

INTRODUCTION

Mentor Graphics is the leading printed circuit board (PCB) design software company. They produce many types of software to create and simulate circuits and to design PCBs. This document outlines one combination of Mentor Graphics software called the **PADS Flow**.



1. PADS Suite

The PADS Suite is a collection of programs with all the tools necessary to create a PCB. It is designed to be used by individuals and small teams, but it is still very powerful.

2. PADS contains three applications

- a. PADS Logic: for design entry, i.e. to create the circuit diagram
- b. PADS Layout: to layout components on a PCB
- c. PADS Router: to create traces between the PCB components

3. DxDesigner

DxDesigner is another design entry program. PADS Logic is designed to be used with Layout and Router, but DxDesigner can be used with many other tools, including PADS and even those from other companies. For example, DxDesigner is tightly integrated with Expedition Enterprise, an application similar to PADS Layout and Router, but is designed for collaboration between large teams.

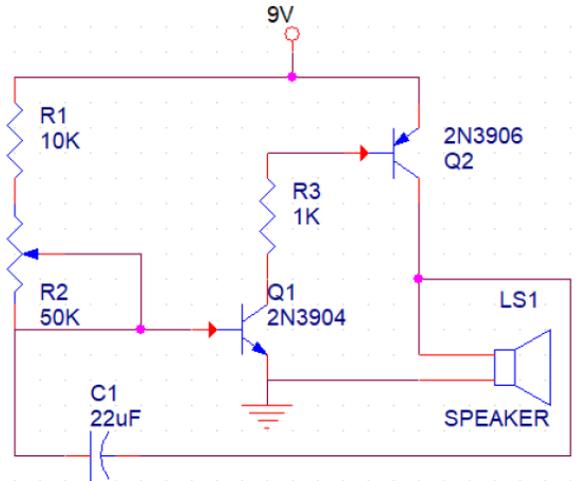
The features present in DxDesigner, such as the ability to break a circuit down into small blocks of components, allows large and complicated circuits to be created more easily than in PADS Logic. It also contains a tool called DxDataBook which allows a user to search a database for components based on a required criteria.

TABLE OF CONTENTS

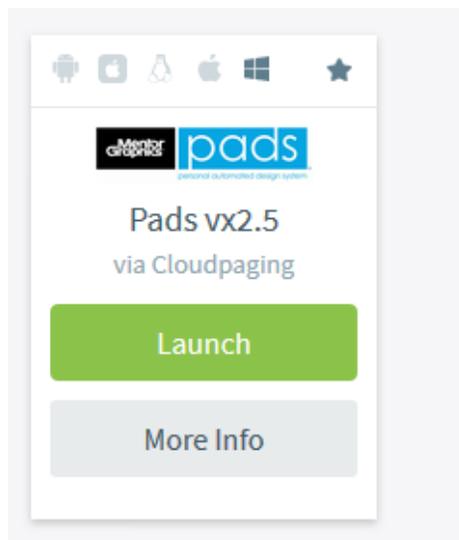
1.	GETTING STARTED	1
2.	TIPS WHEN WORKING WITH PADS	2
3.	START A PADS LOGIC PROJECT	3
4.	ADD/DELETE PARTS.....	4
5.	ARRANGE PARTS	8
6.	MAKE CONNECTION.....	10
7.	PADS LAYOUT	14
8.	ADD BOARD OUTLINE	16
9.	SET ORIGIN	18
10.	SET PAD STACKS	18
11.	DESIGN RULES	19
12.	PADS ROUTER	22
13.	AUTOROUTE.....	23
14.	MEASURE BOARD DIMENSION	25
15.	EXPORTING CAM DOCUMENTS	27
16.	CREATE NEW LIBRARY	32
17.	MAKE CUSTOM COMPONENTS.....	33
18.	MAKE CUSTOM LAYOUT DECAL	39
19.	CHECK CUSTOM CREATED PART	49

1. GETTING STARTED

- a. Obtain a circuit diagram, this tutorial will use the circuit in the figure below as an example



- b. Login into GlobalProtect as you will need the VPN to run PADS VX2.5. Instructions on how to download and login into the VPN can be found here:
<https://employees.senecacollege.ca/spaces/77/it-services/wiki/view/3722/student-vpn>
- c. Login to MyApps and Launch PADS VX2.5



- d. After MyApps is done launching, a folder will pop up. Go into the folder and run PADS Logic VX.2.5

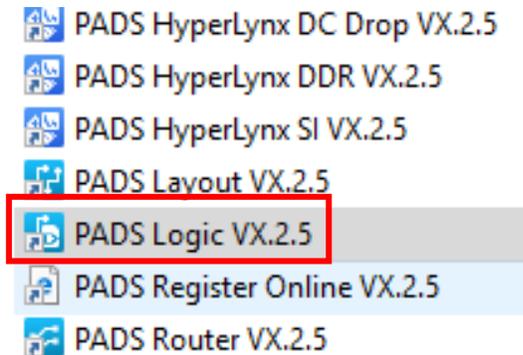
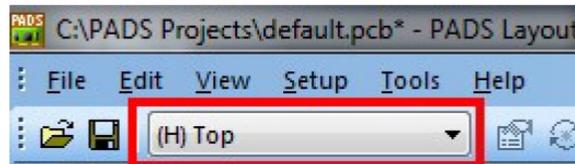


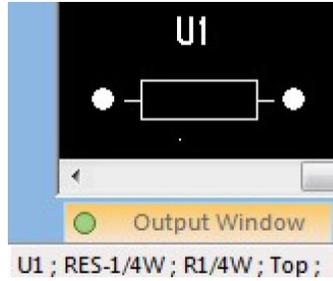
Figure 3: PADS Logic

2. TIPS WHEN WORKING WITH PADS

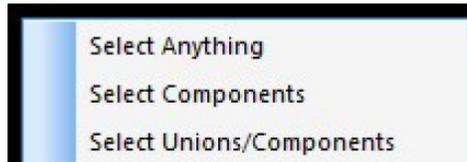
- a. Be aware of 3 things when using PADS (Logic, Layout, Router)
- For Layout and Router: be aware of the layer you are working on. The current layer is shown in the drop-down box in the toolbar. Components should generally go on the top layer and traces on the bottom.



- The selected object is shown in the status bar and is highlighted in white. If you change modes while something is selected, the mode will be directly applied to the selection. For example, when creating routes between parts you normally click where you want to start a trace then click where you want it to end. If an object is already selected, changing to routing mode will set the start of the traces as whatever is selected. This behaviour can be confusing so it is recommended to **make sure that nothing is selected when changing modes**. To ensure that nothing is selected, press the ESC button.

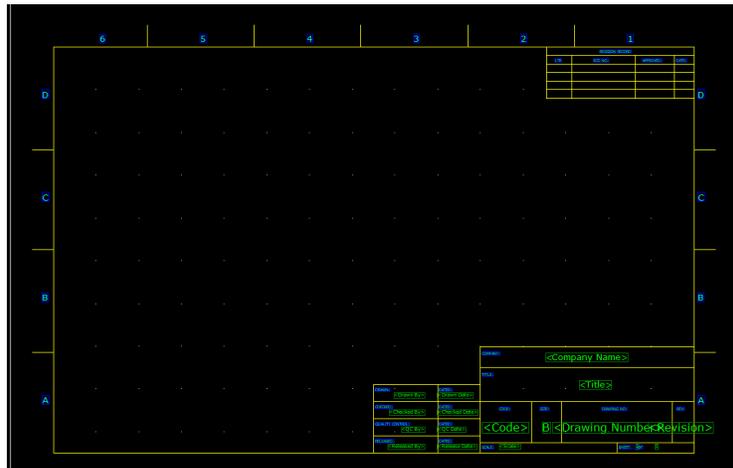


- The selection filter determines what objects can be selected by clicking on them. With nothing selected, you can right-click the working area and choose one of the “Select...” options. **If you cannot select or delete something in the workspace it means you need to change the filter.**

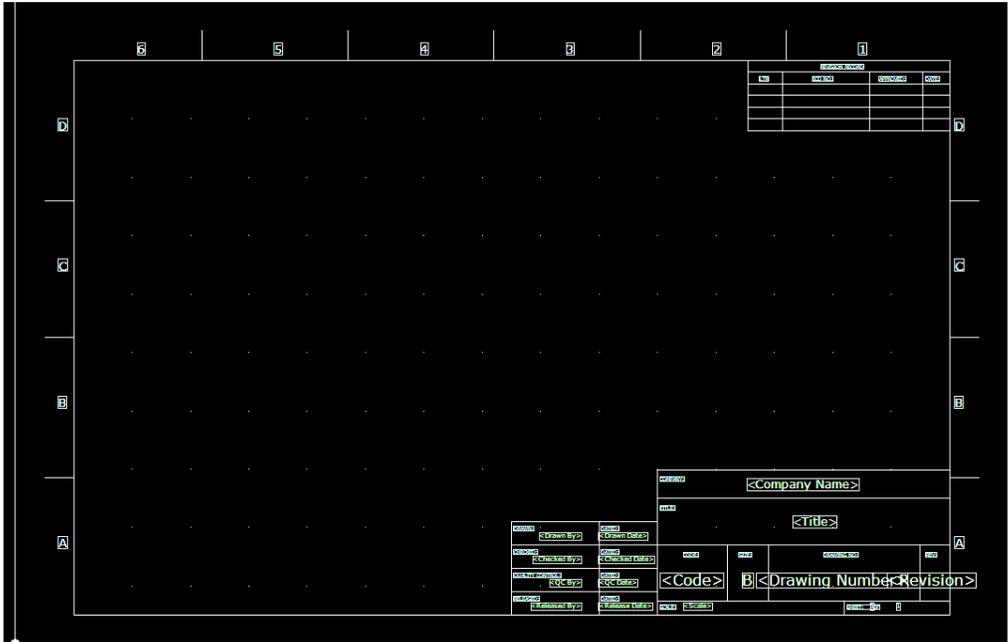


3. START A PADS LOGIC PROJECT

- Start a new project
 - At PADS Logic main screen, select New
 - After a new project is successfully created, the screen below should show



- Put the mouse cursor to one of the yellow outer borders, then left click. The whole block should be white as shown below.



- Press DEL on the keyboard to delete the information block

4. ADD/DELETE PARTS

- Navigate the mouse cursor to Schematic Editing Tool Bar, select Add Part



- Search for Resistor (RES*), then click Add

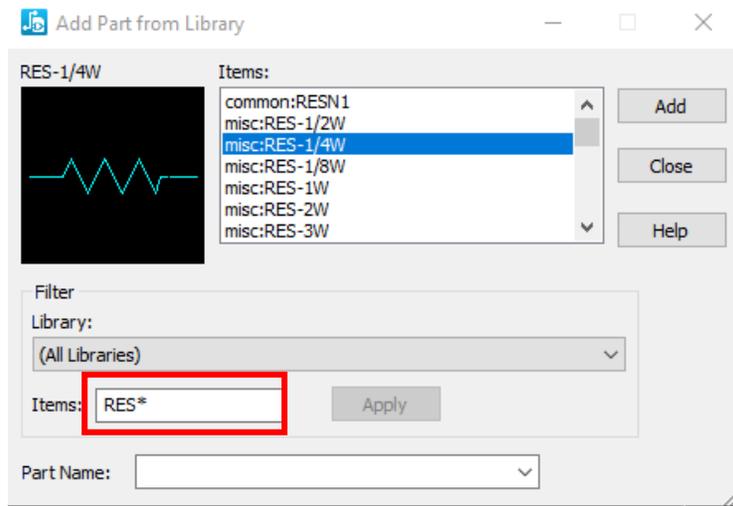


Figure 10: Add resistors

- c. Add as many Resistor as the circuit requires (in this example we need 2 resistors). Then press Esc on the keyboard to exit Adding resistors.
- d. Continue adding other components of the circuit
 - Transistors (2N39*)

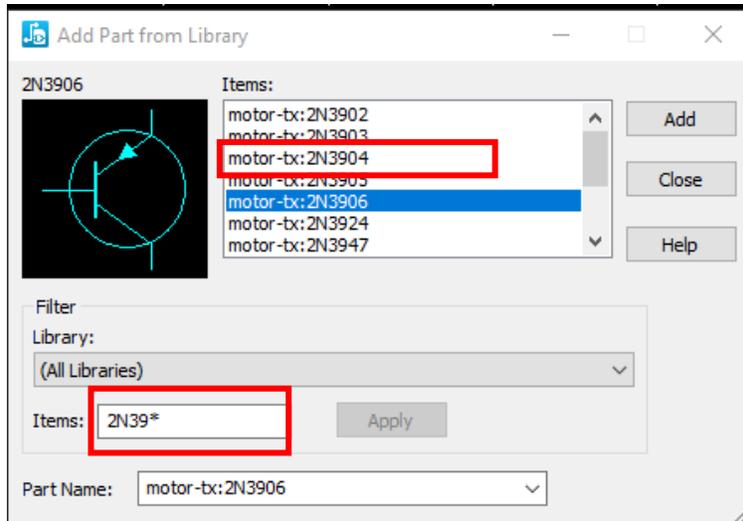


Figure 11: Add transistor

- Capacitors (CAP*)
 - Select the highlighted CAP for through-hole components

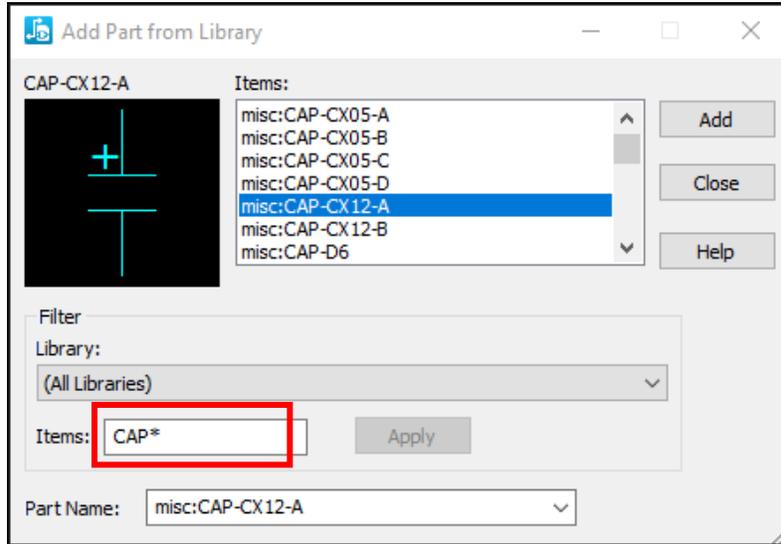


Figure 12: Add capacitor

- Variable Resistors (VRES*)

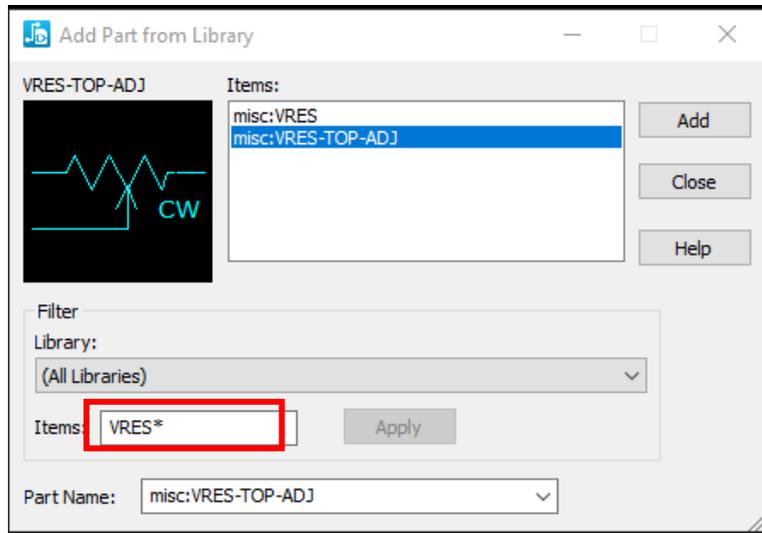


Figure 13: Add variable resistor

- Headers in order to connect speaker wires (HEADER*)

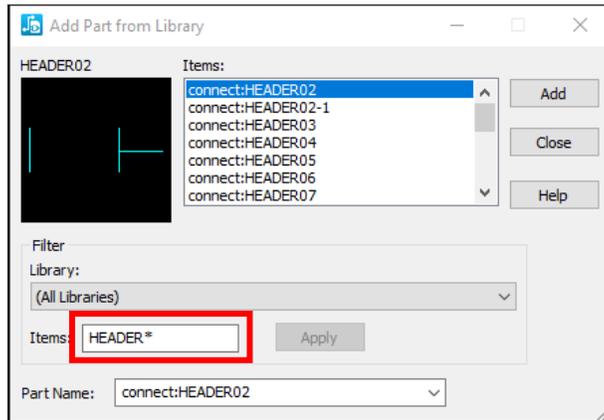
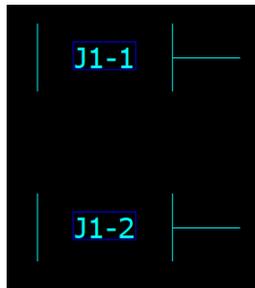
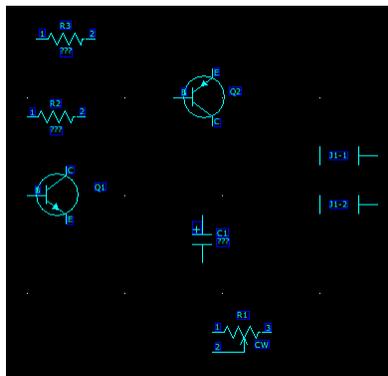


Figure 14: Add speaker headers

Note: Headers come in pair, so make sure to check their name to ensure we add them correctly



- Double-check to see if all the parts from the schematic are no visible in the workspace



- e. Delete an incorrect part
 - Right click on a blank space
 - Choose “Select Parts”
 - Left click on the part
 - Press DEL on the keyboard

5. ARRANGE PARTS

- a. Select a part that needs to be moved (selected part will become white)



- b. Click and hold the left mouse and drag to move parts (or press CTRL + E then move the mouse)
- c. Rotate the part
 - Right click on a selected part then choose “ROTATE 90” (or press CTRL + R)
- d. Mirror the part
 - X - AXIS: Right click on a selected part then choose “X MIRROR” (or press CTRL + F)
 - Y - AXIS: Right click on a selected part then choose “Y MIRROR” (or press CTRL + SHIFT + F)
- e. Add part value
 - Double click on the “???” of the part



- Put the desired value in the popup window, then click OK

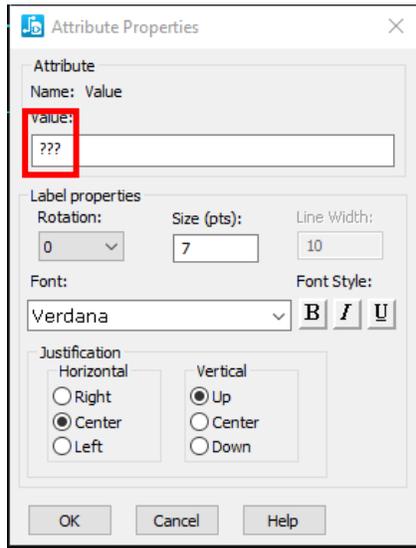


Figure 19: Putting new part value

- f. To change the part label
- Double click on the part label



Figure 20: Select part label

- Put the new label in the popup window, then click OK

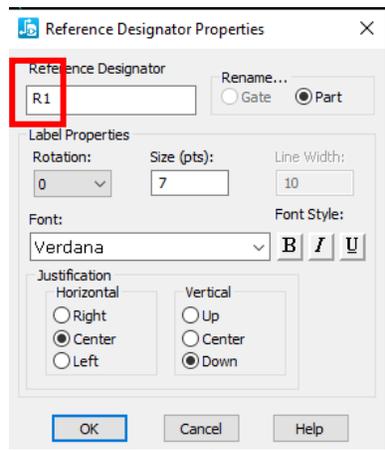
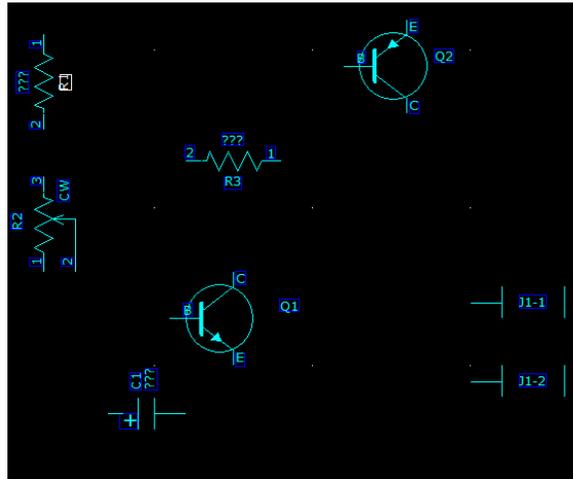


Figure 21: Changing part label

g. Below is an example of parts arrangement



NOTE: For J1-1 and J1-2, X-MIRROR was used to have the correct orientation

6. MAKE CONNECTION

a. Navigate to Schematic Editing Toolbar, select Add connection



NOTE: when the tool is selected, the cursor is as below

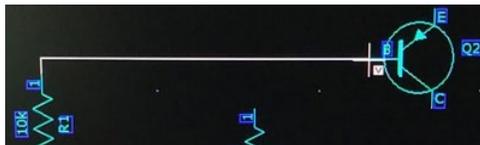


b. To make a connection between two pins

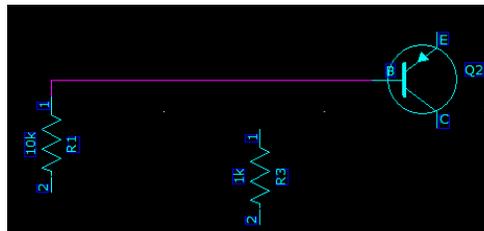
- Left click at the start pin



- Left click again at the end pin



- A successful connection should look like below



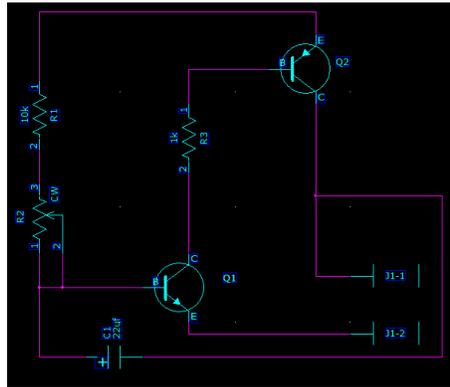
c. To delete a connection

- Right click on a blank space
- Choose "Select Connections"
- Left click on a connection

NOTE: This also applies for deleting part. Sometime when a part cannot be deleted, make sure "Select Parts" is selected.

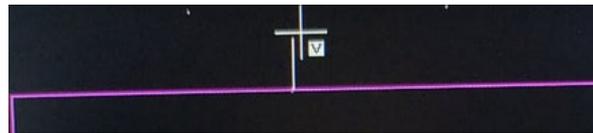
- Press DEL on the keyboard

d. Finish making other connections



e. Add Power Connections

- Draw a connection like below by clicking on the existing line and bringing out a connection



Note: Do not left click, leave the connection white as above

- Right click, select POWER (SHIFT + SPACEBAR)

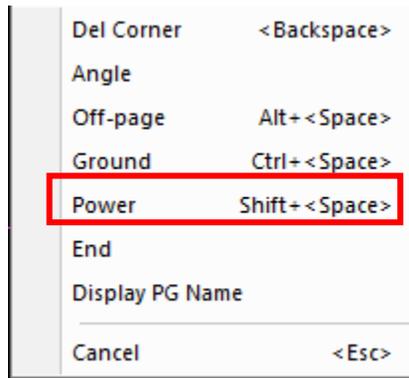
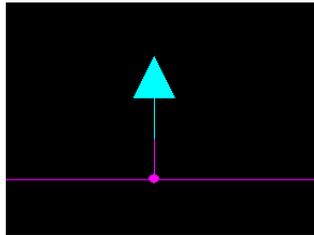


Figure 30: Select Power

- Left click to place VCC



- Repeat from drawing the connecting line in Figure 29 but for GND

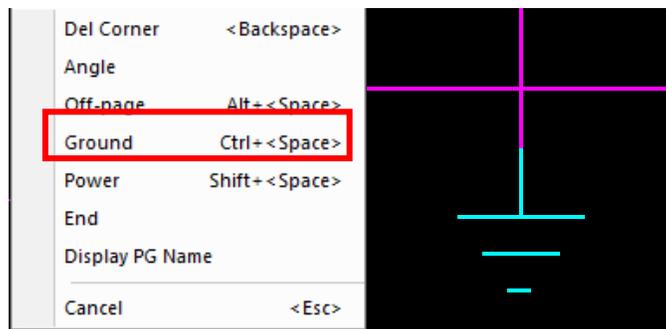
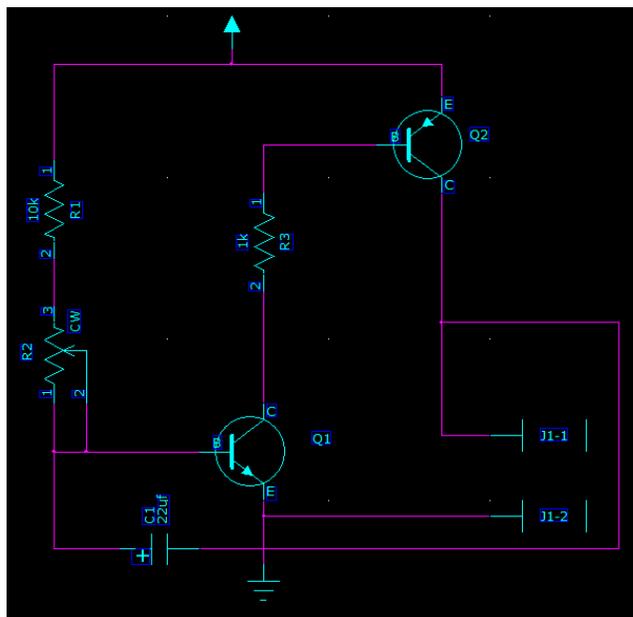


Figure 32: Place Ground

- Below is the finished result of our example circuit



7. PADS LAYOUT

Follow these steps if you have completed and saved a schematic design using PADS Logic.

- a. Navigate to Standard Toolbar, select PADS Layout



- b. Click New on the popup window

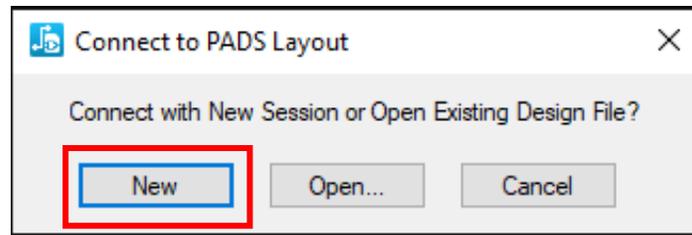


Figure 35: Create New PADS layout project

- c. Click Switch To... if the following window pops up

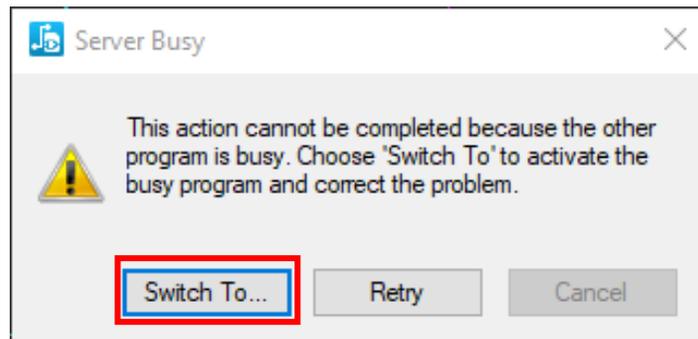


Figure 36: Select to switch to Layout

- d. PADS Layout will launch
- e. Select the "Design" tab, then click Send Net list in the next window

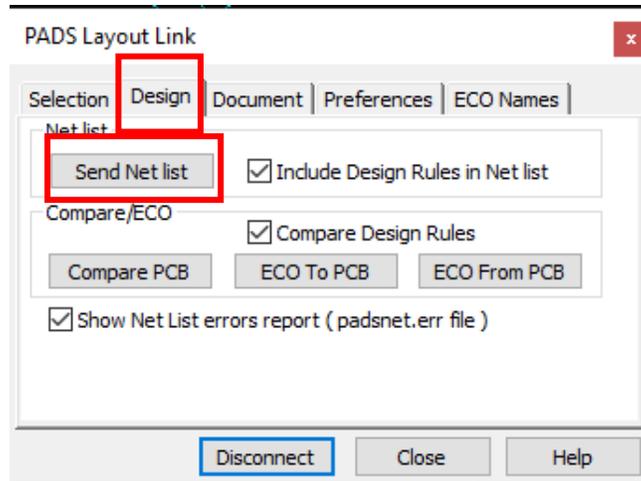
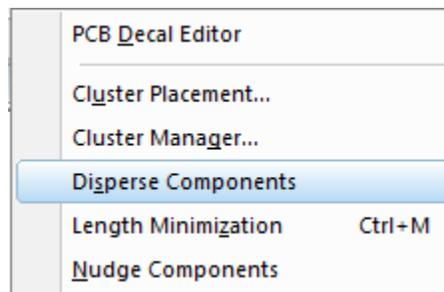
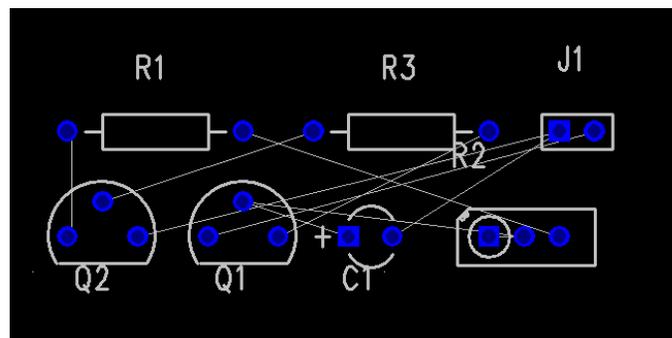


Figure 37: Send netlist to Layout

- f. Navigate to PADS
- g. Layout, select Tools→Disperse Components→ click Yes

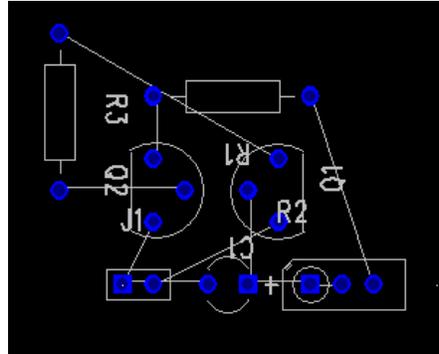


- h. The components will show on the screen



i. Arrange the components

- The movement mechanism is the same with PADS Logic (See page 8). Below is an example of a component arrangement.



8. ADD BOARD OUTLINE

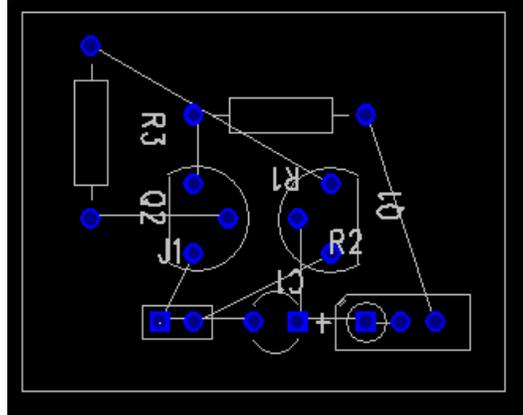
- a. Navigate to Standard Toolbar, select “Drafting Toolbar”.



- b. Navigate to Drafting Toolbar, select “Board Outline and Cut out”

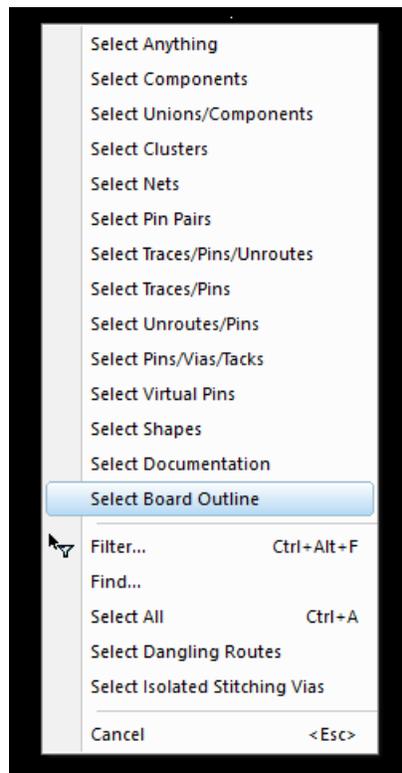


- c. Right click on workspace area, select “Rectangle”
- d. Left click once, then move the mouse to draw a rectangle around the components



a. To modify Board outline (if needed)

- Right click on an empty area of the work space, select “Select Board Outline”



- Left click and hold to select a corner or an edge of the board outline
- Move the mouse to adjust the dimension of board outline

9. SET ORIGIN

Before drawing traces, make sure to set origin correctly

- Navigate to Menu Toolbar, select "Setup\Set Origin"
- Left click to place the origin at the bottom left of the board, then click Yes

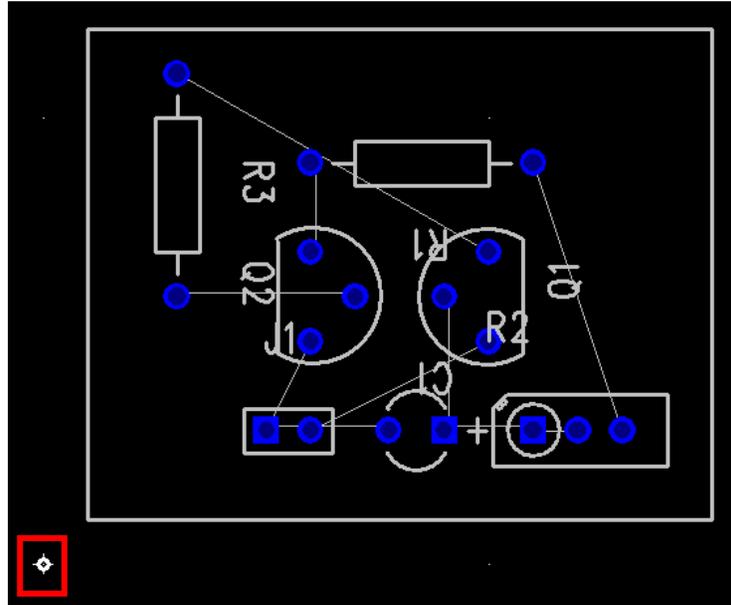


Figure 45: Place origin

10. SET PAD STACKS

- Navigate to Menu Toolbar, select "Setup\Pad stacks"
- For each Decal name, change diameter of the Opposite Side Layer to 80

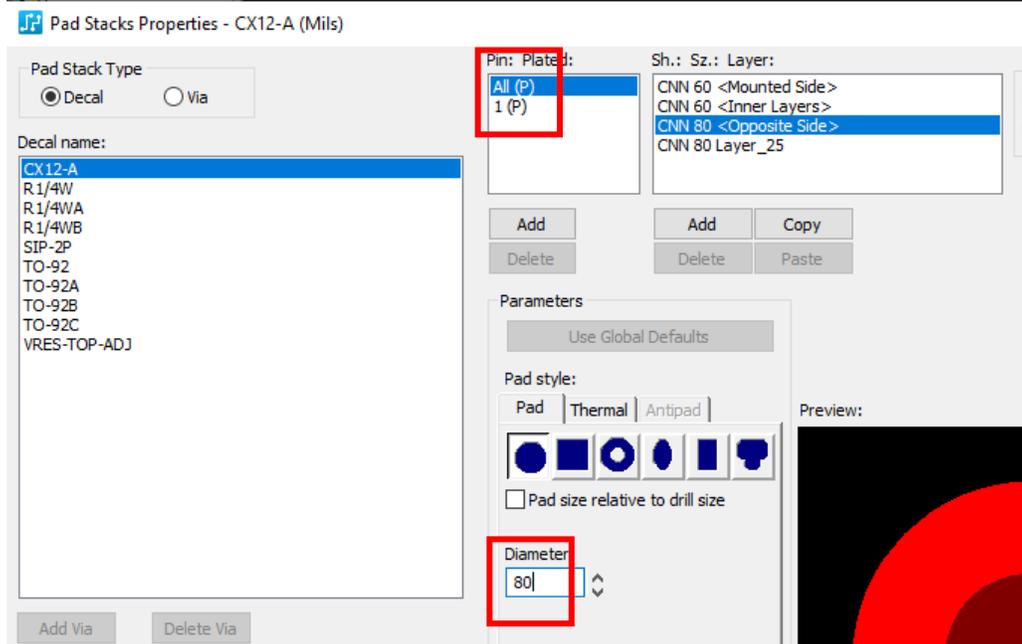


Figure 46: Change Pad stacks

NOTE: Some components have more than 1 pin. Change diameter for all the pins separately

- c. Repeat step 2 for all components. Click Yes if prompted when switching between components
- d. Click OK when finish changing diameter for all components

11. DESIGN RULES

- a. Navigate to Menu Toolbar, select Setup→Design Rules

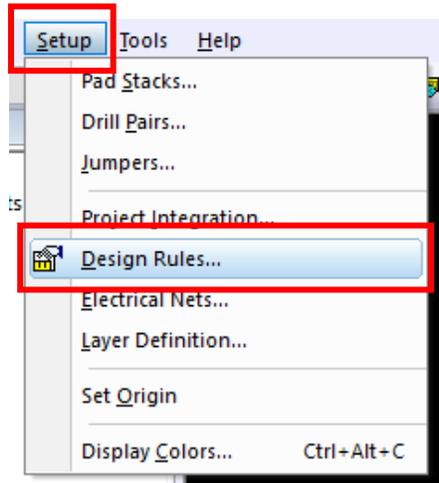


Figure 47: Design Rules tools

b. Click Default → Clearance

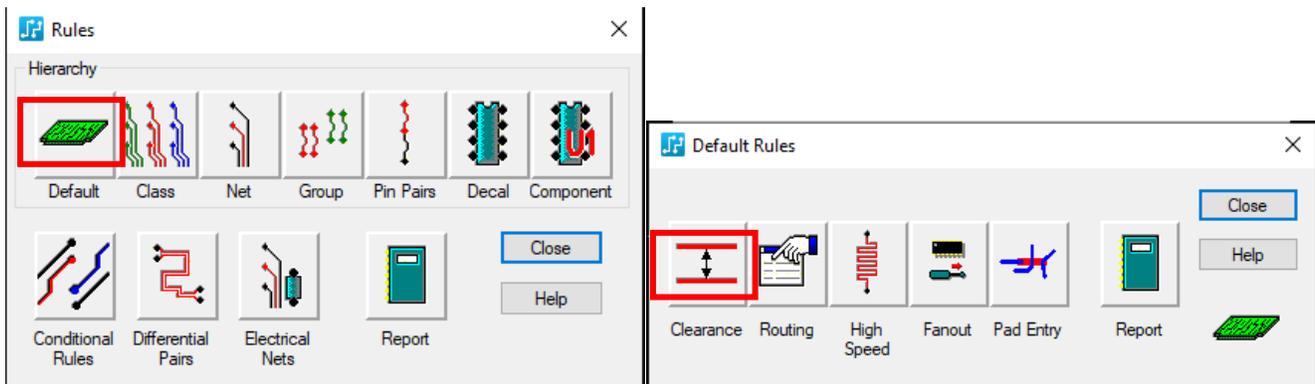


Figure 48: Setting up clearance

c. Click All

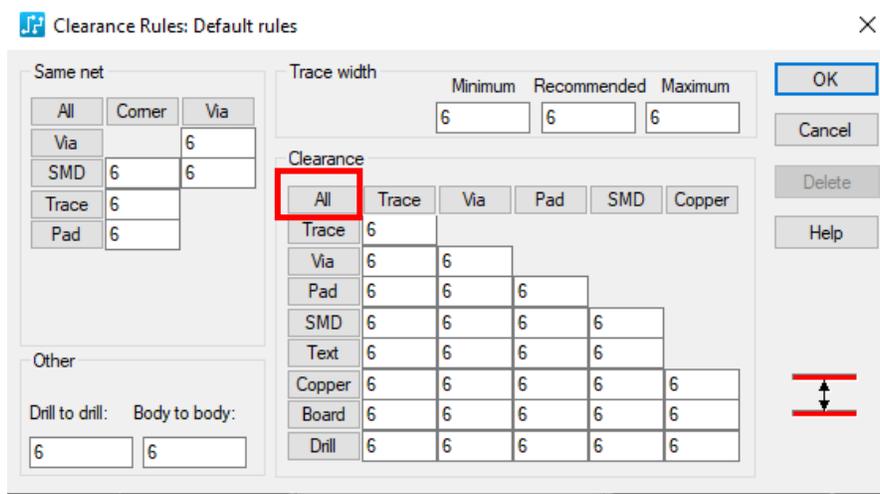
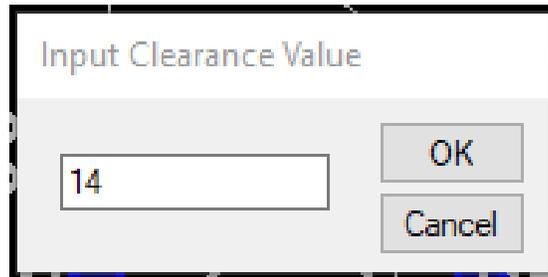


Figure 49: Change Clearance

d. Input clearance Value, then click OK



e. Change Trace width, then click OK, then click Close twice



12. PADS ROUTER

- a. To switch to Router, navigate to Standard Toolbar, select PADS Router



- b. Navigate to Standard Toolbar to switch to work with Bottom layer



NOTE: The pads on the components should look red like below

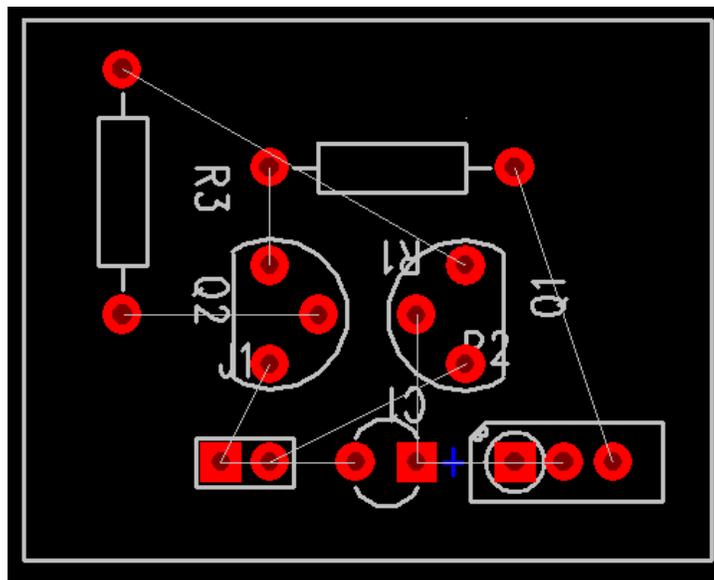


Figure 53: Visual of bottom layer

- c. Navigate to Standard Toolbar, select “Route Editing” to begin drawing traces



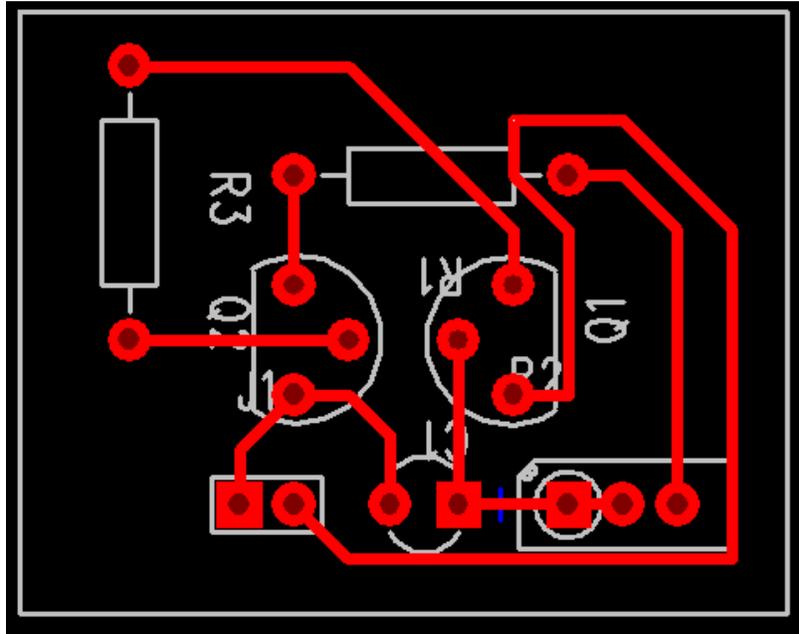
- d. Navigate to Route Editing Toolbar, select “Interactive Route”



- e. Begin drawing a trace by Left clicking at the start pin
- f. Finish a trace by Left clicking again at the end pin

NOTE: The traces will only be made when they match the white connections (which are transferred from PADS Logic)

- g. Below is an example after all the traces are made



- h. To UNROUTE a trace

- Select wanted trace then press BACKSPACE on the keyboard

13. AUTOROUTE

Another way to make traces in PADS Router is to use the AUTOROUTE function

- a. Navigate to Menu Toolbar, select Edit\Properties

- b. Select tab Layer Biasing, then uncheck the box Allow Routing for Top layer, then click OK

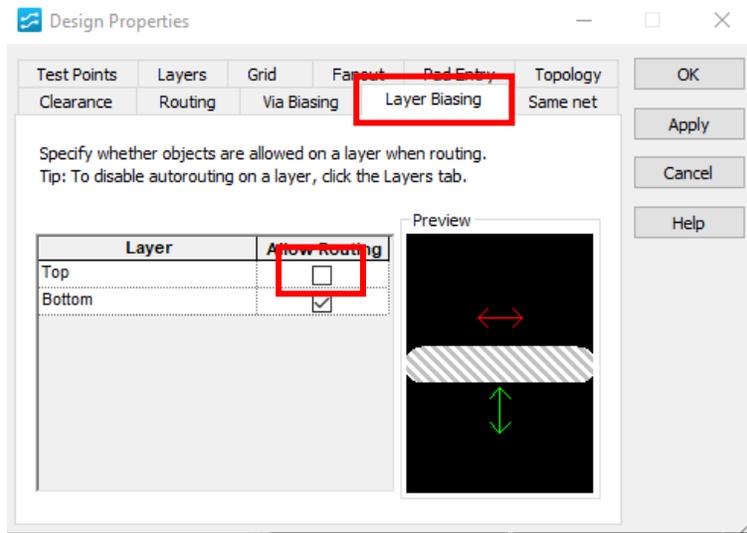
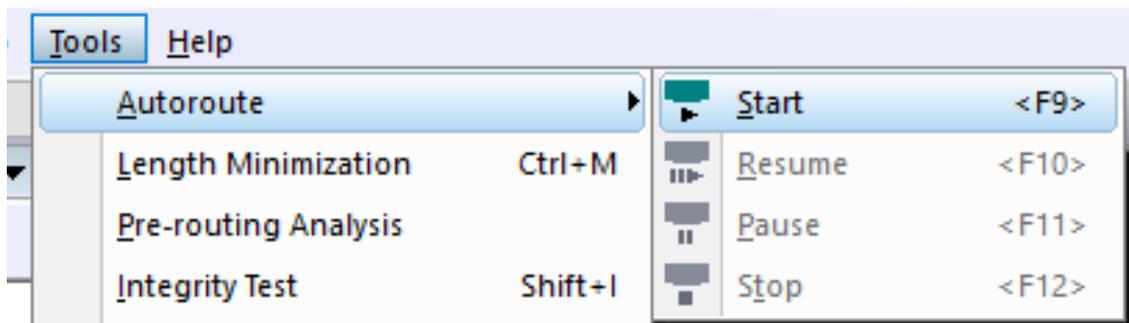


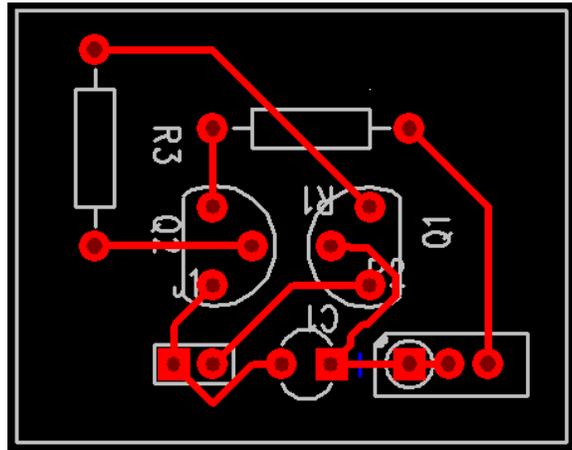
Figure 57: Setup AutoRoute

NOTE: This procedure to make AUTOROUTE run on the bottom layer. In this example the circuit is quite simple, so we do not need to make traces on the top layer. For a more complicated circuit, leave everything as default.

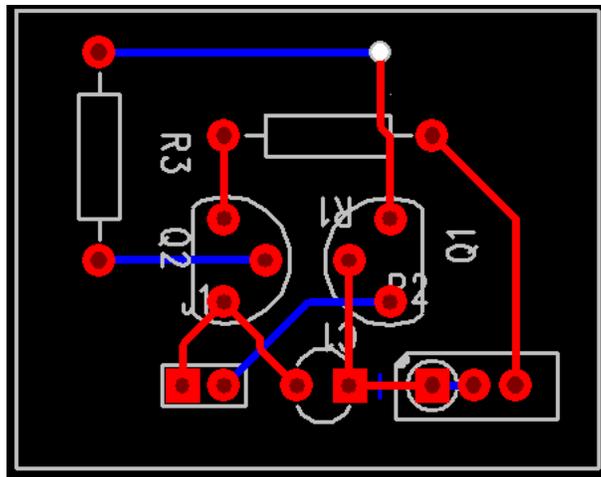
- c. Navigate to Menu Toolbar, select Tools\Autoroute\Start



d. The result using AUTOROUTE with only bottom layer



e. The result using AUTOROUTE on both Bottom and Top layer



14. MEASURE BOARD DIMENSION

a. On Standard Toolbar, click on PADS Layout to return to Layout



Figure 61: Switch back to PADS Layout

b. Check display unit

- Navigate to Menu Toolbar, select Tool\Options

- Change the Design units to Inches (or Metric base on reference)



- Click OK
- Move the mouse cursor to a corner of board outline

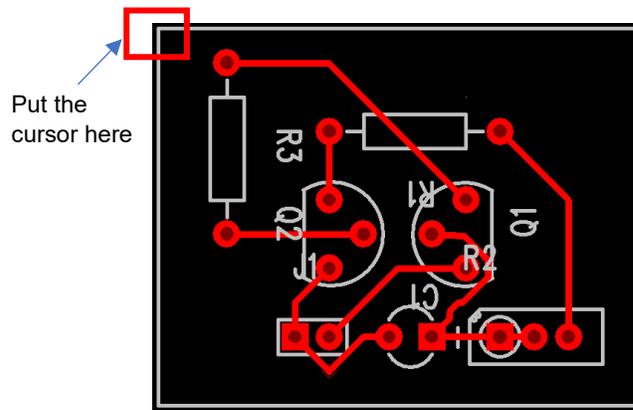
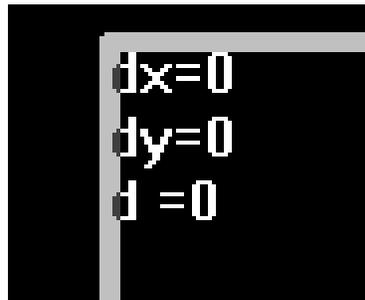


Figure 63: Ready the cursor to measure board dimension

- Press Q then Enter on the keyboard
- The cursor should show dx, dy, d as below



- Move the mouse to the bottom right corner of the board outline

- The dimension should display as below

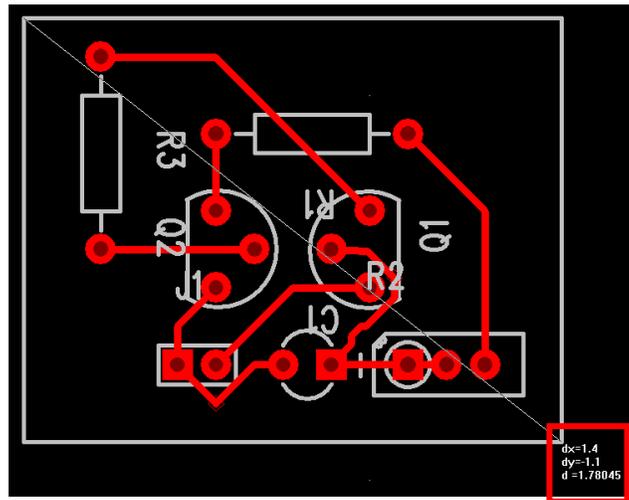


Figure 65: Board dimension

dx = x dimension, dy = y dimension (the unit is selected in step b)

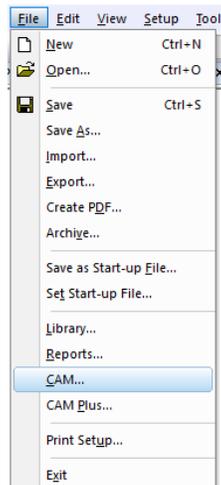
15. EXPORTING CAM DOCUMENTS

- Navigate to Standard Toolbar, select PADS Layout to return to Layout



Figure 66: Go back to PADS Layout

- Navigate to Menu Toolbar, select Files\CAM



c. Click Add in the popup window

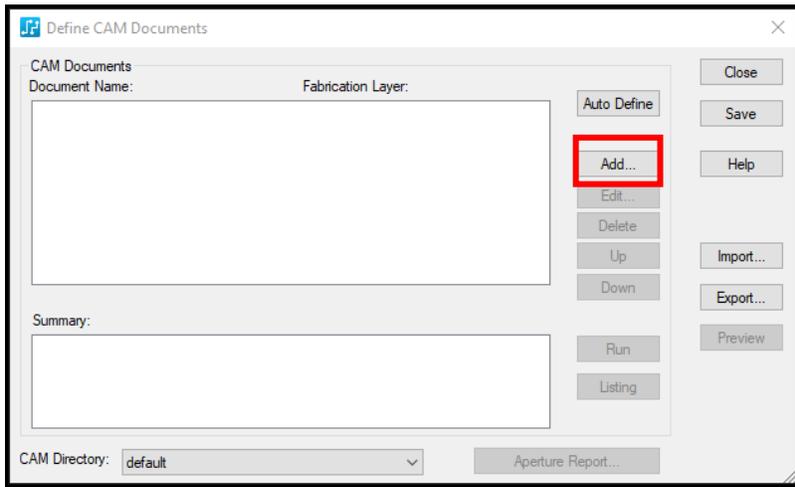


Figure 68: Add CAM Documents

d. Set the option as below

- Change the Document Name: bottom
- Change the Document Type: Routing/Split Plane
- Change the Output File: bottom.pho
- Change Fabrication Layer: Bottom
- Click OK after everything is set

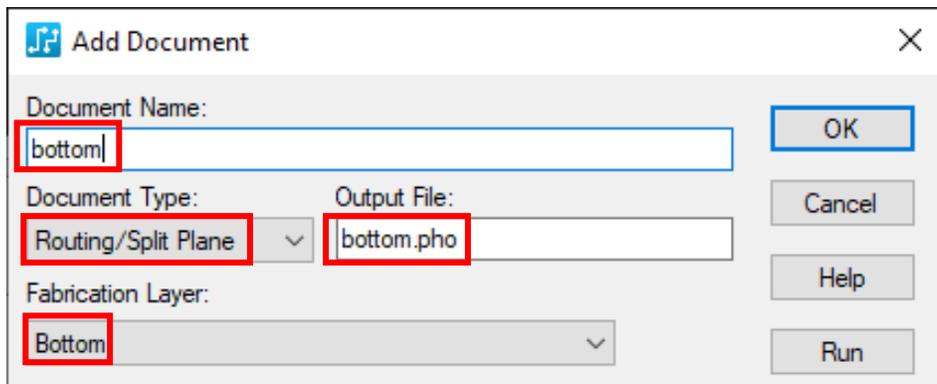


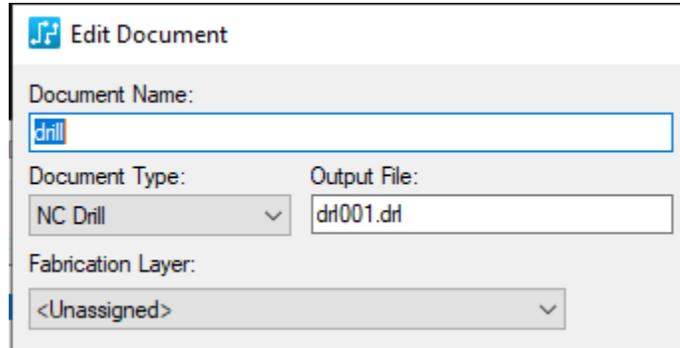
Figure 69: Setup bottom Document Parameters

e. Click Add again at Define CAM Documents window

f. Set the option as below

- Document Name: drill
- Document Type: NC Drill

- Output File: drl001.drl
- Fabrication Layer: unassigned
- Click OK



- g. Click Add again a Define CAM Document window
- h. Click Layers in Add Document window

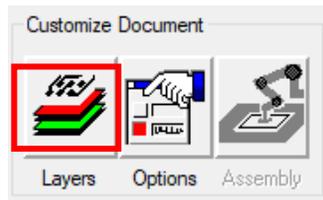


Figure 71: Select Customize Document

- i. Check the Board Outline box in the Select Items window

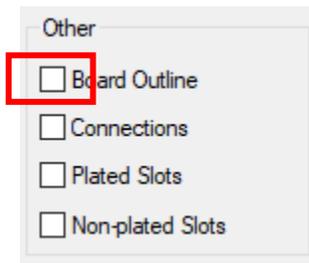


Figure 72: Add Board outline CAM file

- j. Click OK
- k. Set the options as below in Add Document window
 - Document Name: outline

- Document Type: Custom
- Output File: outline.pho
- Fabrication Layer: Unassigned
- Click OK

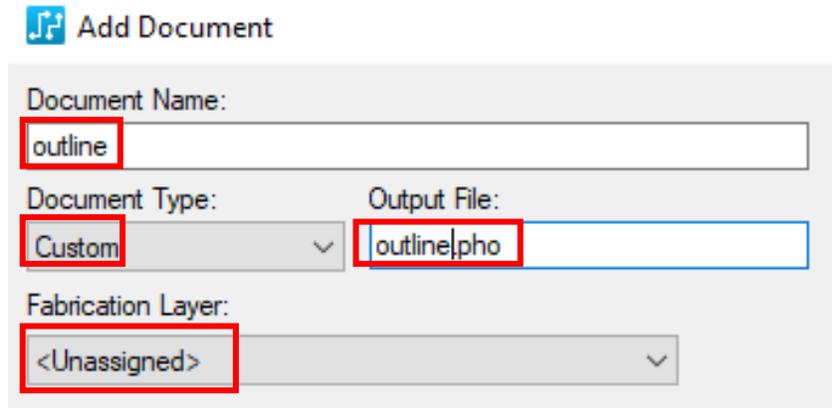
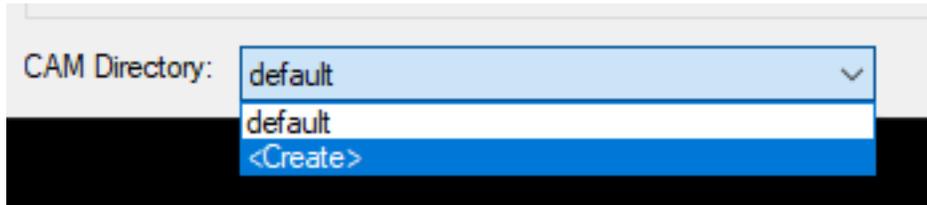


Figure 73: Setup outline document parameters

- I. Back in the Define CAM Documents window, do the following
 - Highlight everything in the Document Name window
 - Click on the drop-down menu CAM Directory, select Create



- In CAM Question window, select Browse, navigate to an appropriate folder to store

- Click Run, then Yes

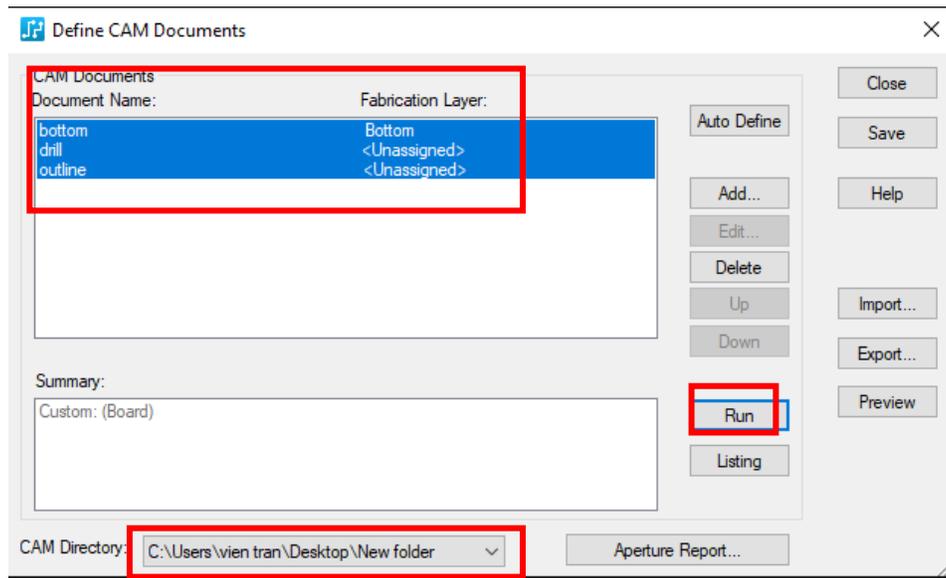
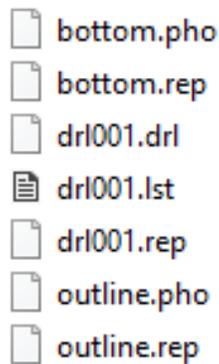


Figure 75: Run CAM files

- Click Close then Yes to save the CAM files
- Using File Explorer, open the folder that was selected in step I to check if all the files are present

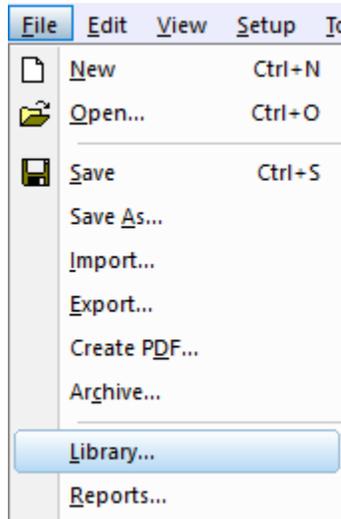


This concludes the process of creating the CAM files for exporting to a PCB machine.

Although PADS has an extensive library, there may be a few special cases in which a component you will need does not exist. If this is the case, follow the instructions below to create a new library and custom components.

16. CREATE NEW LIBRARY

- a. Open PADS Logic
- b. Navigate to Menu Toolbar, select File\Library



- c. Click Create New Lib...

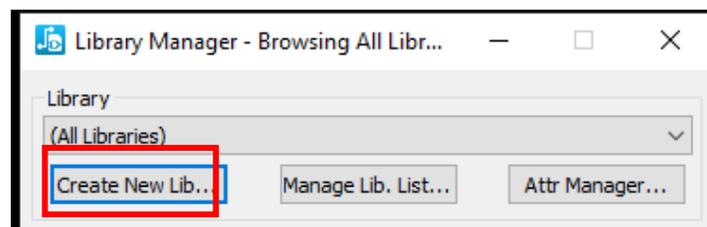


Figure 78: Create new Library

- d. Change the name of the library to user.pt9
- e. Save the library to D:\PADS Library or personal USB Stick

17. MAKE CUSTOM COMPONENTS

- a. Obtain datasheet drawing of the component. We will use a 4-pin tactile push button to demonstrate.

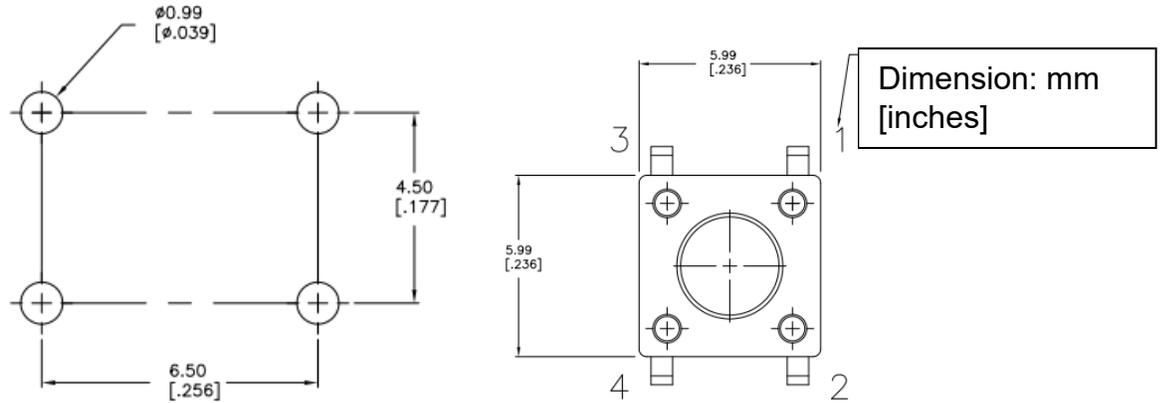
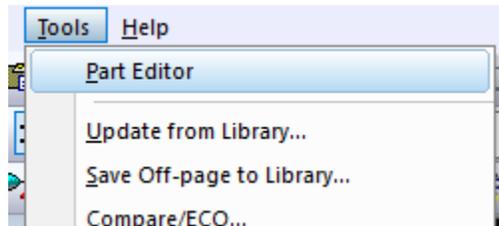


Figure 79: Push button dimension

- b. Open PADS Logic and start a new project
- c. Navigate to Menu Toolbar, select Tools\Part Editor



- d. In "Part:NEW_PART" window, navigate to Part Editor Toolbar then select Edit Graphic



e. Click OK in the next window



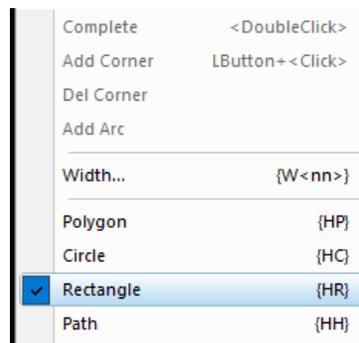
f. Navigate to Symbol Editor Toolbar, click Symbol Editing Toolbar to enable it.



g. Navigate to Symbol Editing Toolbar (SET), click Create 2D Line



h. Right click on an empty area of workspace, select Rectangle



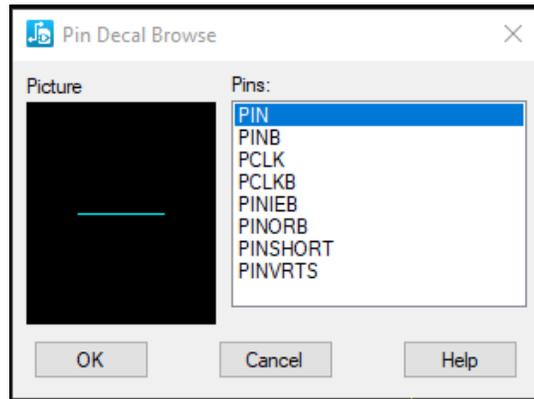
i. Draw a rectangle, then press ESC on the keyboard.

j. On SET, select Add terminal

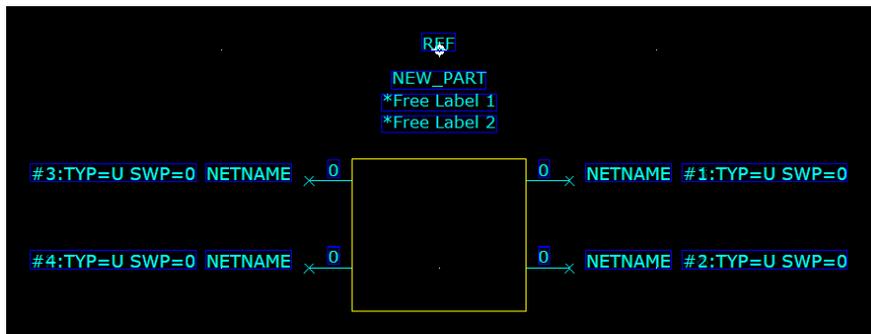


Figure 86: Add terminal

k. Select option PIN in the next window



l. Add for pin on the rectangle



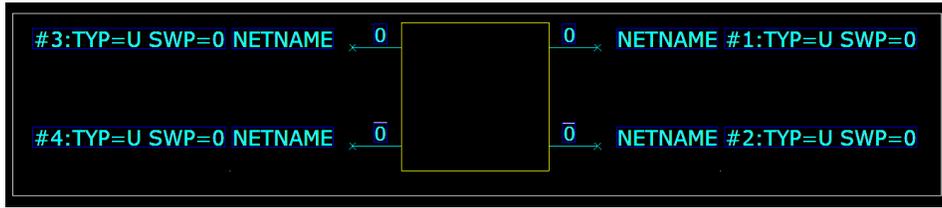
Note: **Make sure the order of the pins match the datasheet**

m. Move the pins and rectangle closer the label "NEW_PART"

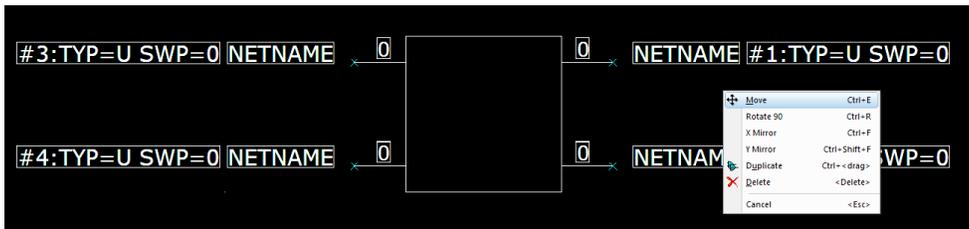
- Move the two labels, "Free Label 1" and "Free Label 2", away



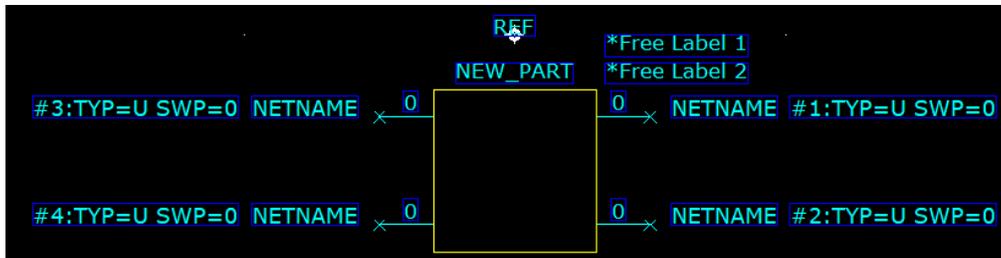
- Left click, hold and drag to select the part and pins



- Right click, then select move



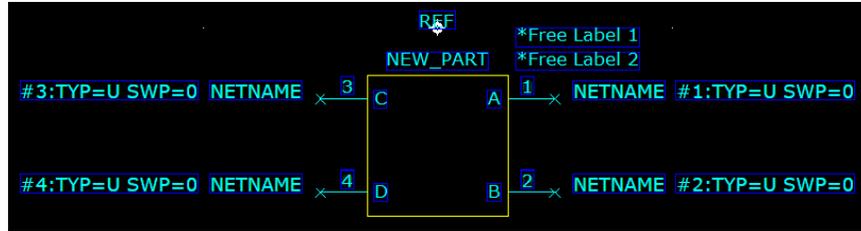
- Move the whole block closer to the label “NEW_PART”



- n. On SET, select Change Pin Number



Figure 93: Change Pin Number tool



Note: A, B, C, D is only used for demonstration since the push button does not have name for its pins. For other components, like connector or IC, put pin names according to their datasheet

- u. Navigate to Menu Toolbar, select File\Return to Part. Click Yes to save
- v. Navigate to Part Editor Toolbar, select Edit Electrical



- w. In Part Information for Part window
 - Select Logic Family: ANA
 - Ref Prefix: U
 - Click OK
- x. Navigate to Menu Toolbar, select File\Save as
- y. In Save part and Gate Decal as, edit as below then click OK

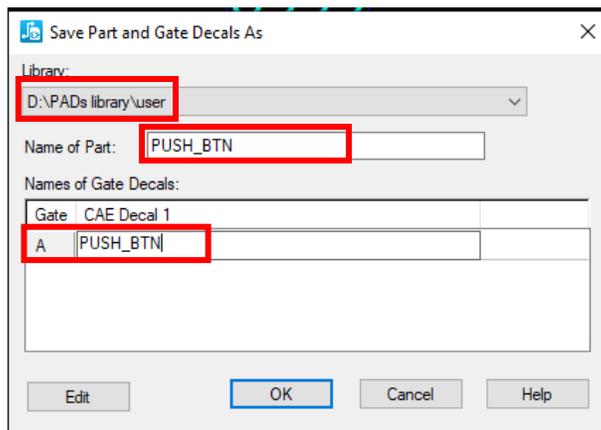


Figure 100: Saving new part

- z. Navigate to Menu Toolbar, select File\Exit Part Editor

18. MAKE CUSTOM LAYOUT DECAL

- a. Run PADS Layout



- b. Navigate to Menu Toolbar, select File\Library
- c. Click on “Manage Lib. List...”

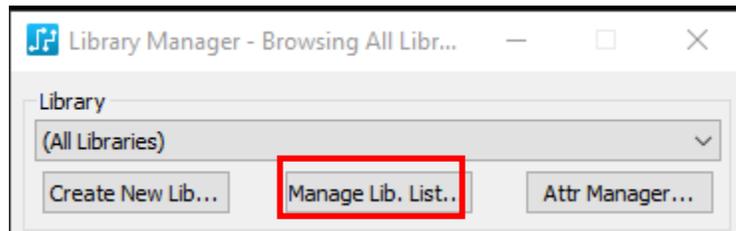


Figure 102: Managing Library list

- d. Click Add

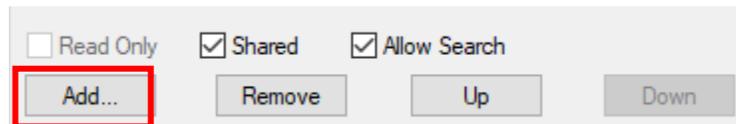


Figure 103: Add library

- e. Navigate to D:\PADS Library (Or USB drive) and select the created library
- f. In PADS Layout, navigate to Menu Toolbar, select Tools\PCB Decal Editor
- g. Click OK then Close
- h. Navigate to Decal Editor Toolbar, click Drafting Toolbar to enable it

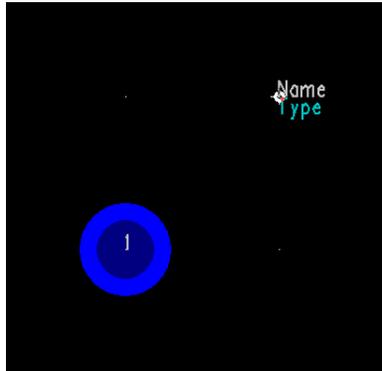


- i. Navigate to Decal Editor Drafting Toolbar, click Terminal to add layout terminal

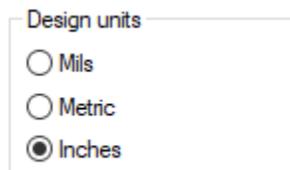


Figure 105: Add layout terminal

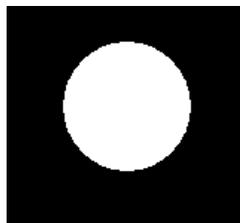
- j. Click OK
- k. Left click on an empty space of the workspace to place a terminal



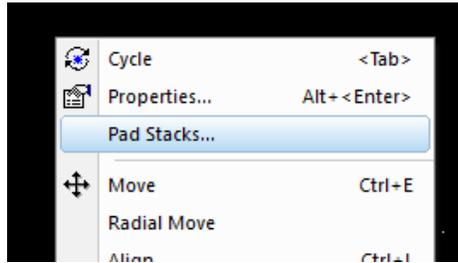
- l. Navigate to Menu Toolbar, select Tools\Option
 - Change the Design Units to Inches



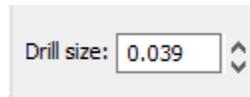
- Click OK
- m. Change drill size of the terminal according to the datasheet
 - Left click to select the terminal (selected terminal should become white)



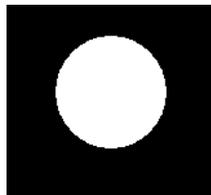
- Right click on an empty area of the workspace, select “Pad Stack...”



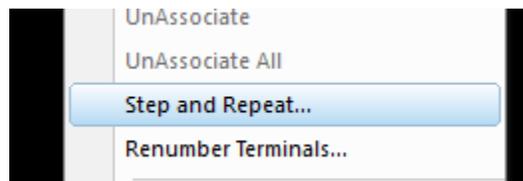
- Change the drill size



- Click OK
- n. Generates other terminals with distance based on datasheet
 - Left click to select the terminal (selected terminal should become white)



- Right click on an empty area of the workspace, select “Step and Repeat...”



- Set the parameter as below then click OK

Direction

Up

Left Right

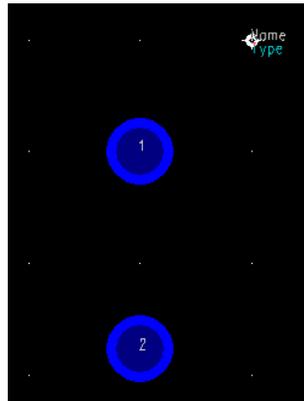
Down

Count: 1

Distance: 0.177

Note: The Count parameter can be set to a number greater than 1 if the part has more than one pin in the same direction.

- Terminal 2 should be generated like below



- Select Terminal 1 and select the “Step and Repeat...” option again
- Set the parameter as below then click OK

Direction

Up

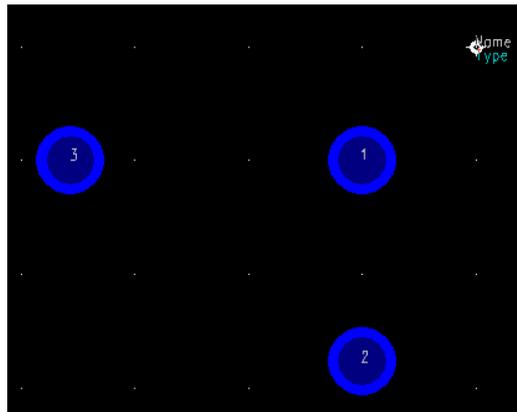
Left Right

Down

Count: 1

Distance: 0.256

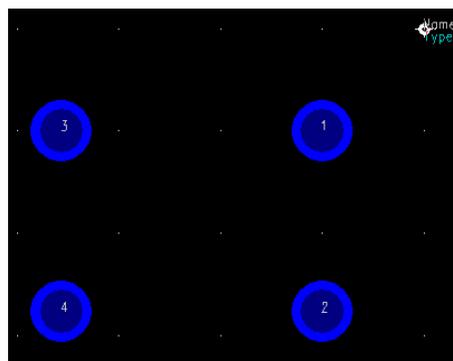
- Terminal 3 should be generated as below



- Select Terminal 3 and select the “Step and Repeat...” option
- Set the parameter as below then click OK



- Terminal 4 should be generated as below



Note: Make sure the numerical order of the terminals match with the datasheet

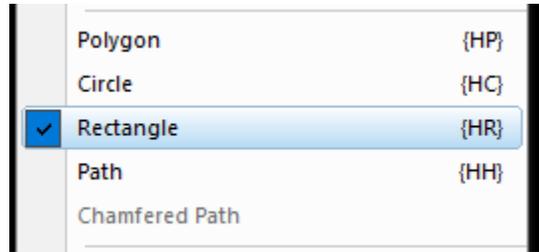
o. Draw part outline

- Navigate to Decal Editor Drafting Toolbar, select 2D Line

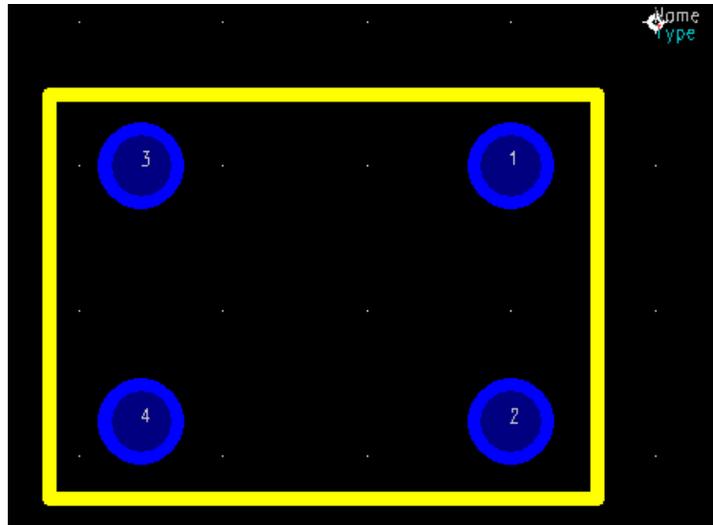


Figure 119: 2D Line to draw part outline

- Right click on an empty area of the workspace, select “Rectangle”



- Left click once then move the mouse the draw a rectangle around the terminals



- Navigate to Menu Toolbar, select Setup\Origin

- Left click to put the origin like below

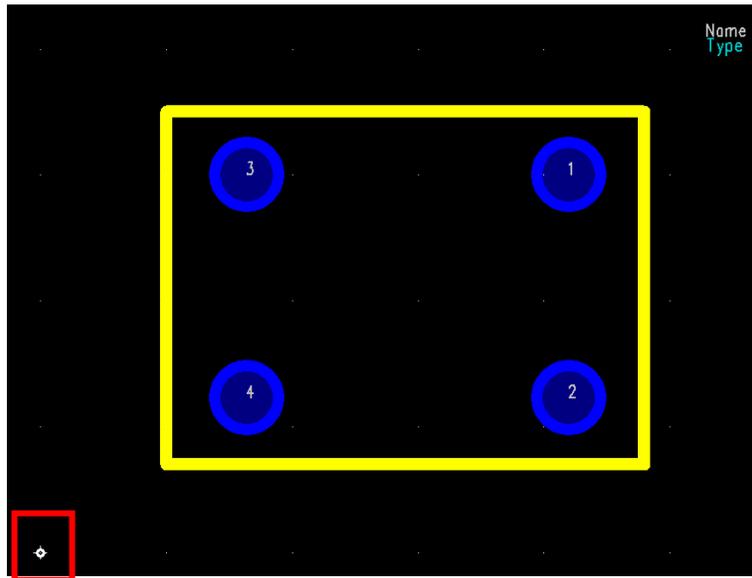
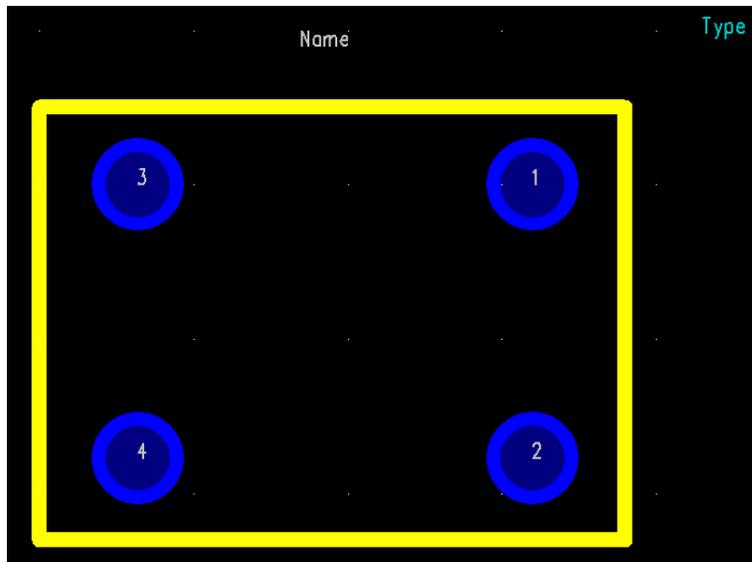
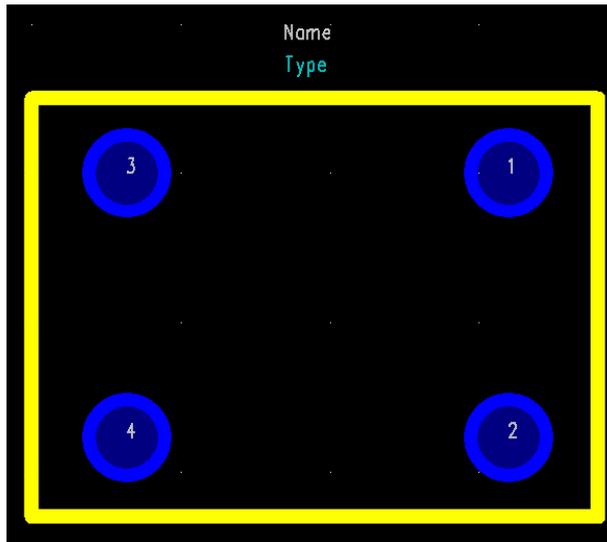


Figure 122: Set new origin

- Left click and hold to select the label "Name"
- Drag to move it the position like below



- Repeat the previous 2 steps with the label “Type”



- p. Navigate to Menu Toolbar, select File\Save Decal to save
- q. Change the Name of the PCB Decal to “Push_BTN”, change the Library path then click OK

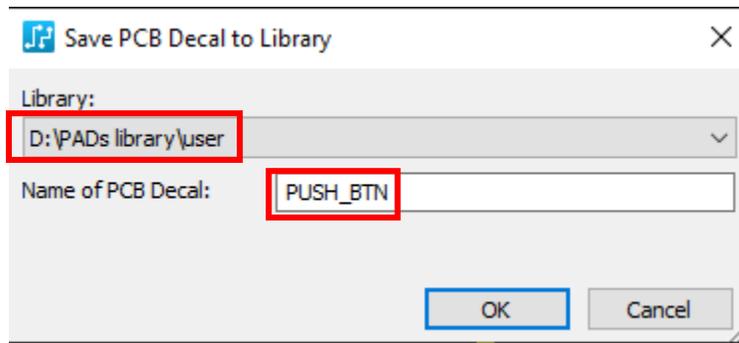


Figure 125: Save new decal to library

- r. Click Yes when prompted to create new Part Type
 - In General tab, set Logic Family to ANA



Figure 126: Set Logic family for new part

- Move to PCB Decals tab, check the Assigned Decals and preview decal

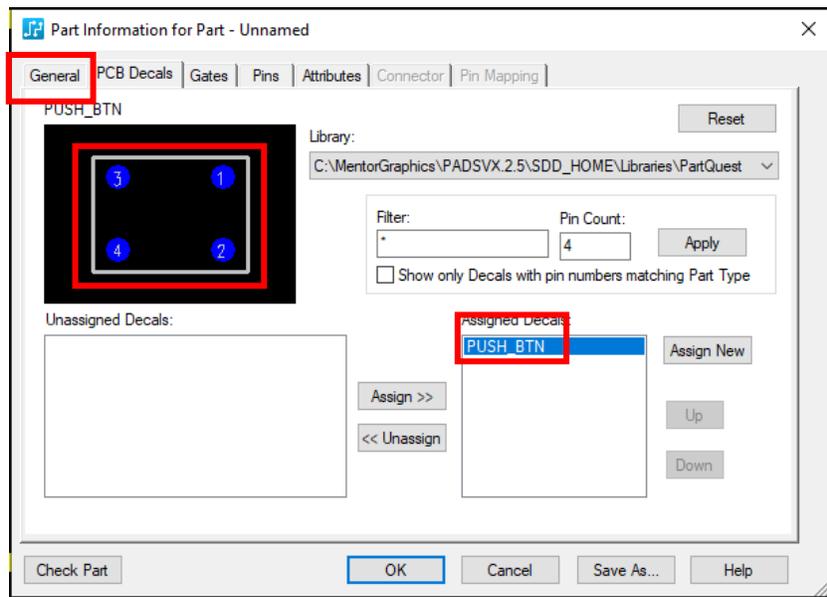


Figure 127: Check decal information

- Move to tab Gates, Click on the empty box under CAE Decal 1

Gate	Pins	Swap	CAE Decal 1	CAE Decal 2	CAE Decal 3	CAE Decal 4
A	4	0				

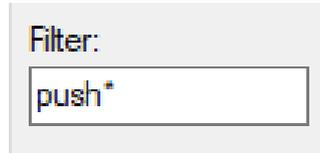
Figure 128: Select Gates decal for new PCB decal

- Select Edit, then click on the “...” beside the empty box

Gate	Pins	Swap	CAE Decal 1	CAE Decal 2	CAE Decal 3	CAE Decal 4
A	4	0				

Figure 129: Select Gates decal for new PCB decal (cont.)

- Type “push” in the Filter box



- Select the decals PUSH_BTN the click Assign>>

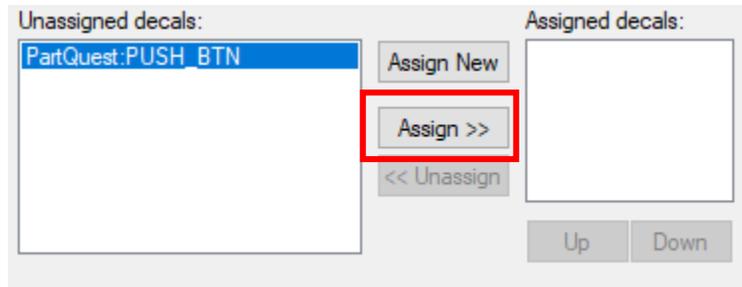


Figure 131: Assign logic decal to PCB decal

- After assigning part decals, click OK

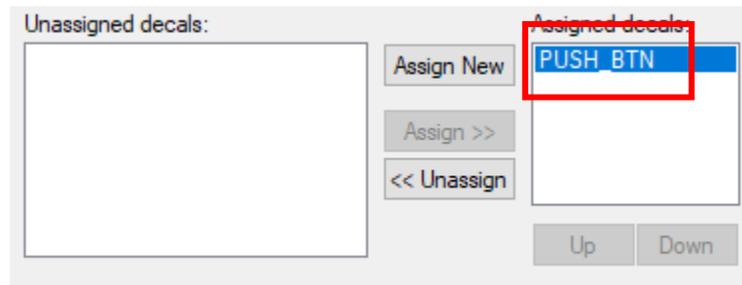


Figure 132: Decal is assigned

- Click OK again
- s. Navigate to Menu Toolbar, click File\Exit Decal Editor
- t. Close PADS Layout

- u. Close PADS Logic

19. CHECK CUSTOM CREATED PART

- a. Open PADS Logic and start a new project
- b. On Menu Toolbar, select File\Library
- c. Click “Manage Lib. List...”
- d. Click Add
- e. Navigate to D:\PADS Library (or USB Drive)
- f. Select “user.pt9” then click Open
- g. Click OK then Close
- h. Click Add part on Schematic Editing Toolbar
- i. Search for PUSH_BTN
- j. Click Add

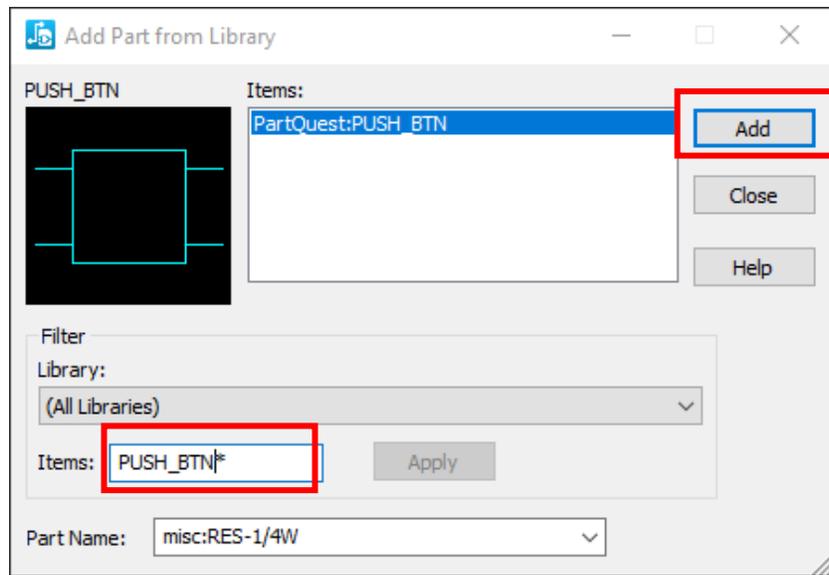


Figure 133: Add new created push button logic part

k. Search for Resistor then add 4 resistors

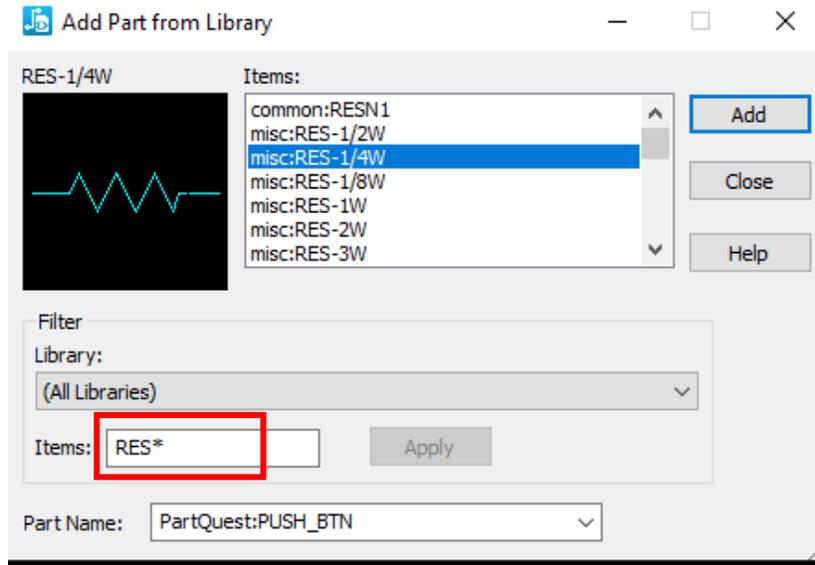
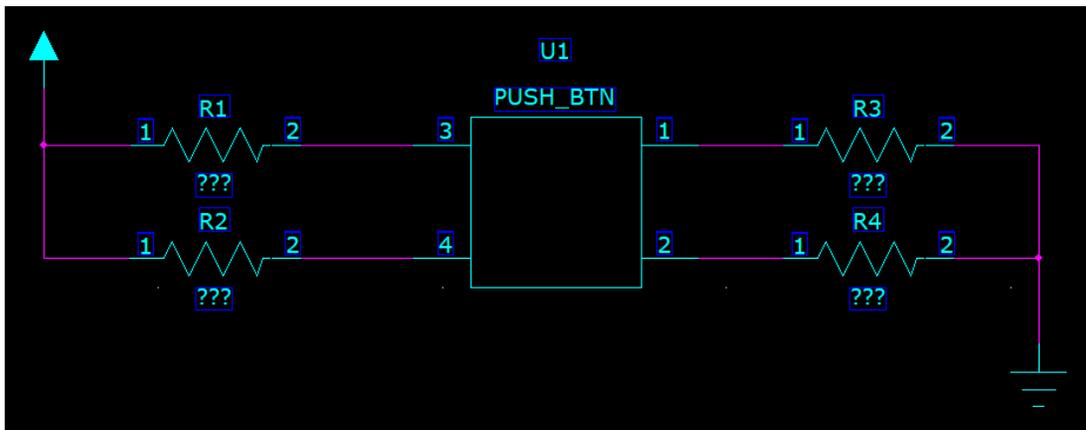


Figure 134: Add resistor

l. Make a simple circuit as below



- m. On Standard Toolbar, click PADS Layout
- n. Click New then Switch To
- o. After PADS Layout has launched, go to PADS Layout
- p. Repeat from step b to g to add the library for PADS Layout

q. Select tab Design then click Send Net list

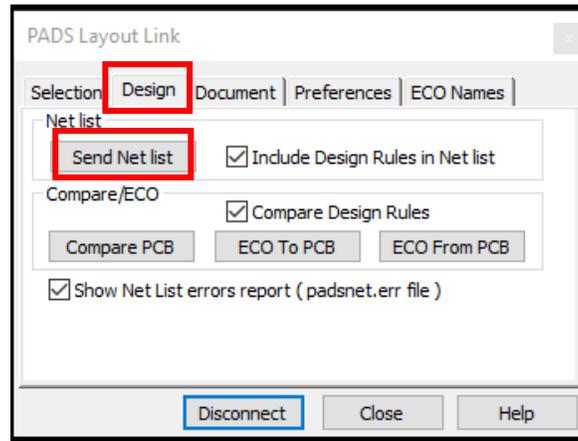


Figure 136: Send netlist to PADS Layout

- r. Go to PADS Layout, the component should be there
- s. On Menu Toolbar, select Tools\Disperse Components
- t. Click Yes

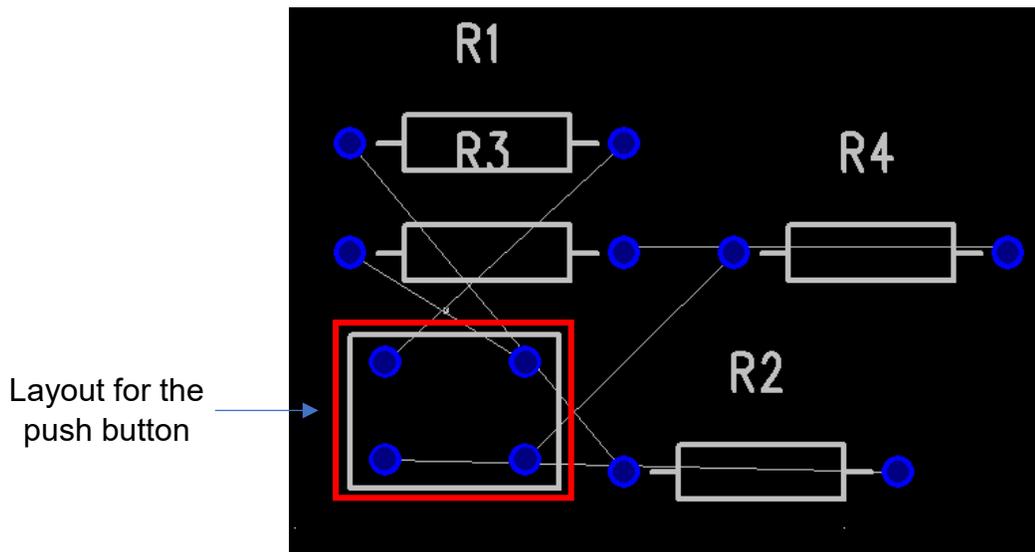
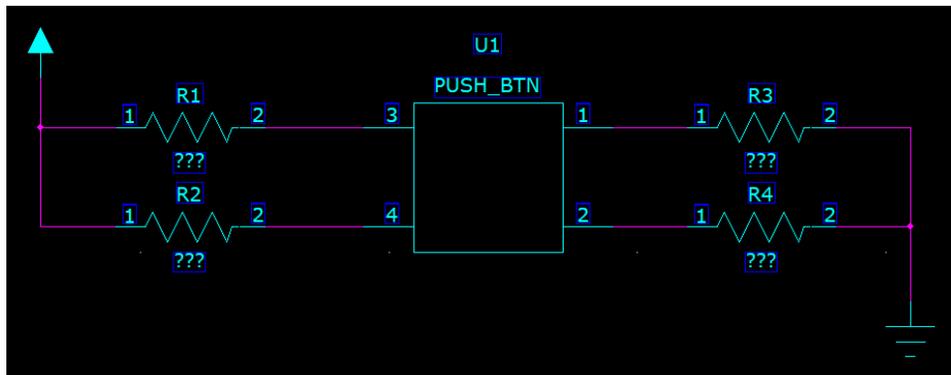
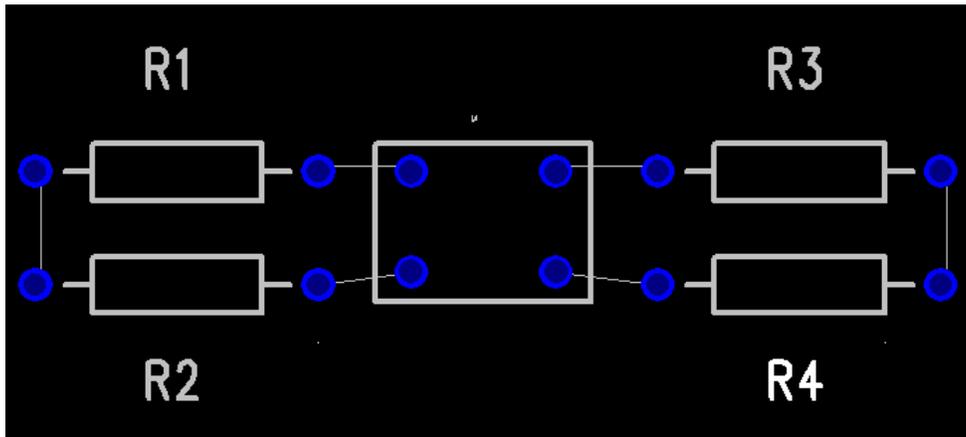


Figure 137: Verify push button layout is showing

u. Arrange the components to verify the connection match with PADS Logic circuit



- R1 is connected to PIN 3
- R2 is connected to PIN 4
- R3 is connected to PIN 1
- R4 is connected to PIN 2