design Rule Type dialog will appear. Select the rule types you wish to import and click OK. The Import File dialog will then appear, from where you can browse to and open, the particular PCB Rule file you wish to import.

12 Making Connections - Routing

Altium provides multiple options for routing, or connecting components. Two of the preferred methods for routing are discussed below.

1. **Interactive Routing:** This routing method can be selected from the menubar: "Route" → "Interactive Routing". Once this method of routing is selected, the mouse pointer changes into a "+" sign. Connections can be made by selecting a particular component PIN/ connection point and dragging the mouse. In the interactive routing method, Altium generates a white "trace"/line that guides the user to the next connection point.

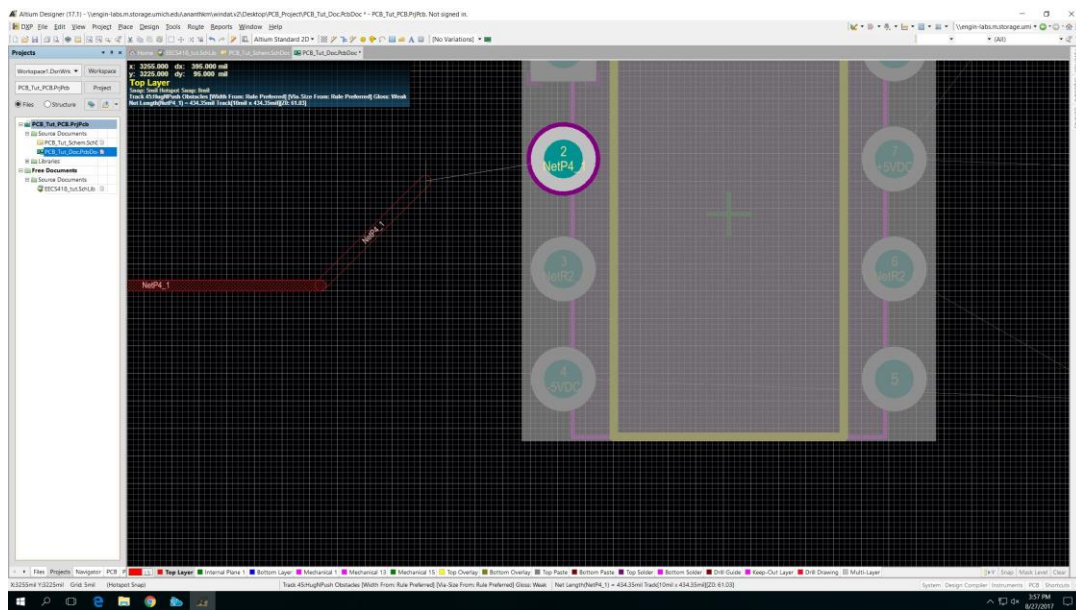


Figure 29: Interactive Routing with white routing guide line

2. Connection can be made both on the top and bottom layers. Use the "L" key to switch between bottom and top layers. By default, all traces in the top layer are marked in red, while the traces in the bottom layer are marked in blue.
3. **AutoRouting:** When the Auto-Route option is selected, Altium makes connections between components, based on user specified Design Rules and Routing Guidelines. To use the Auto-Route option, select "Route" → "Auto Route" → "All...". Selecting this option will open the "Situs Routing Strategies" window. This window allows the user to Edit Layer Directions, Edit Rules etc.

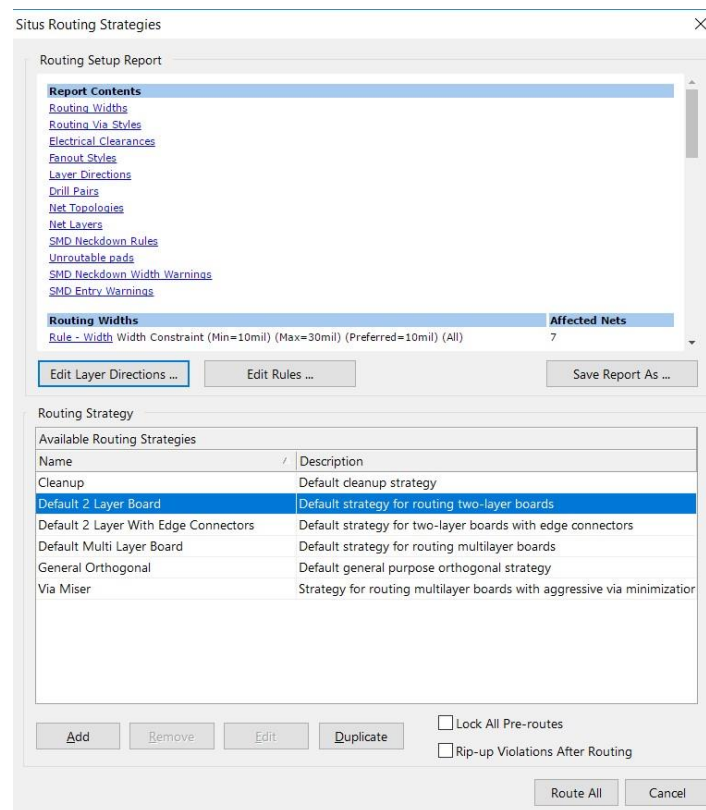


Figure 30: Situs Routing Strategies Window

4. The layers to be used for routing can be edited by selecting the "Edit Layer Directions..." option. In the "Layer Directions" window, a layer can be classified as "Automatic" (routing allowed) or "Not Used" (routing not allowed).

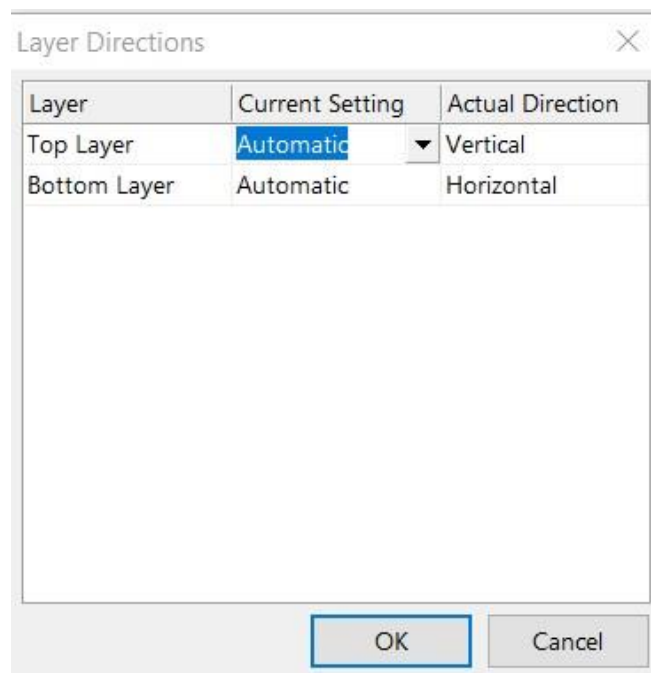


Figure 31: Select Routing Layer

5. The "Edit Rules Option" allows the user to define any design rules that are necessary.
6. Once all layer rules and design rules have been set, select the "Route All" option to complete the Auto Routing procedure.
7. Once the routing has been completed, a "Messages" window opens to display all the connections that have been successfully routed, along with a routing status report.
8. To increase the trace width of a particular connection, zoom in to identify the net name. Once the correct trace has been identified, double click the trace.
9. Double-clicking the trace will open the "Track [mil]" window. On the window set the required trace width in the "Width" field. Make sure that the width specified is less than or equal to the maximum trace width specified in the Design Rules and Constraints Editor window.

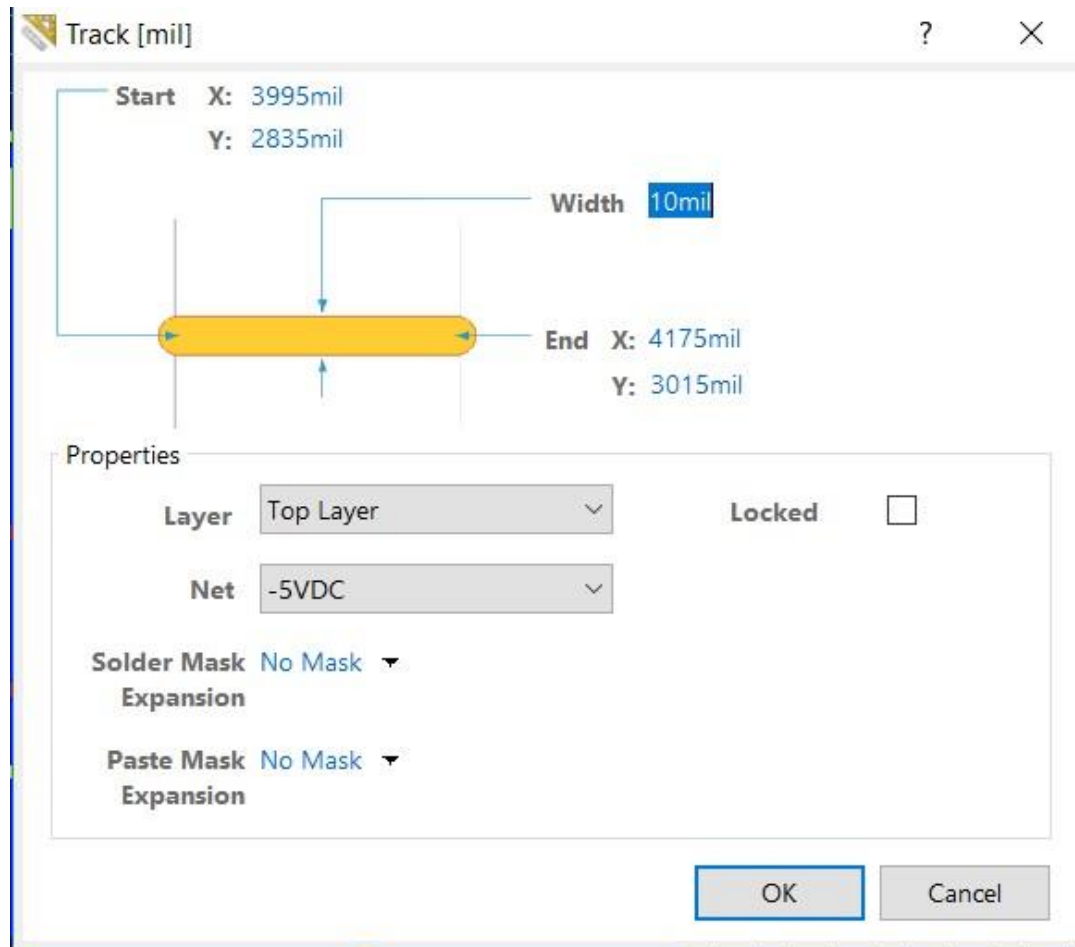


Figure 32: Setting trace width

13 Handling Design Rule Violations

1. Occasionally, while adding/arranging components or making connections, it is possible that component or connection that was added violates a particular design rule.
2. A design rule violation will cause the associated components/connections to be highlighted in green.
3. To check the nature of the error, right click the components/connections highlighted in green. From the dropdown menu, select "Violations..." to view the design rule violations.
4. Most of the design rule errors can be fixed by slightly adjusting/re-orienting the trace or component. If need be, the design rules can also be changed to accommodate the connection or component.

14 Adding a Ground Plane - Specific to the EECS 418 Project

1. This section will cover a method to add a ground plane specific to the EECS418 project. Generally, ground planes are added as internal layers and connections to them are made by adding "vias" on the board. However, in the EECS418 project, as routing is done only on the top layer, a very simple method to add a ground plane (on the bottom layer) is discussed here.
2. Select "Bottom Layer" on the layer selector toolbar, as shown in the diagram below.



Figure 33: Select Bottom Layer

3. On the menu bar, navigate to "Place" → "Polygon Pour". Selecting the "Polygon Pour" option allows the user to define a polygon of choice to serve as the ground plane.
4. Selecting the "Polygon Pour" option will open the Polygon Pour options window. In this window, replace the contents of the "Name" field by "Bottom Layer - Ground". In the Net Options section, select the dropdown menu associated with the "Connected to Net" field and select GND. Ensure that in the "Layer" field, "Bottom Layer" is selected, as shown in the diagram below.

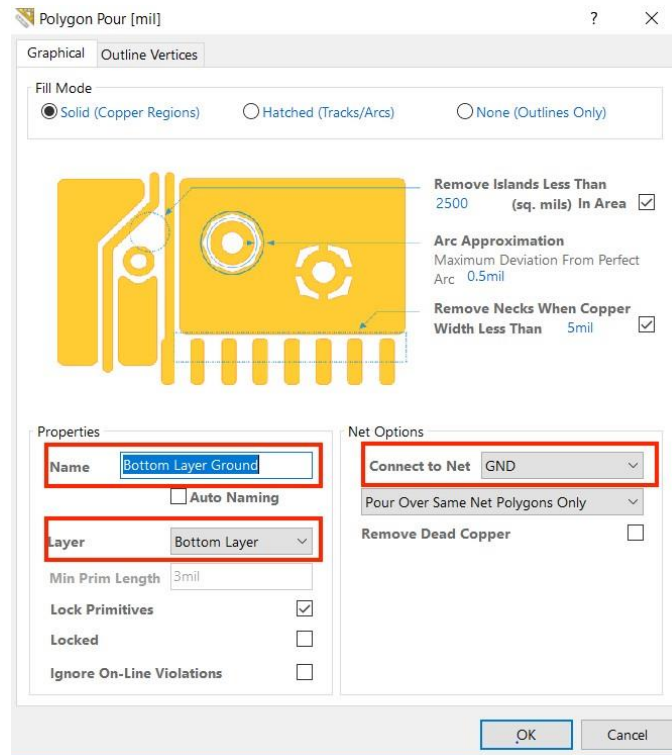


Figure 34: Polygon Pour Setup Window

5. Drag the mouse pointer to draw lines. Clicking on any point on the board defines a vertex of the polygon.
6. Define polygons such that PINS/Pads with the label "GND" lie within the polygon. (To see the NET name of a Pad/PIN, zoom in to the PIN/pad - shown in the diagram).

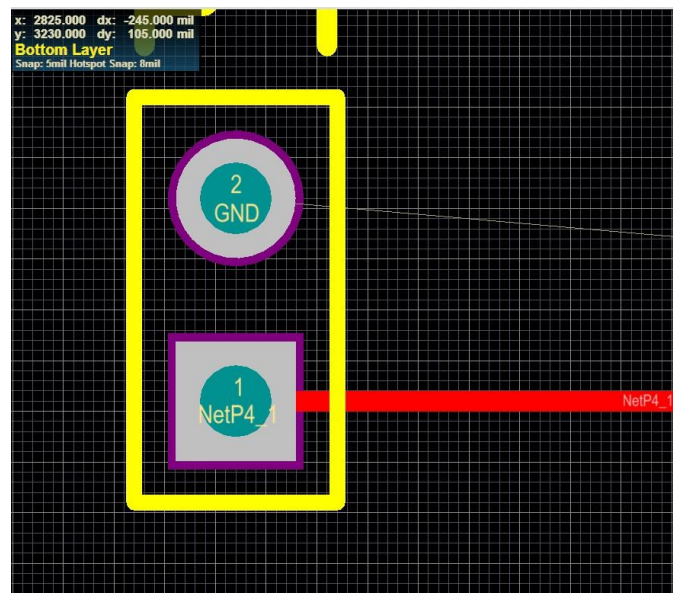


Figure 35: GND Net Label

7. For the EECS 418 project, define two Polygon Pour Regions, such that one covers all the GND NETS of the Power Stage and the other covers all the GND NETS of the control stage. (Reasons for this will be discussed in class)
8. Once you have defined a Polygon Pour Region, select the region by clicking on it and on the menu bar, navigate to "Tools" → "Polygon Pours" → "Repour Selected".
9. If you have multiple Polygon Pours serving as the GND plane, they have to be connected together. This can be done by placing a track between the two Polygon Pour Regions.
10. To place a track between two Polygon Pour Regions, while still on the bottom layer on the menu bar, navigate to "Place" "Track". Connect the Polygon Pour regions using the Track as shown in the figure below.

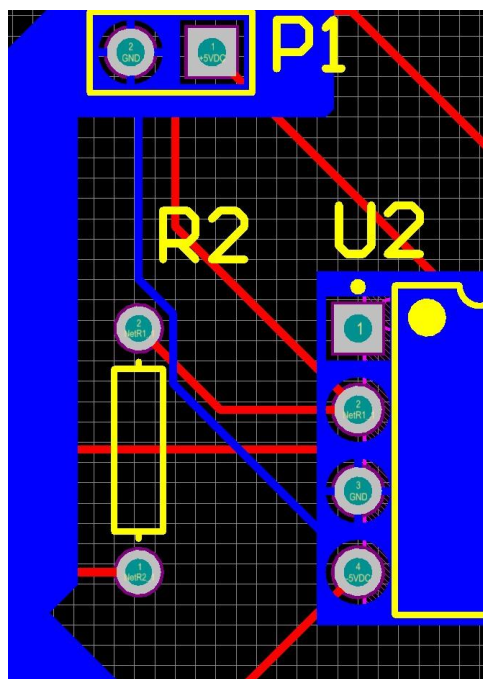


Figure 36: Polygon Pour Regions Connected by a Track

11. Occasionally, while connecting two Polygon Pour regions, the regions may be highlighted in green after the connection. This indicates a minimum clearance violation, which can be fixed by repositioning the track used to connect the two regions.
12. An example of a board which uses Polygon Pours to define GND planes is shown below.

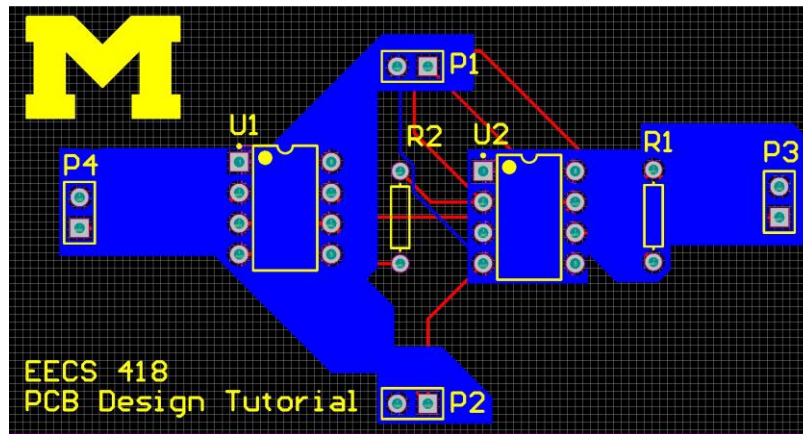


Figure 37: Polygon Pour Ground Planes

15 Running a Design Rule Check

1. Once you have finished making all connections on the PCB, it is important to run a Design Rule check to ensure that there are no critical errors on the board.
2. To run a design rule check, on the menu bar, navigate to "Tools" → "Design Rule Check" to open the Design Rule Checker window. On this window select "Run Design Rule Check".
3. A Design Rule Verification Report is generated after the Design Rule Check has completed executing. This report lists Warnings and Violations on the PCB Layout. However, not all violations and warnings are critical. The most critical errors that a user should be concerned about are "Un-Routed Nets" and "Short Circuit Violations". Other warnings and violations such as "Clearance Constraints", "Silk to Silk" constraints can be ignored.
4. Address all critical design errors listed in the Design Rule Verification Report.

16 Generating Gerber Files and NC Drill Files

1. The last step in the PCB design process is to generate Gerber and Drill files. These files tell the PCB manufacturer where to place polygon pours, pads and circuit traces.
2. To generate Gerber files, on the menu bar, navigate to "File" → "Gerber Files". Selecting "Gerber Files" will open the Gerber Setup window.

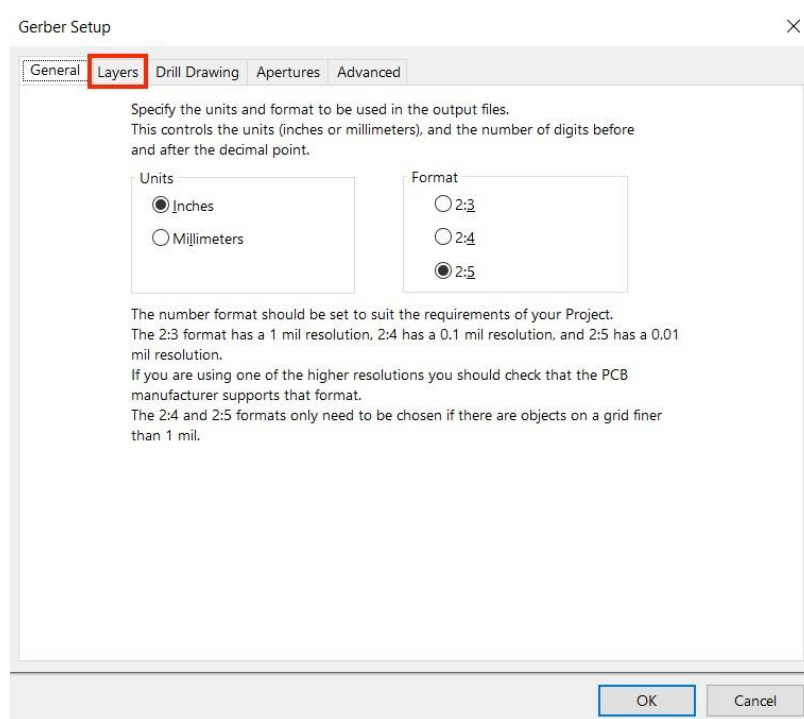


Figure 38: Gerber Setup Window with the Layers Tab highlighted

3. Unless mentioned, do not change any of the settings on the Gerber Setup window.
4. In the Gerber Setup window select the "Layers" tab. This allows the user to select the layers for which Gerber Files are to be generated. For the EECS 418 project, it is required to generate Gerber Files only for "Top Layer" and "Bottom Layer". The layers for which Gerber Files are required will be mentioned in class. Select the appropriate layers in the window and select OK.
5. The last output file to be generated is the NC Drill file. This lets the manufacturer know where to provide holes on the PCB.
6. To generate the NC Drill Files, on the menu bar, navigate to "Fabrication Outputs" → "NC Drill Files". This will open the NC Drill Setup window.
7. On the NC Drill Setup window, check the last item: "Generate EIA Binary Drill File" and select OK.
8. The files that need to be sent to the PCB Manufacturer are the Gerber Top layer (.GTL extension), Gerber Bottom Layer (.GBL extension) and Drill Drawing (.DRL extension).