
មូលដ្ឋានឌីហ្សាញបត PCB

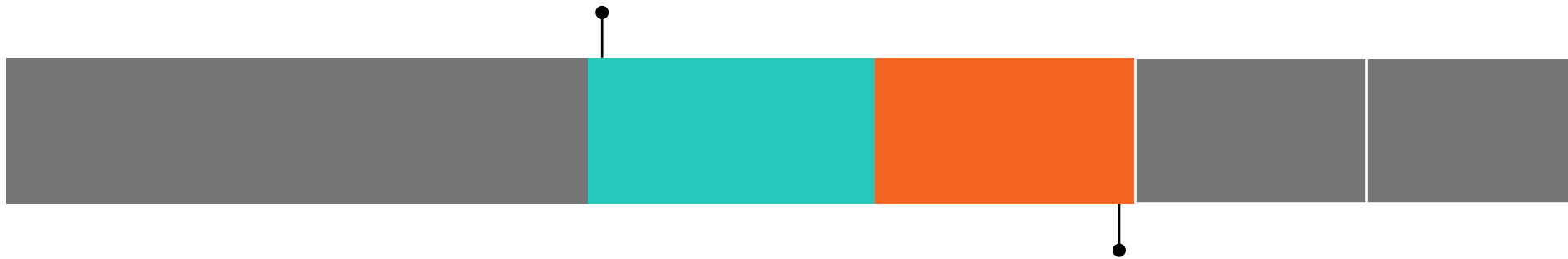
ជាមួយ KiCAD

Online Course

Fundamental



1



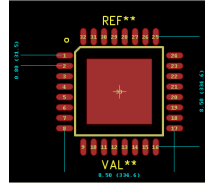
ដើម្បីឱ្យហ្សាញបានជាបន្ទះ PCB ត្រូវឆ្លងកាត់ ៣ ដំណាក់កាលខាងក្រោម

1



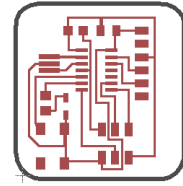
Schematic

2



Footprint

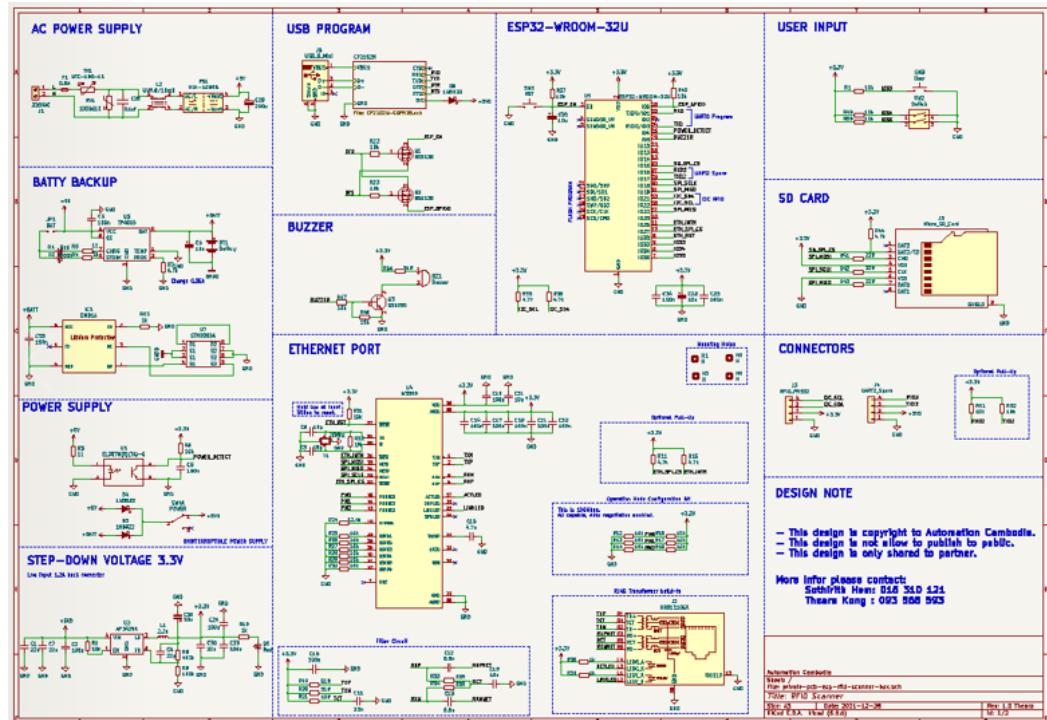
3



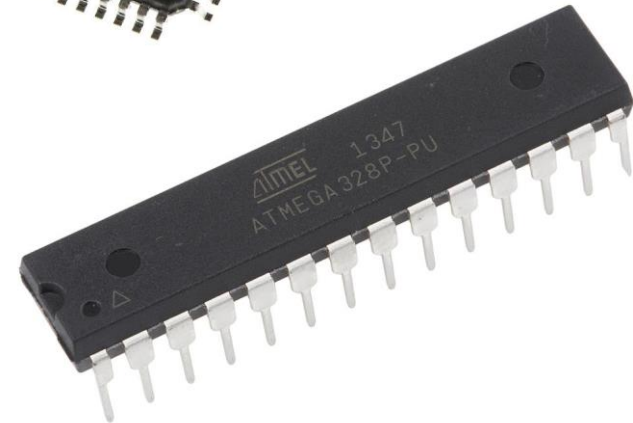
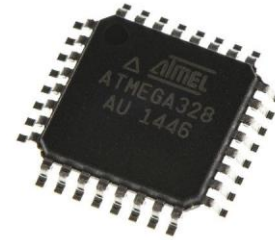
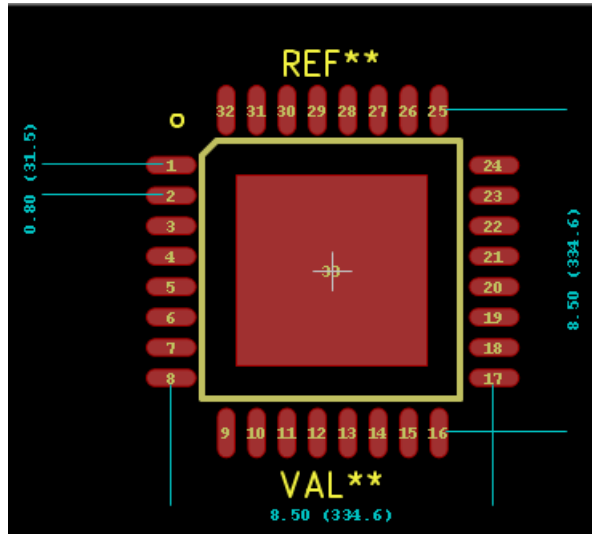
PCB

ដំណាក់កាលដើម្បីច្បាស់ជា PCB

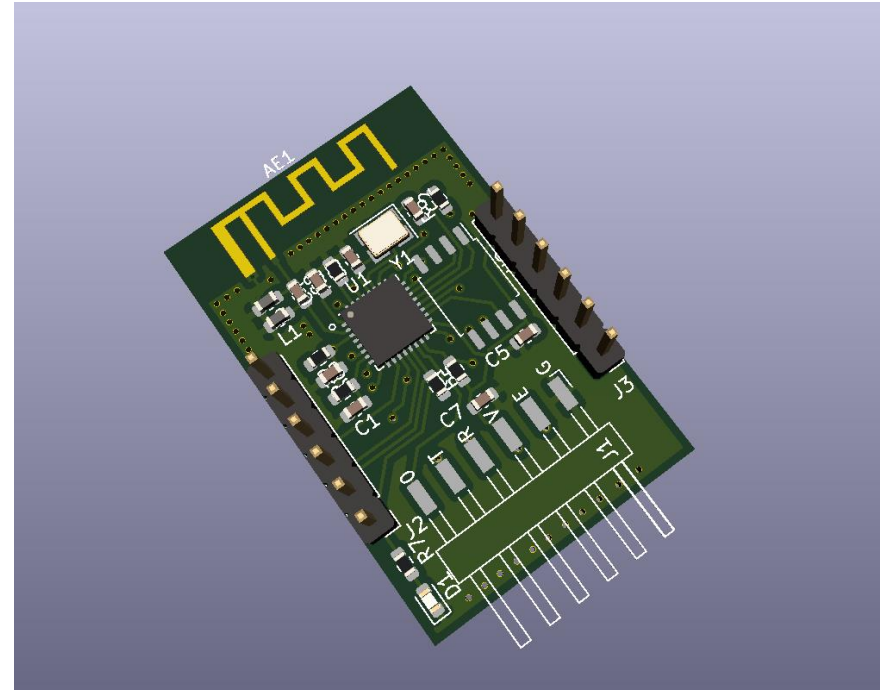
The schematic diagram illustrates the internal circuitry of the MAX3221CPE module. It features a MAX3221CPE integrated circuit (IC1) at the center, which is a dual in-line package (DIP) with pins labeled 1 through 15. The IC is connected to a target system via a 15-pin connector labeled 'Target ES232' and 'SUB-D Female'. The target system provides power (+Ub, +3V3, +5V) and ground (GND) connections. The module's internal components include a 1MΩ resistor (R2), a 47nF capacitor (C1), a 3k6 resistor (R3), a 33k resistor (R4), a 100 resistor (R10), a 33k resistor (R8), a 3k6 resistor (R7), a 1MΩ resistor (R6), a 470nF capacitor (C3), a 470nF capacitor (C2), a 470nF capacitor (C4), a 100nF capacitor (C7), and two diodes (D1, D2) labeled 'PATW6'. The module is powered by a +3V3 supply and has a +5V supply connected to its Vcc pin. The module's ground (GND) is connected to the target system's ground. The module's output pins are connected to the target system's UART1_TX and UART1_RX pins. The module's VSS pin is connected to the target system's VSS (Target) pin.



2. Footprint



ដំណាក់កាលដើម្បីច្បាស់ជា PCB



៤ ចំណុចដើម្បីអាចចេះឌីហ្សាញ PCB

1



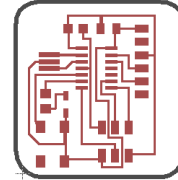
ចេះប្រើកម្មវិធី

2



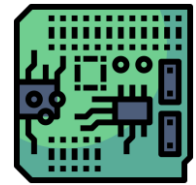
អេឡិចត្រូនិច

3



ឌីហ្សាញ

4



ផលិត

1. កម្មវិធី

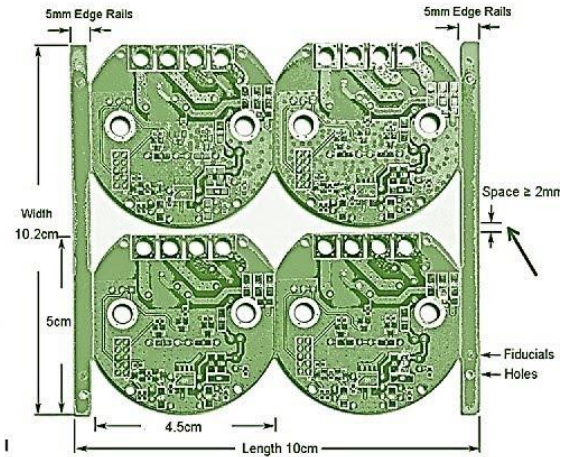
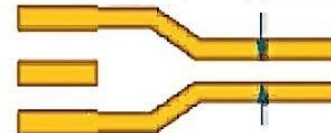
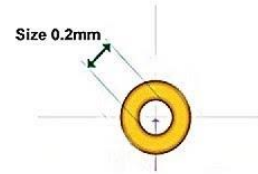
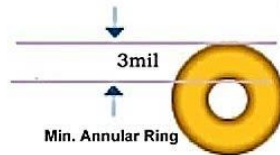


3. ឌីហ្សាញ PCB

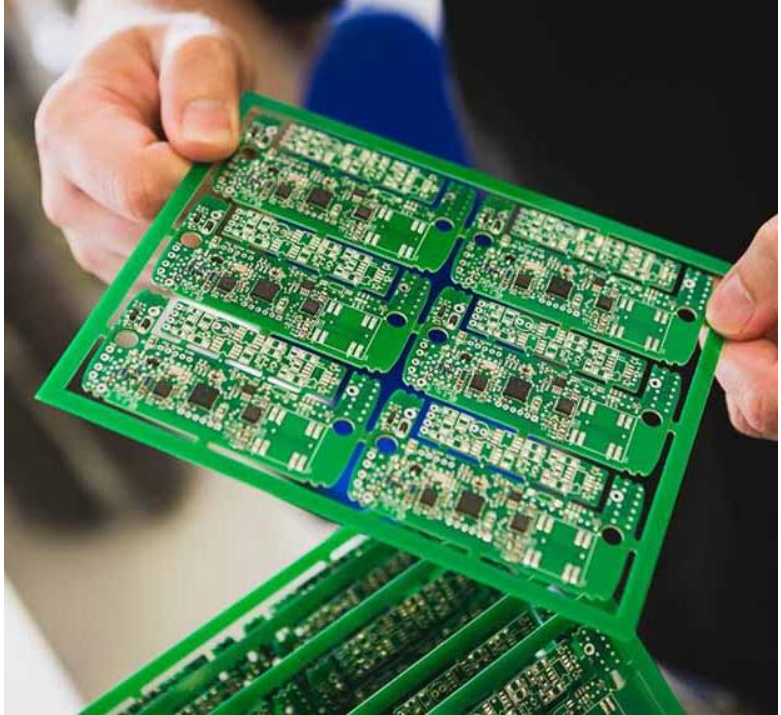
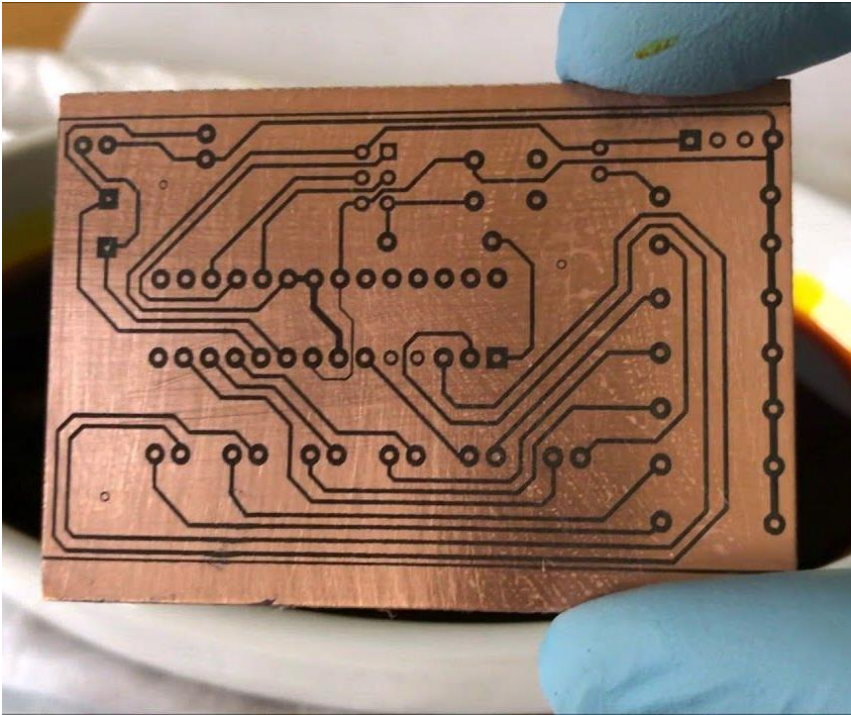
Basic PCB Design Rules

Clearance					
All	Trace	Via	Pad	SMD	Copper
Trace	0.1524				
Via	0.1524	0.1524			
Pad	0.1524	0.1524	0.1524		
SMD	0.1524	0.1524	0.1524	0.1524	
Text	0.1524	0.1524	0.1524	0.1524	
Copper	0.1524	0.1524	0.1524	0.1524	0.1524
Board	0.1524	0.1524	0.1524	0.1524	0.1524
Drill	0.1524	0.1524	0.1524	0.1524	0.1524

C1
32mil
6mil



4. ផលិតបន្ទះ PCB



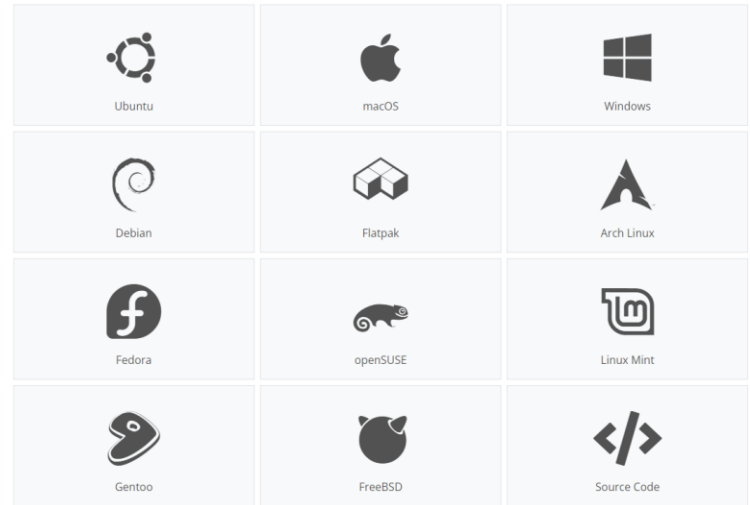


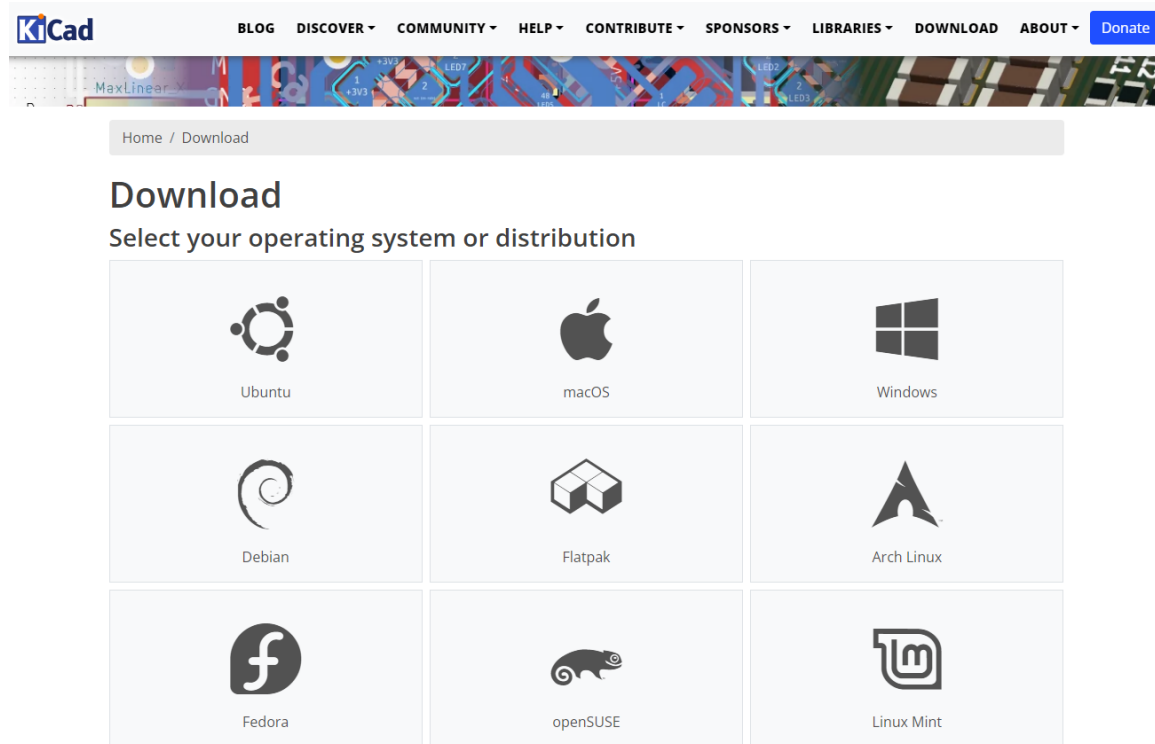
- មិនគិតថ្លៃ (មិនជាប់ License)
- មានស្ទើរគ្រប់ OS
- ក្រុមហ៊ុនធំៗទាំងក្នុង និងក្រៅស្រុកប្រើច្រើន
- សំទុះអ្នកប្រើប្រាស់កាន់តែច្រើន

KiCad Licenses

Program License

KiCad is free software. The majority of KiCad's source code is developed and distributed under the GNU General Public License (GPL) version 3 or greater. Additional third party licenses may also exist in the code base such as the MIT License, see the source code for complete and up to date details.

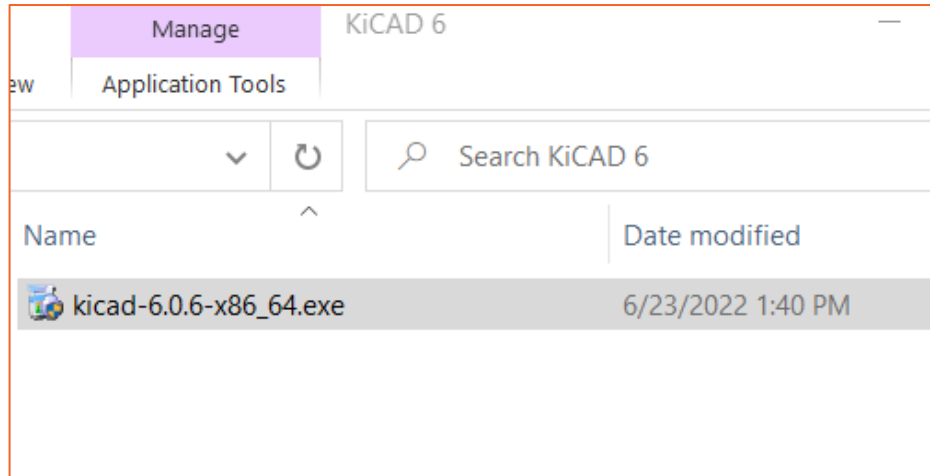




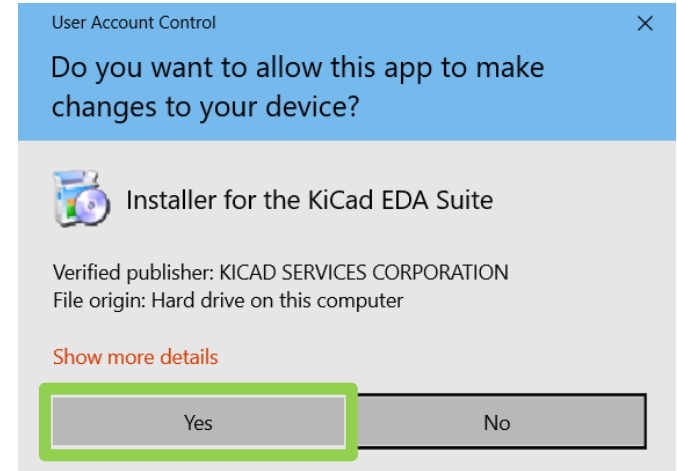
1. ទាញយកកម្មវិធី [Download](#) ជ្រើសរើសតាម OS

The screenshot shows the KiCad website's 'Windows Downloads' page. The header includes the KiCad logo and navigation links: BLOG, DISCOVER, COMMUNITY, HELP, CONTRIBUTE, SPONSORS, LIBRARIES, DOWNLOAD, ABOUT, and a blue 'Donate' button. Below the header, a breadcrumb trail reads 'Home / Download / Windows Downloads'. The main heading is 'Windows Downloads', followed by a note: 'KiCad supports Windows 8.1, 10 and 11. See [System Requirements](#) for more details.' Below this is the 'Stable Release' section, indicating the 'Current Version: 6.0.6'. A tab labeled '64-bit (recommended)' is active. The page is organized by region: 'Worldwide' (with links to OSDN and GitHub), 'Europe' (with links to CERN - Switzerland and Futureware - Austria), and 'China' (with links to AlibabaCloud and Chongqing University). On the right side, there is a sidebar titled 'All Platforms' with a list of operating systems: Ubuntu, macOS, Windows (highlighted in blue), Debian, Flatpak, Arch Linux, Fedora, openSUSE, Linux Mint, Gentoo, FreeBSD, and Source Code.

2. ជ្រើសរើស Server ទាញយកដែលស្ថិតនៅជិតអ្នក



3. Double Click លើកម្មវិធីដែលបានទាញយក



4. ចុច Yes ដើម្បីដំណើរការ

3. Double Click លើកម្មវិធីដែលបានទាញយក



Schematic Editor

Edit the project schematic



Symbol Editor

Edit global and/or project schematic symbol libraries



PCB Editor

Edit the project PCB design



Footprint Editor

Edit global and/or project PCB footprint libraries



Gerber Viewer

Preview Gerber files



Image Converter

Convert bitmap images to schematic symbols or PCB footprints



Calculator Tools

Show tools for calculating resistance, current capacity, etc.



Drawing Sheet Editor

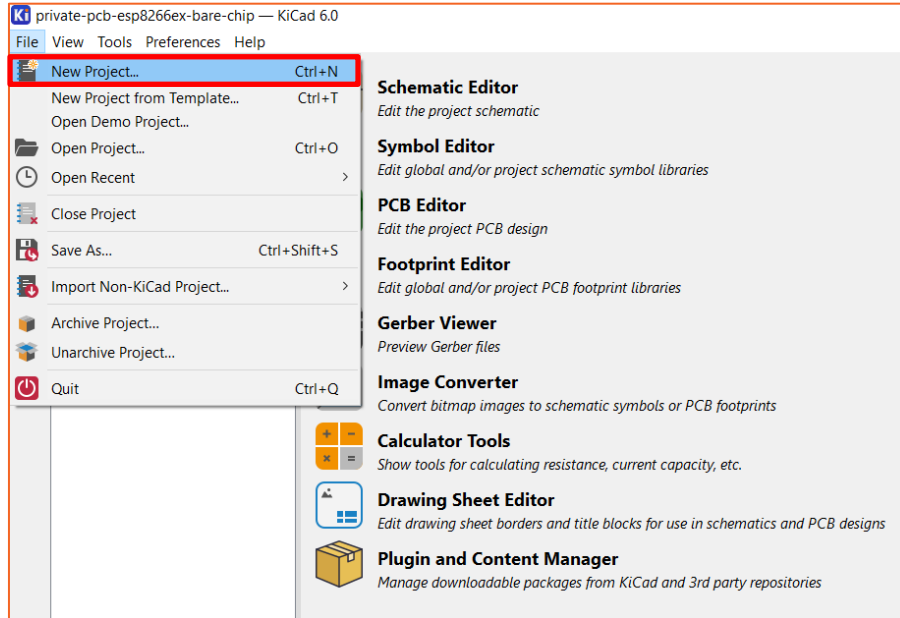
Edit drawing sheet borders and title blocks for use in schematics and PCB designs



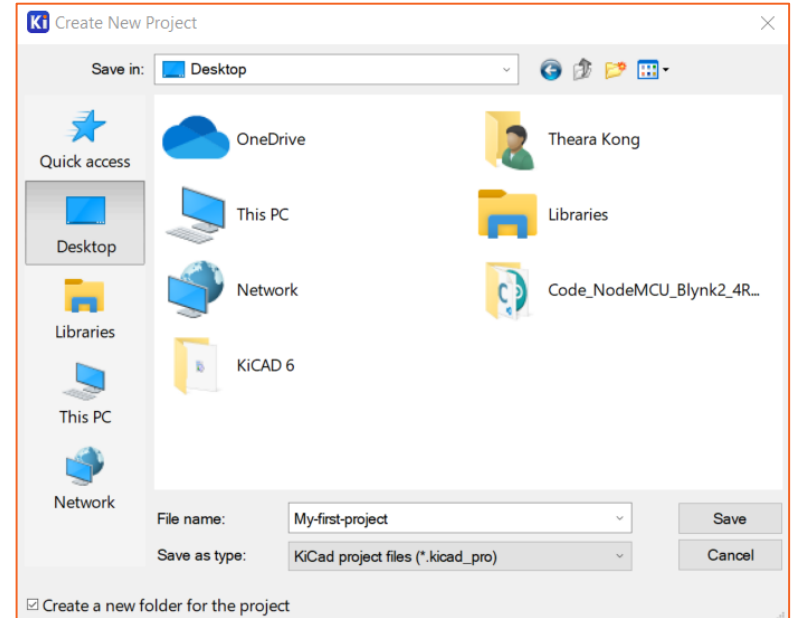
Plugin and Content Manager

Manage downloadable packages from KiCad and 3rd party repositories

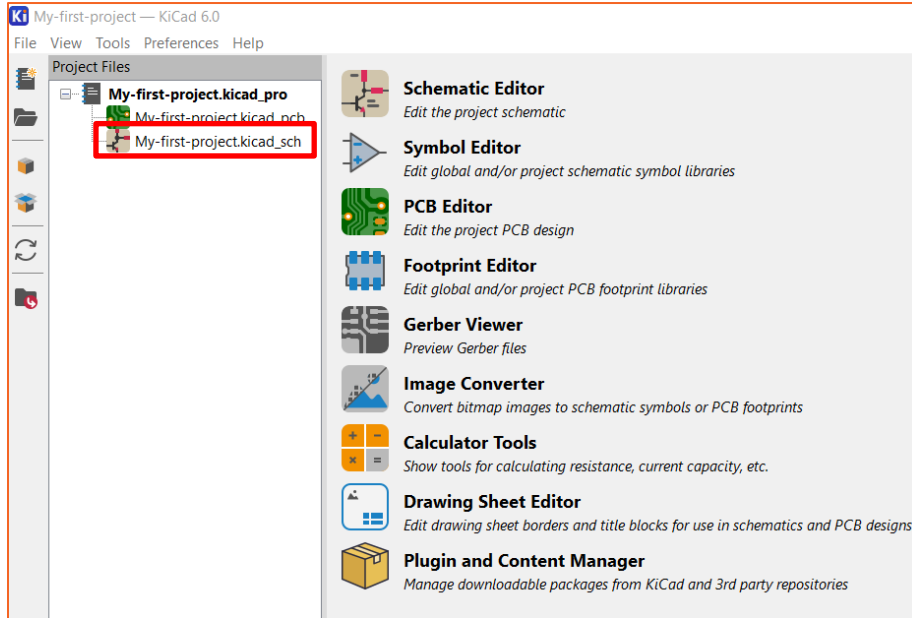
ចូលទៅក្នុង Symbol Editor ដើម្បីធ្វើការកែ ឬបង្កើត
Symbol ថ្មី



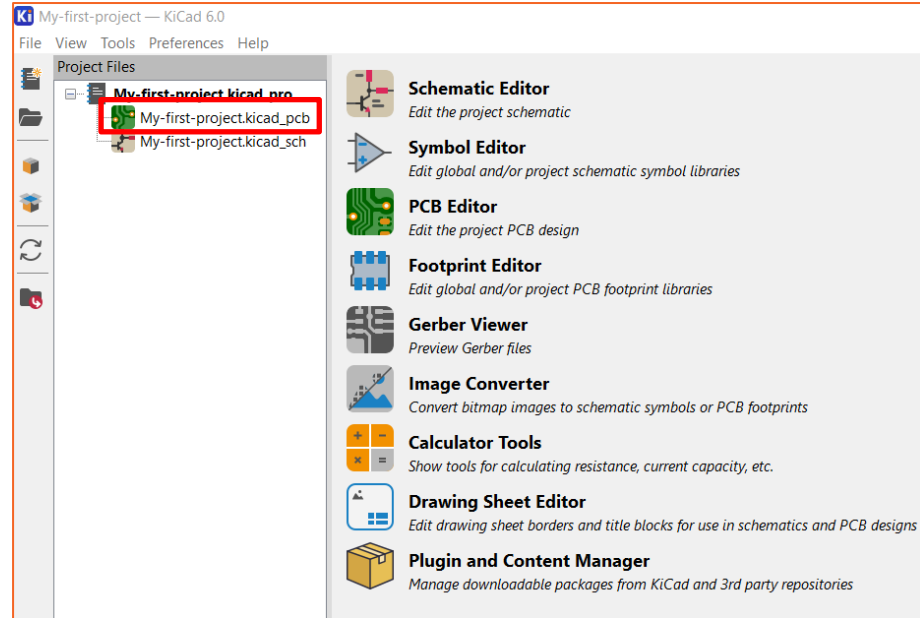
1. ចុចលើ File រួចជ្រើសរើស New Project (Ctrl+N)



2. ដាក់ឈ្មោះឱ្យ Project របស់អ្នក



3. ចុចលើ .kicad_sch ដើម្បីបើក Schematic



4. ចុចលើ .kicad_pcb ដើម្បីបើក PCB

2. Design Software

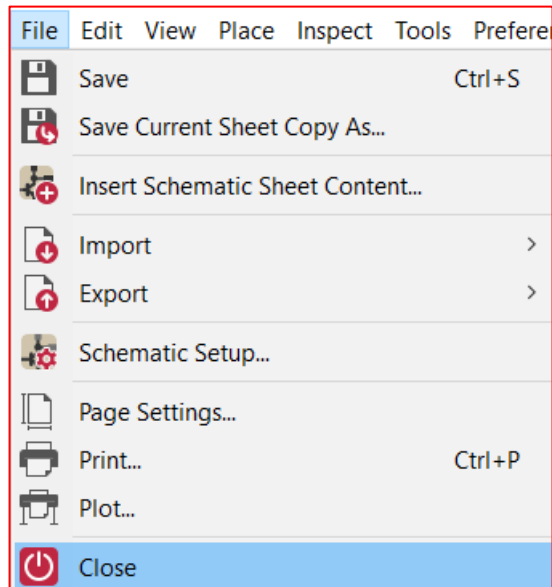
2.3 Tools សម្រាប់ប្រើលើ Schematic

File	Edit	View	Place	Inspect	Tools	Preferences	Help
Save					Ctrl+S		
Save Current Sheet Copy As...							
Insert Schematic Sheet Content...							
Import							
Export							
Schematic Setup...							
Page Settings...							
Print...					Ctrl+P		
Plot...							
Close							
















Place	Inspect	Tools	Preferences	Help
Add Symbol				A
Add Power				P
Add Wire				W
Add Bus				B
Add Wire to Bus Entry				Z
Add No Connect Flag				Q
Add Junction				J
Add Label				L
Add Global Label				Ctrl+L
Add Hierarchical Label				H
Add Sheet				S
Import Sheet Pin				
Add Lines				I
Add Text				T
Add Image				

Inspect	Tools	Preferences	Help
Electrical Rules Checker			
Previous Marker			
Next Marker			
Exclude Marker			
Simulator...			
Configure Paths...			
Manage Symbol Libraries...			
Preferences...			Ctrl+,
Set Language			

Tools	Preferences	Help	
Update PCB from Schematic...		F8	
Update Schematic from PCB...			
Switch to PCB Editor			
Symbol Editor			
Update Symbols from Library...			
Rescue Symbols...			
Remap Legacy Library Symbols...			
Edit Symbol Fields...			
Edit Symbol Library Links...			
Annotate Schematic...			
Bus Definitions...			
Assign Footprints...			
Generate BOM...			

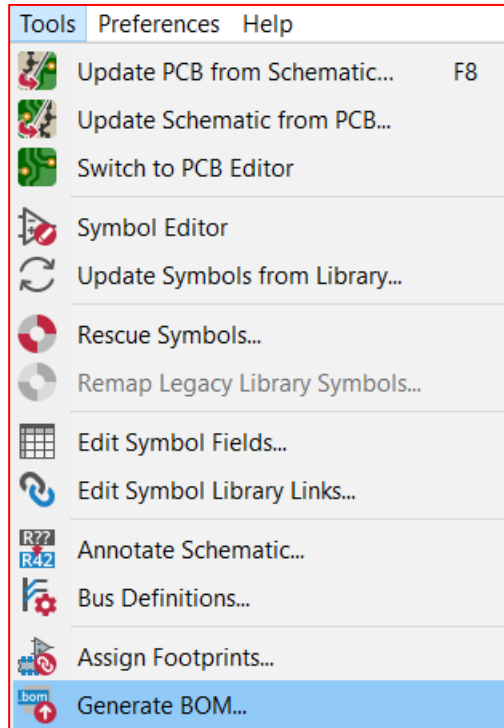


- **Save** រក្សាទុកនៅលើ File ដដែល
- **Save Current Sheet Copy As** រក្សាក្រដាសកំពុង Edit ទៅកន្លែងផ្សេង
- **Insert Schematic Sheet Content** បញ្ចូលក្រដាសដែលមានស្រាប់
- **Import** ដាក់បញ្ចូល File
- **Export** បញ្ចេញ File
- **Schematic Setup** កំណត់ពីសេម៉ា
- **Page Setting** កំណត់ពីទំព័ររបស់សេម៉ា
- **Print** ព្រីនសេម៉ា
- **Plot** បញ្ចេញសេម៉ាទៅជា File ផ្សេងៗ
- **Close** បិទសេម៉ា

Place	Inspect	Tools	Preferences	Help
	Add Symbol			A
	Add Power			P
	Add Wire			W
	Add Bus			B
	Add Wire to Bus Entry			Z
	Add No Connect Flag			Q
	Add Junction			J
	Add Label			L
	Add Global Label			Ctrl+L
	Add Hierarchical Label			H
	Add Sheet			S
	Import Sheet Pin			
	Add Lines			I
	Add Text			T
	Add Image			
















អក្សរនៅខាងស្តាំជា Shortcut ទៅកាន់ Tools នោះ




















- **Add Symbol** ចូលទៅជ្រើសរើស Symbol ផ្សេងដាក់លើសេម៉ា
- **Add Power** ចូលទៅជ្រើសរើស Symbol Power មកដាក់លើសេម៉ា
- **Add Wire** គូសខ្សែភ្ជាប់រវាង Symbol
- **Add Bus** គូសខ្សែ Bus
- **Add Wire to Bus Entry** គូសខ្សែភ្ជាប់ទៅកាន់ខ្សែ Bus
- **Add No Connect Flag** ដាក់និមិត្តសញ្ញាបញ្ជាក់ថាមិនមានភ្ជាប់
- **Add Junction** ដាក់ចំណុចភ្ជាប់ដែលខ្សែភ្ជាប់គ្នា
- **Add Label** ដាក់អក្សរសម្គាល់ខ្សែ(ស្គាល់តែក្នុងក្រដាសកំពុងធ្វើការ)
- **Add Global Label** ដាក់អក្សរសម្គាល់ខ្សែ(ស្គាល់ពីគ្រប់ក្រដាស)
- **Add Hierarchical label** ដាក់អក្សរសម្គាល់របស់ក្រដាស
- **Add Sheet** បង្កើតក្រដាសថ្មី
- **Add Line** គូសបន្ទាត់






















អក្សរនៅខាងស្តាំជា Shortcut ទៅកាន់ Tools នោះ

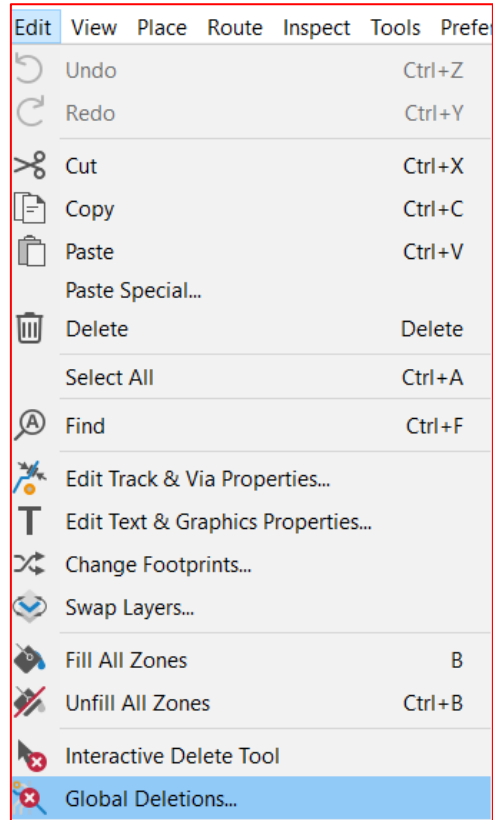
- **Update PCB from Schematic** Update PCB ពីសេម៉ា
- **Update Schematic from PCB** Update សេម៉ា ពី PCB វិញ
- **Switch to PCB Editor** ប្តូរទៅផ្ទាំងកែ PCB
- **Symbol Editor** កែសម្រួល Symbol
- **Annotate Schematic** រៀបចំសេម៉ា
- **Bus Definition** ដាក់ឈ្មោះឲ្យខ្សែ Bus
- **Assign Footprints** កំណត់ Footprints
- **Generate BOM** បង្កើតតារាងរាយចំនួនគ្រឿង Build Of Material

Edit	View	Place	Route	Inspect	Tools	Prefer
	Undo				Ctrl+Z	
	Redo				Ctrl+Y	
	Cut				Ctrl+X	
	Copy				Ctrl+C	
	Paste				Ctrl+V	
	Paste Special...					
	Delete				Delete	
	Select All				Ctrl+A	
	Find				Ctrl+F	
	Edit Track & Via Properties...					
	Edit Text & Graphics Properties...					
	Change Footprints...					
	Swap Layers...					
	Fill All Zones				B	
	Unfill All Zones				Ctrl+B	
	Interactive Delete Tool					
	Global Deletions...					

Place	Route	Inspect	Tools	Preferences	Help
	Add Footprint				A
	Add Vias				Ctrl+Shift+V
	Add Filled Zone				Ctrl+Shift+Z
	Add Rule Area				Ctrl+Shift+K
	Add Microwave Shape				>
	Add Text				Ctrl+Shift+T
	Draw Line				Ctrl+Shift+L
	Draw Arc				Ctrl+Shift+A
	Draw Rectangle				
	Draw Circle				Ctrl+Shift+C
	Draw Graphic Polygon				Ctrl+Shift+P
	Add Aligned Dimension				Ctrl+Shift+H
	Add Orthogonal Dimension				
	Add Center Dimension				
	Add Leader				
	Add Board Characteristics				
	Add Stackup Table				
	Add Layer Alignment Target				
	Drill/Place File Origin				
	Grid Origin				S
	Auto-Place Footprints				>

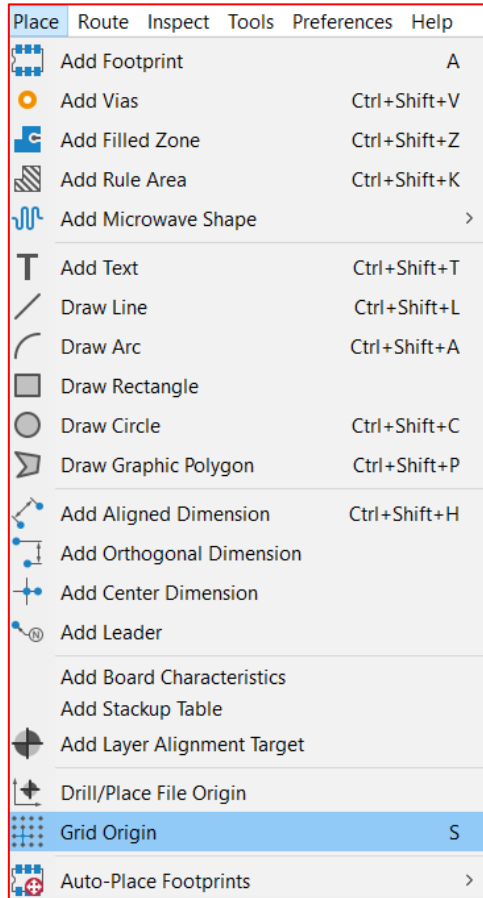
Route	Inspect	Tools	Preferences	Help
	Set Layer Pair...			
	Route Single Track			X
	Route Differential Pair			6
	Tune length of a single track			7
	Tune length of a differential pair			8
	Tune skew of a differential pair			
	Interactive Router Settings...			

Tools	Preferences	Help
	Update PCB from Schematic...	F8
	Switch to Schematic Editor	
	Footprint Editor	
	Update Footprints from Library...	
	Cleanup Tracks & Vias...	
	Remove Unused Pads...	
	Cleanup Graphics...	
	Repair Board	
	Geographical Reannotate...	
	Update Schematic from PCB...	
	Scripting Console	
	External Plugins	>



អក្ខរនៅខាងស្តាំជា Shortcut ទៅកាន់ Tools នោះ








- **Cut** កាត់
- **Copy** ចម្លង
- **Paste** ដាក់
- **Edit Track & Via Properties** កំណត់ Track និង Via
- **Edit Text & Graphics Properties** កែអក្ខរ ឬក្រាហ្វិក
- **Change Footprint** ប្តូរ Footprint
- **Swap Layer** ផ្លាស់ប្តូរ Layer
- **Fill All Zones** បំពេញ Zone ទាំងអស់
- **Unfill All Zones** ដកការបំពេញតារាងទាំងអស់



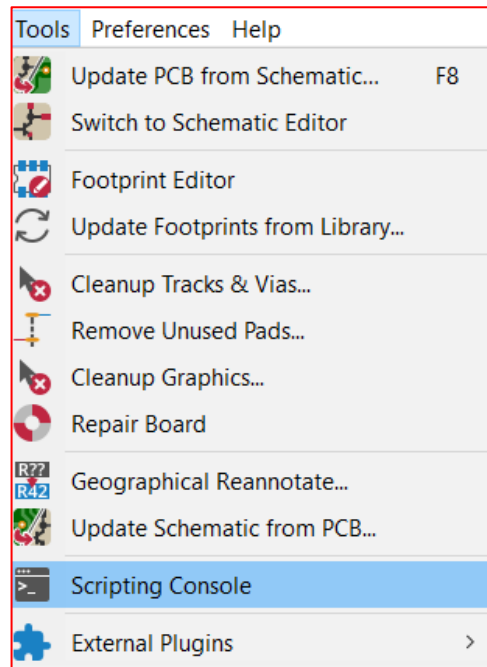
អក្សរនៅខាងស្តាំជា Shortcut ទៅកាន់ Tools នោះ

- **Add Footprint** ទាញយក Footprint ពី Library
- **Add Vias** ដាក់រន្ធចម្លង
- **Add Filled Zone** ដាក់ផ្ទៃទង់ដែង
- **Add Rule Area** ដាក់តំបន់ដែលមានការកំណត់
- **Add Microwave Shape** ប្រើរូបរាងបែប Microwave
- **Text** ដាក់អក្សរ
- **Draw Line** គូសបន្ទាត់
- **Draw Arc** គូសខ្សែកោង
- **Draw Rectangle** គូសចតុកោណកែង
- **Draw Circle** គូសរង្វង់
- **Add Aligned Dimension** ដាក់រង្វាស់ដែលត្រង់
- **Add Leader** ដាក់កំណត់ចំណាំ

អក្សរនៅខាងស្តាំជា Shortcut ទៅកាន់ Tools នោះ

Route	Inspect	Tools	Preferences	Help
	Set Layer Pair...			
	Route Single Track			X
	Route Differential Pair			6
	Tune length of a single track			7
	Tune length of a differential pair			8
	Tune skew of a differential pair			9
	Interactive Router Settings...			Ctrl+Shift+,

- **Set Layer Pair** កំណត់ Layer
- **Route Single Track** រត់ខ្សែមួយសរសៃ
- **Route Differential Pair** រត់ខ្សែគូ
- **Tune length of a single track** ធ្វើការកែសម្រួលប្រវែងខ្សែមួយ
- **Tune length of a differential pair** កែសម្រួលប្រវែងខ្សែគូ
- **Tune Skew of a differential pair** កែសម្រួលប្រវែងគួររបស់ខ្សែគូ
- **Interactive Router Setting** គូកំណត់របៀបនៃការរត់ខ្សែ



អក្សរនៅខាងស្តាំជា Shortcut ទៅកាន់ Tools នោះ

- **Update PCB from Schematic** Update PCB ពី Schematic
- **Switch to Schematic Editor** ប្តូរទៅធ្វើការលើ Schematic
- **Footprint Editor** កែសម្រួល ឬបង្កើត Footprint
- **Update Footprint from Library** ធ្វើបច្ចុប្បន្នភាព Footprint ពី Library
- **Remove Unused Pads** ដកចេញនូវ Pad ដែលមិនប្រើ
- **Geographical Reannotate** ធ្វើការ Annotate ក្នុង PCB
- **External Plugins** ឧបករណ៍ Plugins ផ្សេងៗ

3.1 ការប្រើប្រាស់ក្រដាស (Sheet)

Page Settings

✕

Paper

Size:

A4 210x297mm

Orientation:

Landscape

Custom paper size:

Height: 279.4 mm

Width: 431.8 mm

☐ Export to other sheets

Preview

Drawing Sheet

File:

📁

Title Block

Number of sheets: 1

Sheet number: 1

Issue Date:

2022-07-18

<<<

7/18/2022

⬆

☐ Export to other sheets

Revision:

1.02

☐ Export to other sheets

Title:

Beginner Project

☐ Export to other sheets

Company:

www.etrionicskh.com

☐ Export to other sheets

Comment1:

Prototype #1

☐ Export to other sheets

Comment2:

Deadline June 12 2022

☐ Export to other sheets

Comment3:

☐ Export to other sheets

Comment4:

☐ Export to other sheets

Comment5:

☐ Export to other sheets

Comment6:

☐ Export to other sheets

Comment7:

☐ Export to other sheets

Comment8:

☐ Export to other sheets

Comment9:

☐ Export to other sheets

OK

Cancel

Deadline June 12 2022	
Prototype #1	
www.etrionicskh.com	
Sheet: /	
File: My-first-project.kicad_sch	
Title: Beginner Project	
Size: A4	Date: 2022-07-18
KiCad E.D.A. kicad (6.0.6)	Rev: 1.02
	Id: 1/1

Designe by: Theara Kong Email: theara729@gmail.com Approve by: Mr. Stark www.etrionicskh.com	
Sheet: / File: My-first-project.kicad_sch	
Title: Beginner Project	
Size: A4	Date: 2022-07-18 KiCad E.D.A. kicad (6.0.6)
	Rev: 1.02 Id: 1/1

Page Settings សម្រាប់បង្ហាញព័ត៌មានពីឌីហ្សាញ
របស់គម្រោង រួមទាំងព័ត៌មានរបស់អ្នកឌីហ្សាញ។

Page Settings

Paper

Size: A4 210x297mm

Orientation: Landscape

Custom paper size:

Height: 279.4 mm

Width: 431.8 mm

☐ Export to other sheets

Preview

Drawing Sheet

File:

Title Block

Number of sheets: 1 Sheet number: 1

Issue Date: 2022-07-18 <<< 7/18/2022

Revision: 1.02

Title: Beginner Project

Company: www.etronicskh.com

Comment1: Prototype #1

Comment2: Deadline June 12 2022

Comment3:

Comment4:

Comment5:

Comment6:

Comment7:

Comment8:

Comment9:

☐ Export to other sheets

☐ Export to other sheets

☐ Export to other sheets

☐ Export to other sheets

☐ Export to other sheets

☐ Export to other sheets

☐ Export to other sheets

☐ Export to other sheets

☐ Export to other sheets

☐ Export to other sheets

OK Cancel

Deadline June 12 2022

Prototype #1

www.etronicskh.com

Sheet: /

File: My-first-project.kicad_sch

Title: Beginner Project

Size: A4 Date: 2022-07-18

Rev: 1.02

KiCad E.D.A. kicad (6.0.6)

Id: 1/1

Design by: Theara Kong

Email: theara729@gmail.com

Approve by: Mr. Stark

www.etronicskh.com

Sheet: /

File: My-first-project.kicad_sch

Title: Beginner Project

Size: A4 Date: 2022-07-18

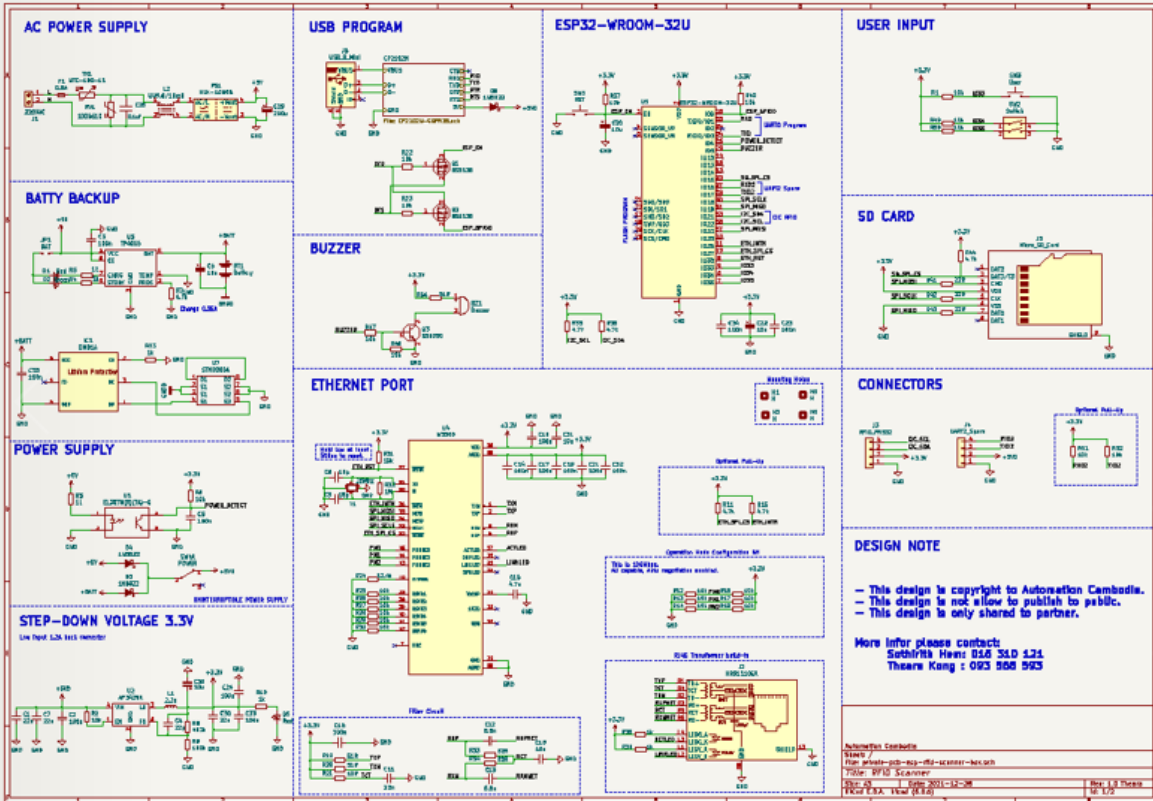
Rev: 1.02

KiCad E.D.A. kicad (6.0.6)

Id: 1/1

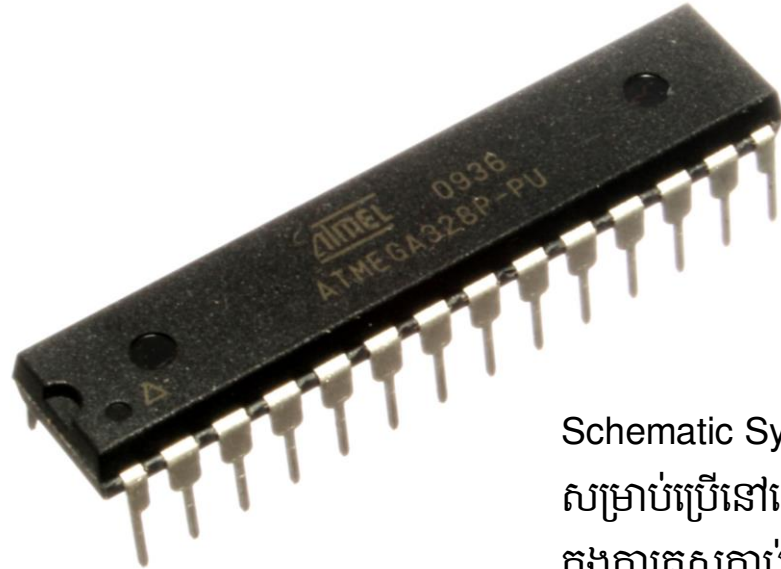
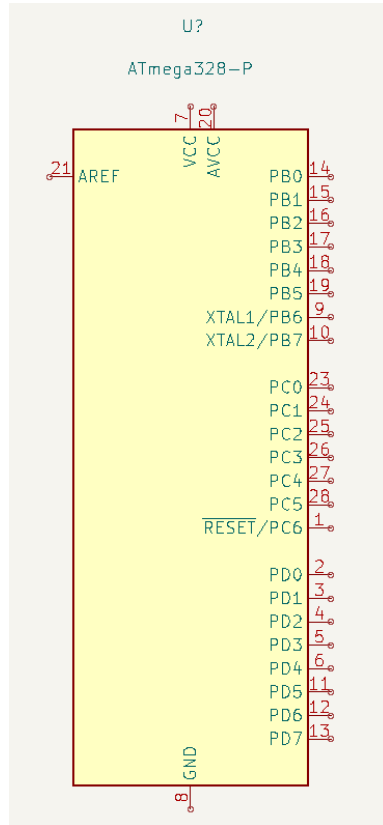
Page Settings សម្រាប់បង្ហាញព័ត៌មានពីឌីហ្សាញ
របស់គម្រោង រួមទាំងព័ត៌មានរបស់អ្នកឌីហ្សាញ។

3.1 ការរៀបចំក្រដាស (Sheet Layout)

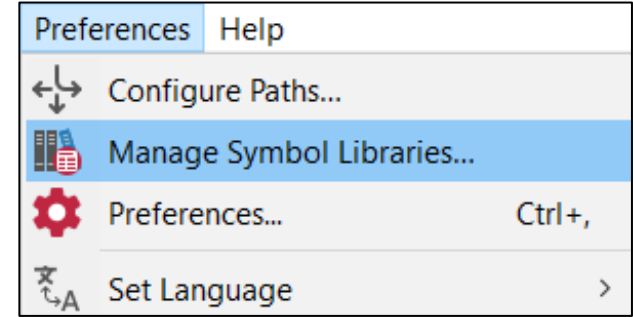
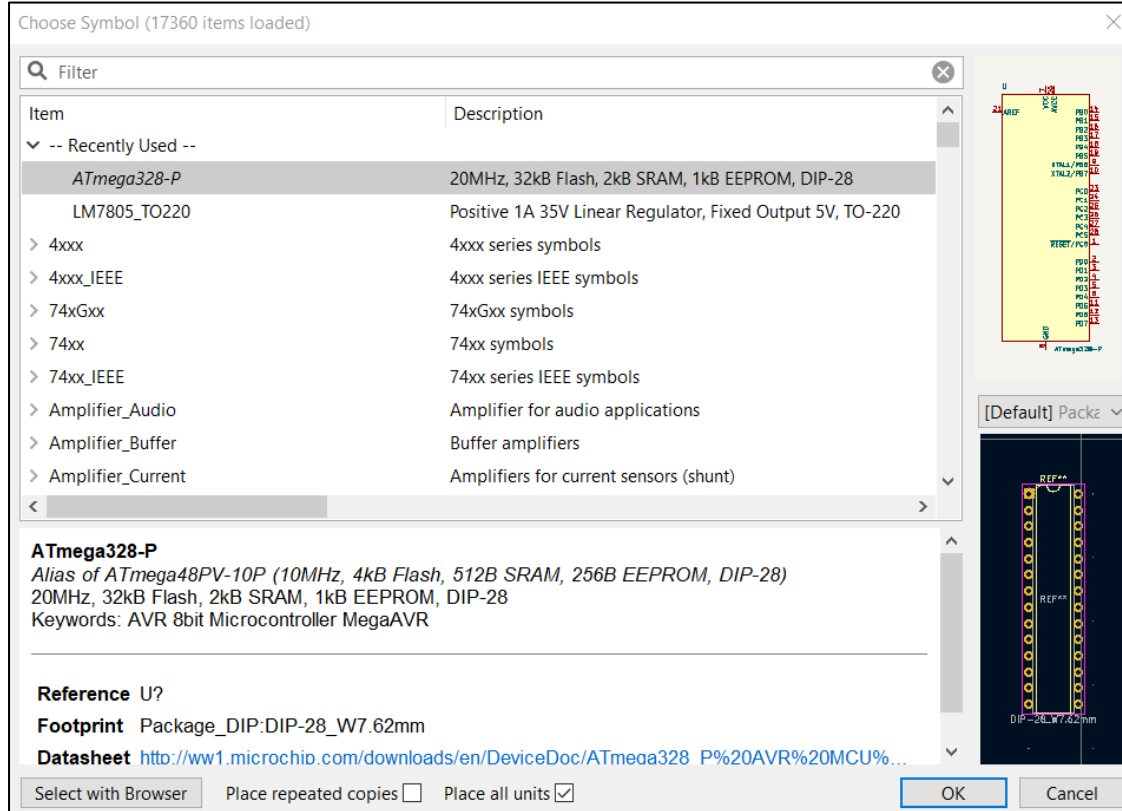


Page Layout ជាការរៀបចំក្រដាសឲ្យ
ងាយស្រួលមើល។ នេះការរៀបចំឲ្យមាន
តំបន់ដូចជា៖

- Input Terminal
- Power Supply
- Voltage regulation
- Main circuit
- Additional circuit
- User Input
- Indicators
- Output Terminal



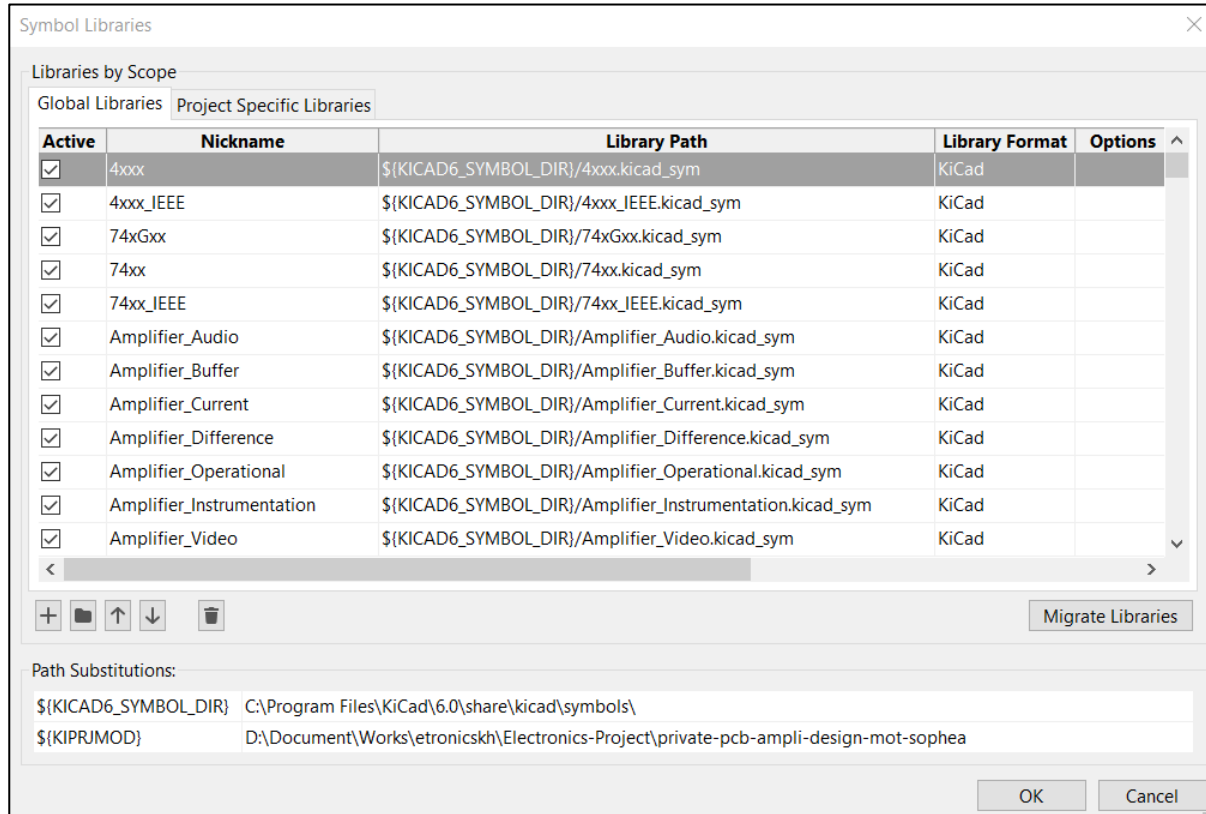
Schematic Symbol ជានិមិត្តសញ្ញា សម្រាប់ប្រើនៅលើសេម៉ា ដើម្បីងាយស្រួល ក្នុងការគូសភ្ជាប់។ ស្ទើរតែទាំងអស់នៃនិមិត្តសញ្ញាមិនដូចទៅនឹងរូបរាងពិតរបស់គ្រឿងនោះទេ។



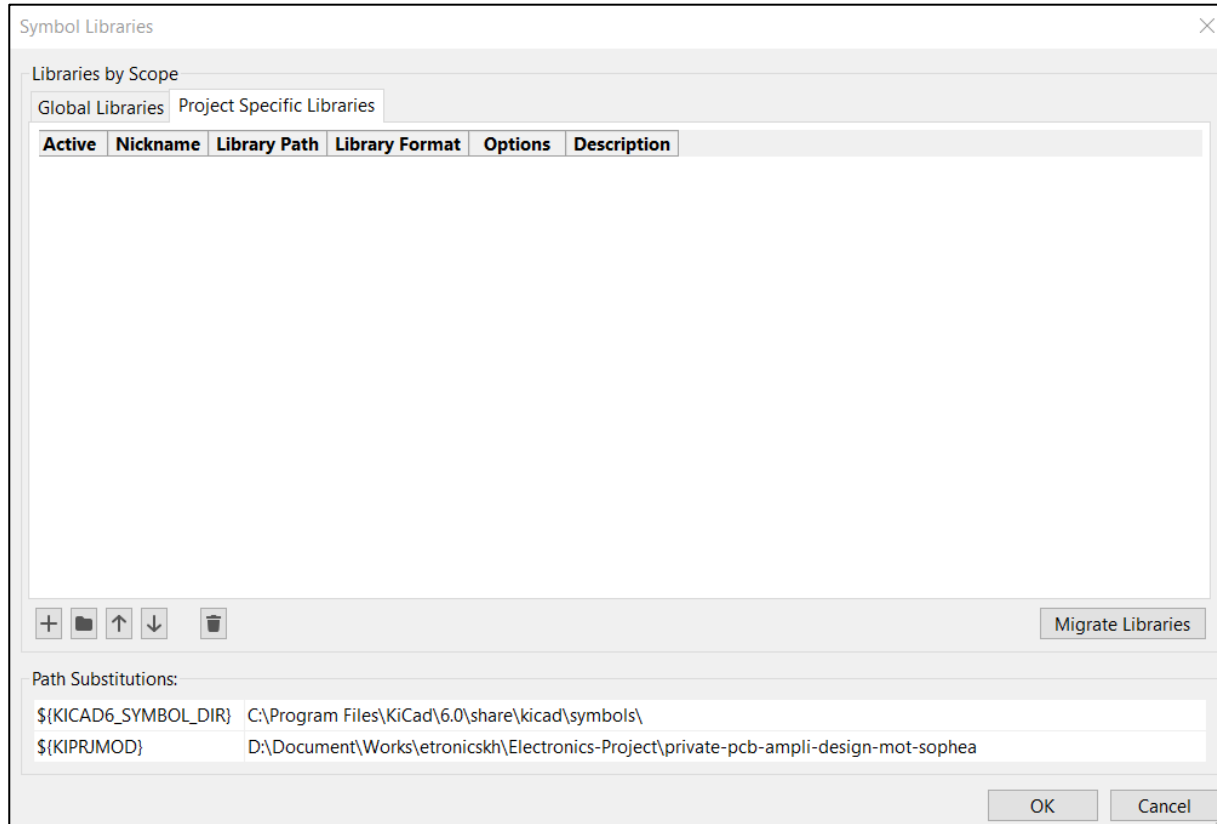
Libraries ជាតារាង Symbol សម្រាប់កម្មវិធីទាញយកទៅប្រើនៅក្នុងឌីហ្សាញ។

Libraries ចែកជា ២ ៖

- Global ប្រើបានគ្រប់គម្រោងនៅលើកុំព្យូទ័រ។
- Project ប្រើបានតែគម្រោងដែលបានដាក់បញ្ចូល។



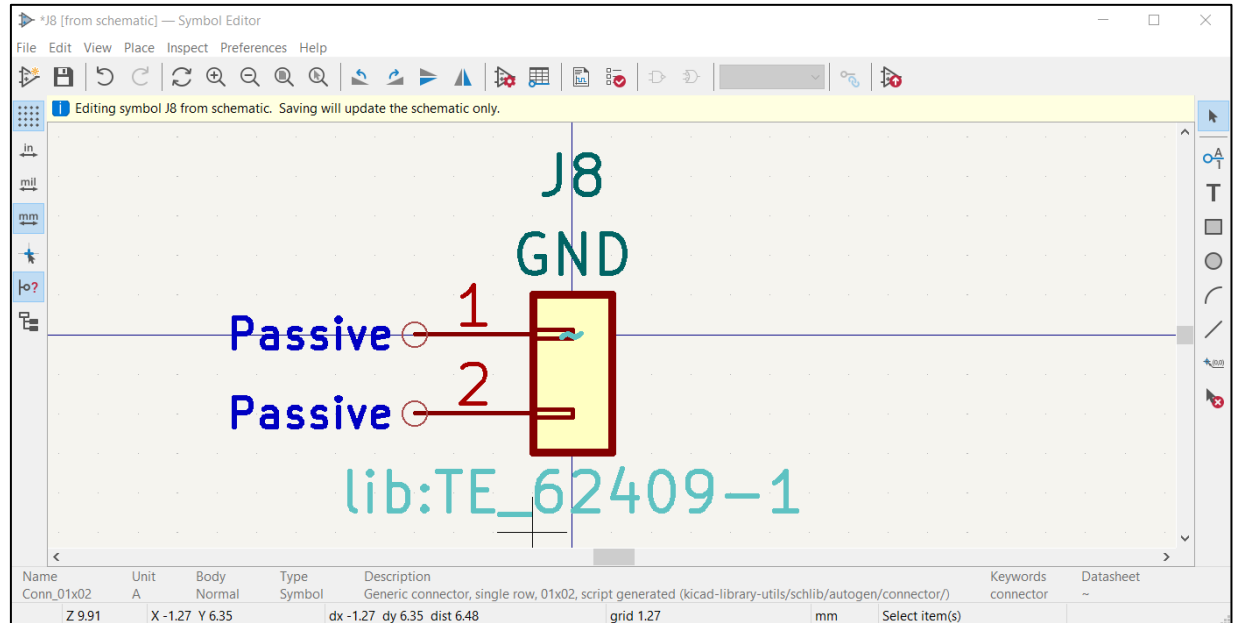
នៅពេលដែល Libraries ត្រូវបាន
បញ្ចូលទៅក្នុង Global Libraries នោះ
វានឹងអាចហៅយកទៅប្រើបានពីគ្រប់
គម្រោងនៅក្នុងកុំព្យូទ័រ។



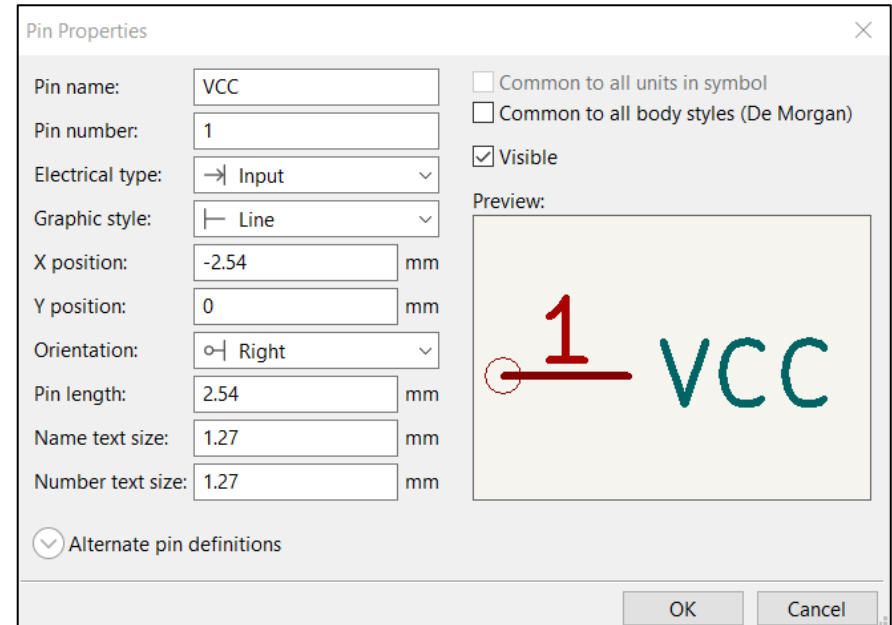
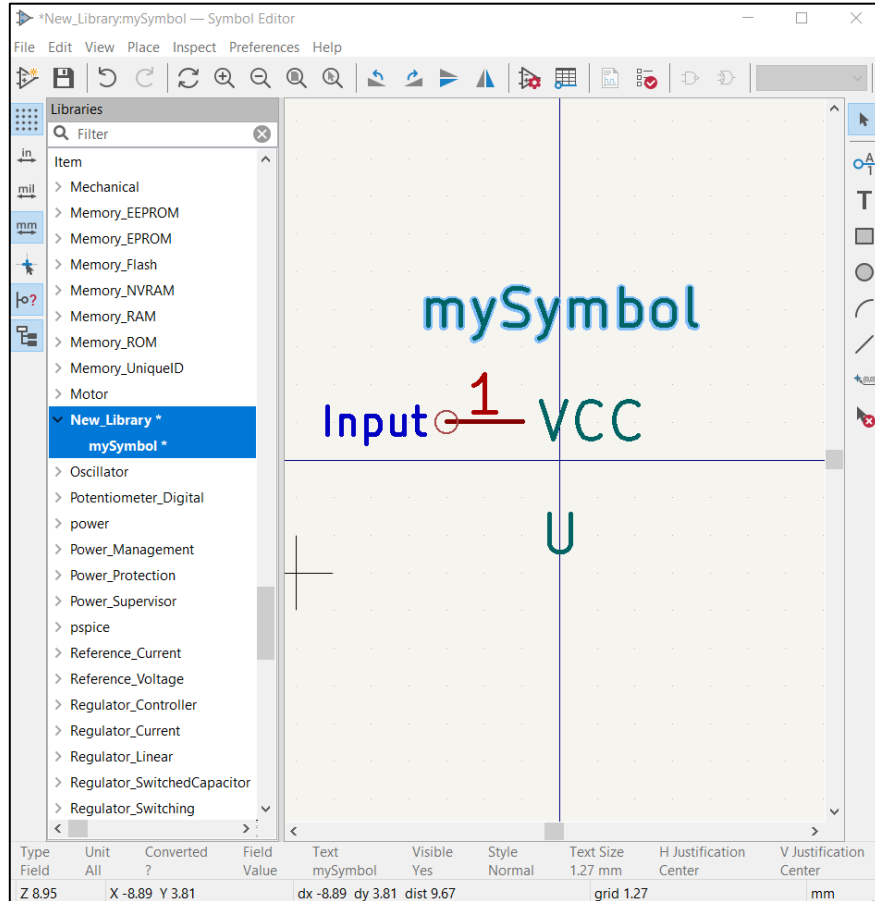
Symbol ដែលបានជា Project Libraries ប្រើបានតែគម្រោងដែលបានដាក់វាចូលជា Libraries ។ ភាគច្រើន Symbol ទាំងនោះត្រូវបានបញ្ចូលទៅក្នុង Directory របស់គម្រោង ដើម្បីងាយស្រួលក្នុងការចែករំលែង។

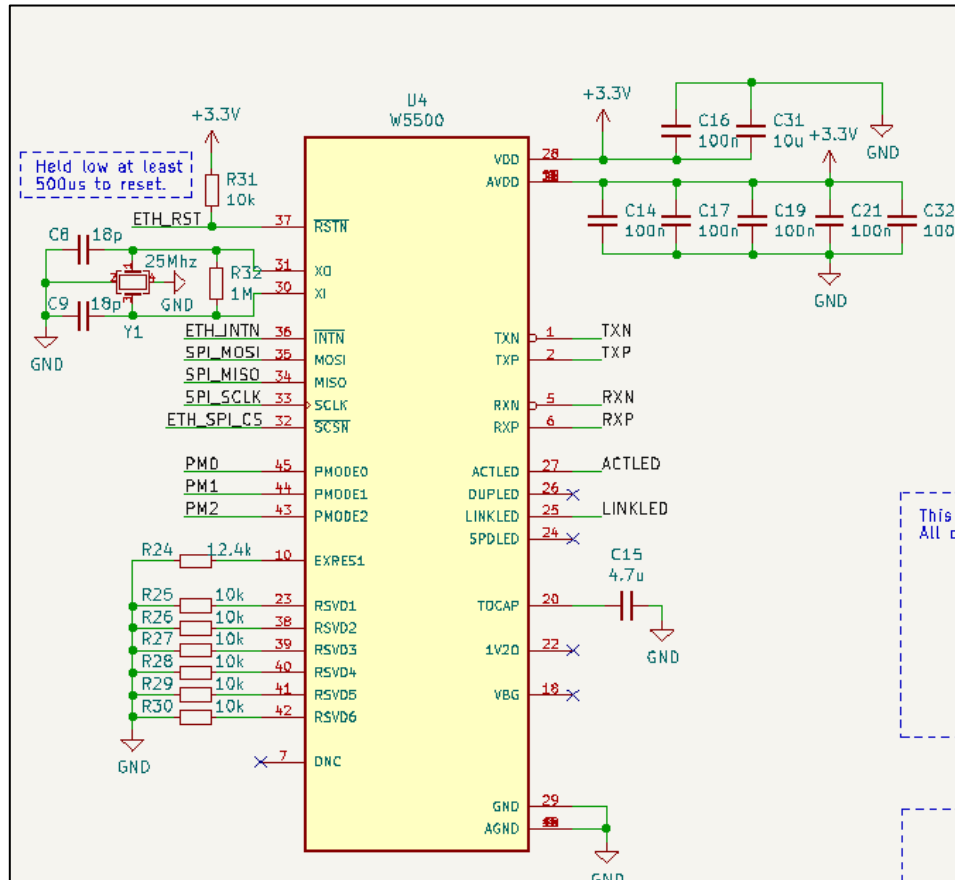
ដើម្បីកែ Symbol ដែលស្ថិតនៅលើ Sheet ស្រាប់ ដោយគ្រាន់តែចុច Right Click រួចជ្រើសរើសយក “Edit With Symbol Editor”

Move	M
Drag	G
Align Elements to Grid	
Rotate Counterclockwise	R
Rotate Clockwise	
Mirror Vertically	Y
Mirror Horizontally	X
Properties...	E
Edit Reference Designator...	U
Edit Value...	V
Edit Footprint...	F
Autoplace Fields	O
De Morgan Conversion	
Edit with Symbol Editor	Ctrl+E
Change Symbol...	
Update Symbol...	
Show Datasheet	D
Cut	Ctrl+X
Copy	Ctrl+C
Paste	Ctrl+V



ធ្វើការបង្កើត Library ថ្មី ដោយដាក់ឈ្មោះ និងអក្សរតាង
និមិត្តសញ្ញា រួចធ្វើការគូស Symbol ដោយប្រើប្រាស់
ឧបករណ៍នៅផ្នែកខាងស្តាំនៃផ្ទាំងបង្កើត Library ។



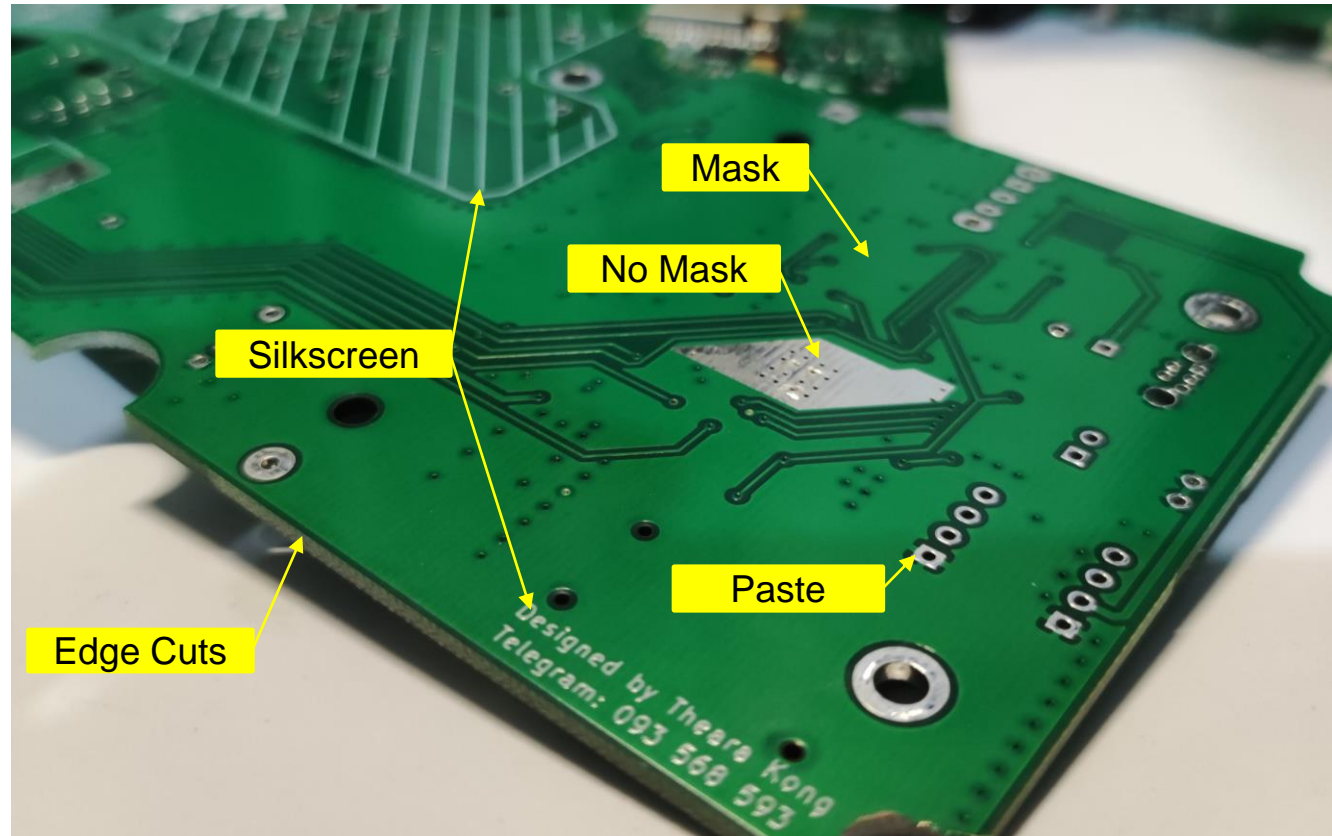
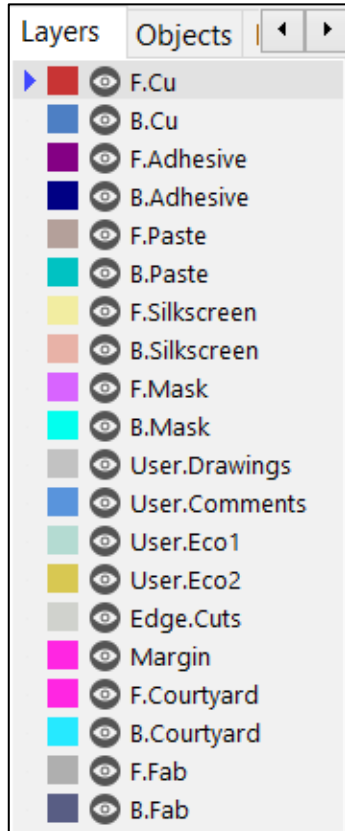


ដើម្បីបានដ្យាក្រាមល្អ ដ្យាក្រាមត្រូវមានដូចខាងក្រោម៖

- តំបន់ផ្សេងៗគ្នា
- អាចមើលគ្រប់អក្សរបានច្បាស់
- មិនមានខ្សែកាត់គ្នាច្រើន
- មានអក្សរកត់ចំណាំ
- ចែកដាច់នូវ Power Symbol
- បញ្ជាក់ពីជើងដែលមិនប្រើ
- ប្រើប្រាស់ Symbol ដូចៗគ្នា

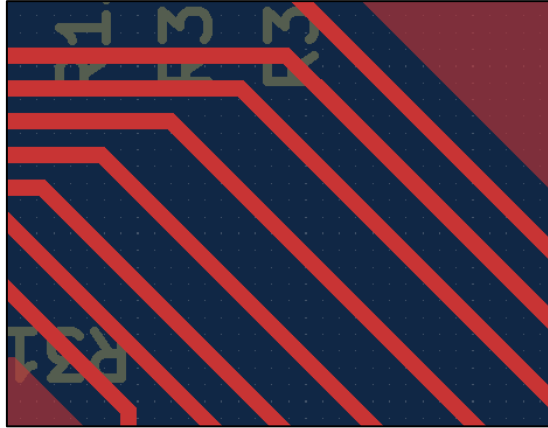
4. PCB Design

4.1 PCB Layers



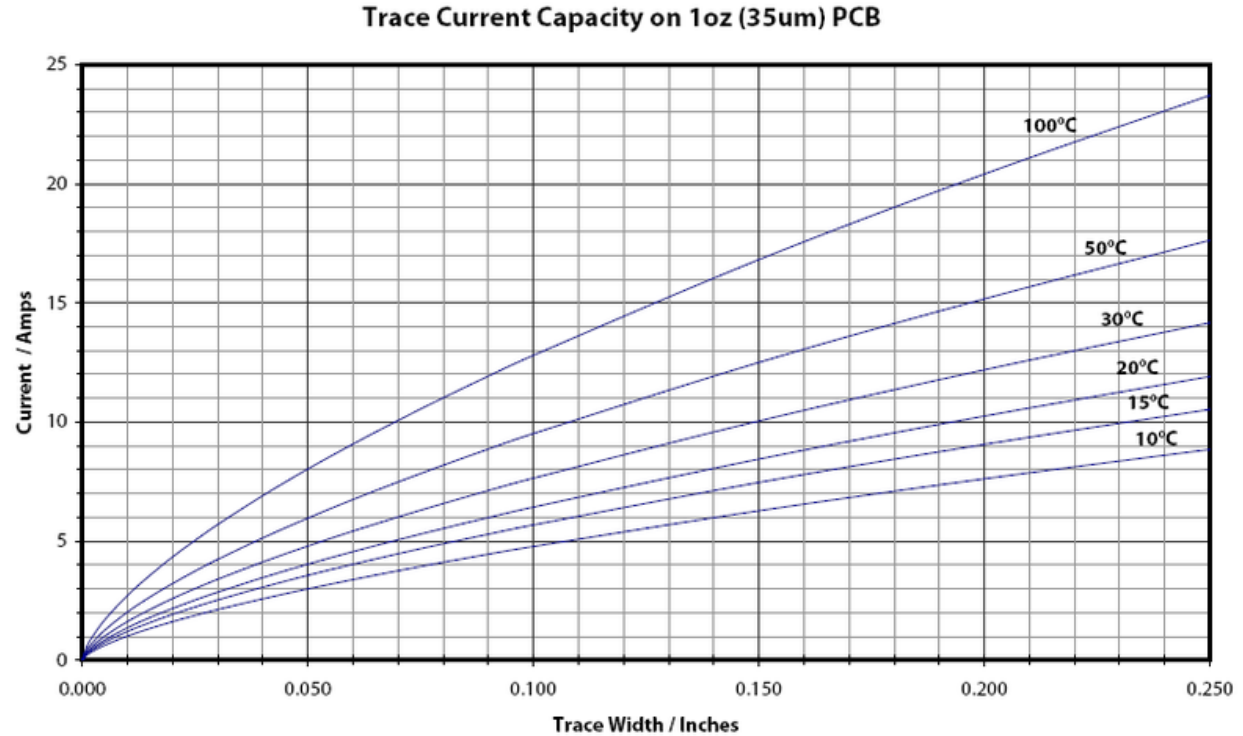
4. PCB Design

4.2 Trace (and Current Rate)

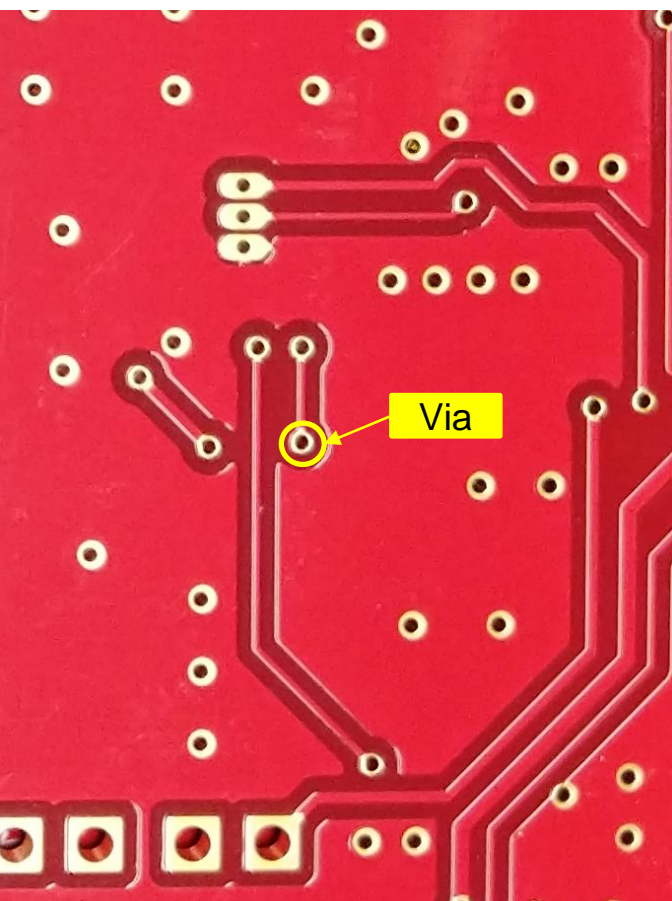


IPC Recommended Track Width For 1 oz cooper PCB and 10 °C Temperature Rise

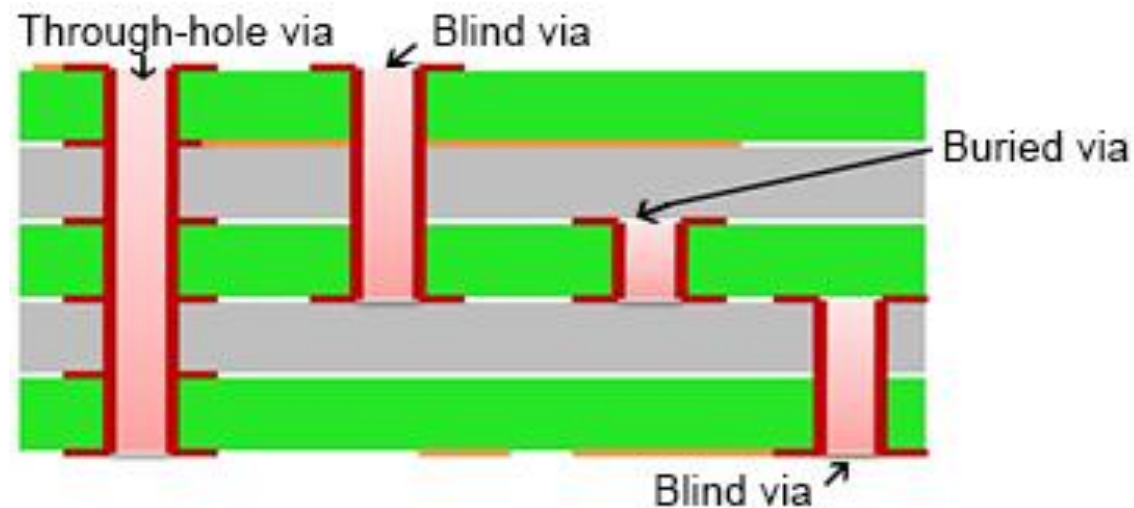
Current/A	Track Width(mil)	Track Width(mm)
1	10	0.25
2	30	0.76
3	50	1.27
4	80	2.03
5	110	2.79
6	150	3.81
7	180	4.57
8	220	5.59
9	260	6.60
10	300	7.62



[Online Calculator](#)



Via ប្រើដើម្បីចម្លងចរន្តពី Layer មួយទៅ Layer មួយទៀត។
Via មានច្រើនប្រភេទ និងអាចធ្វើបានទៅតាមការកំណត់របស់រោងចក្រ។

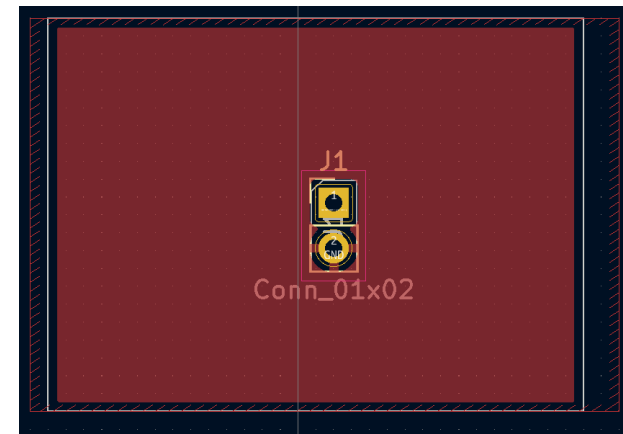
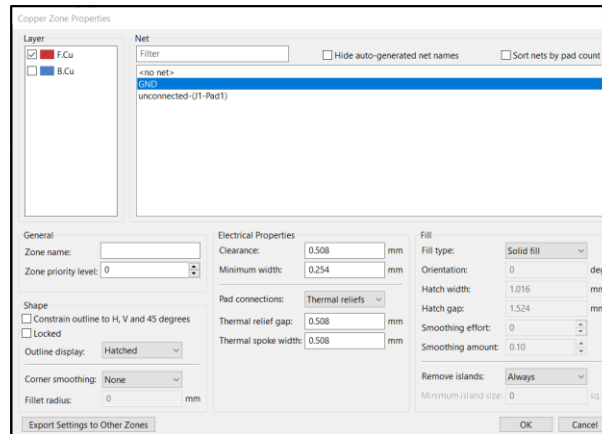
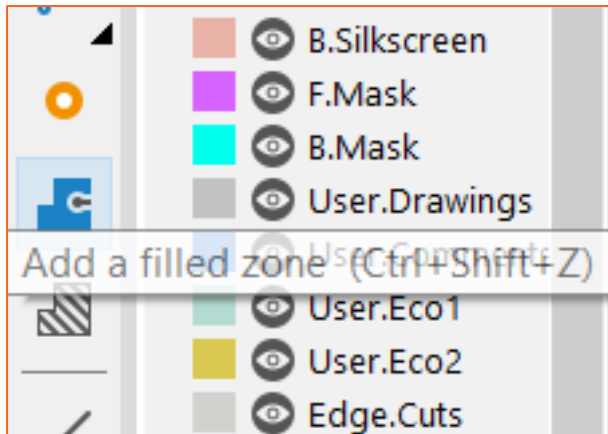
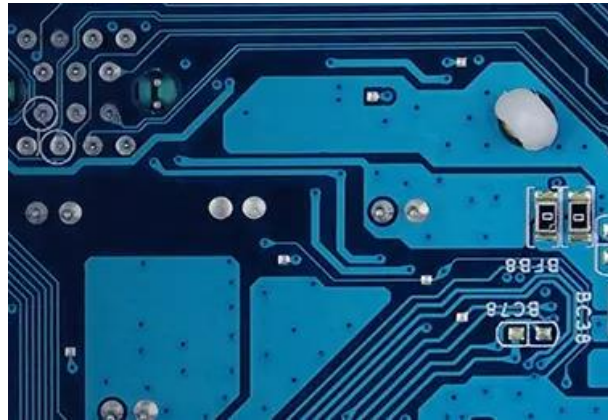


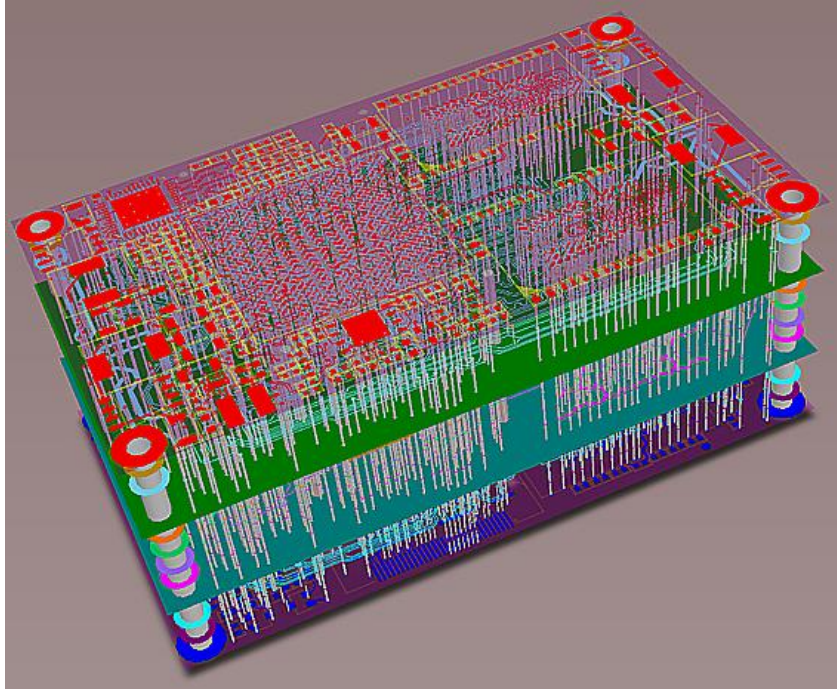
4. PCB Design

4.4 Zone ឬ Plane

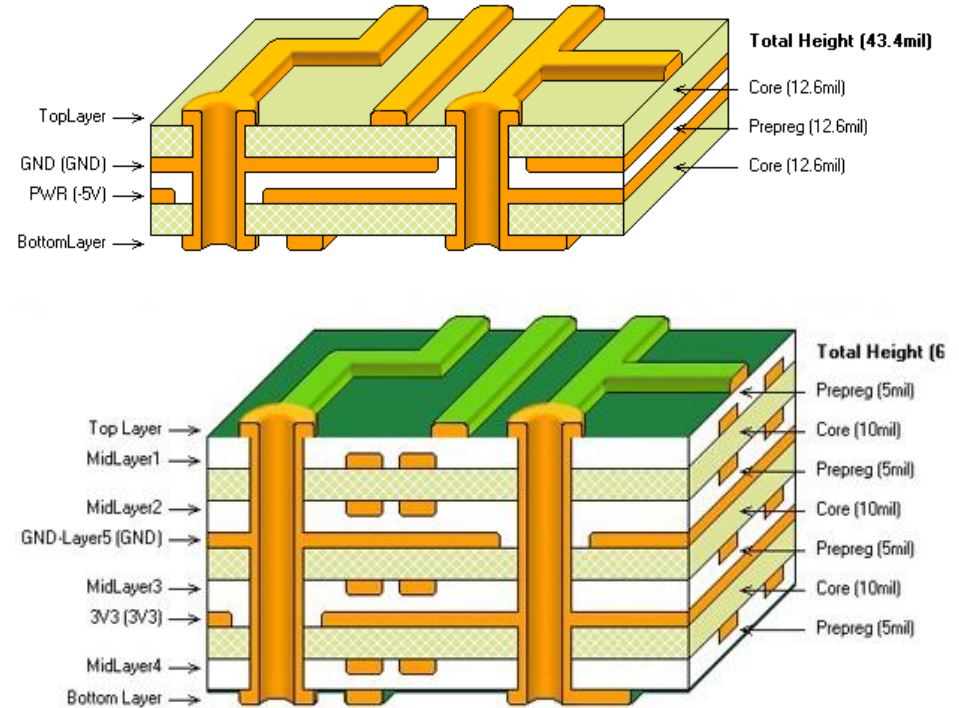
Zone ឬ Plane ប្រើសម្រាប់ជាផ្ទៃចែកចាយ Ground ឬ Power ដែលអាចចែកចាយចរន្តបានធំ។

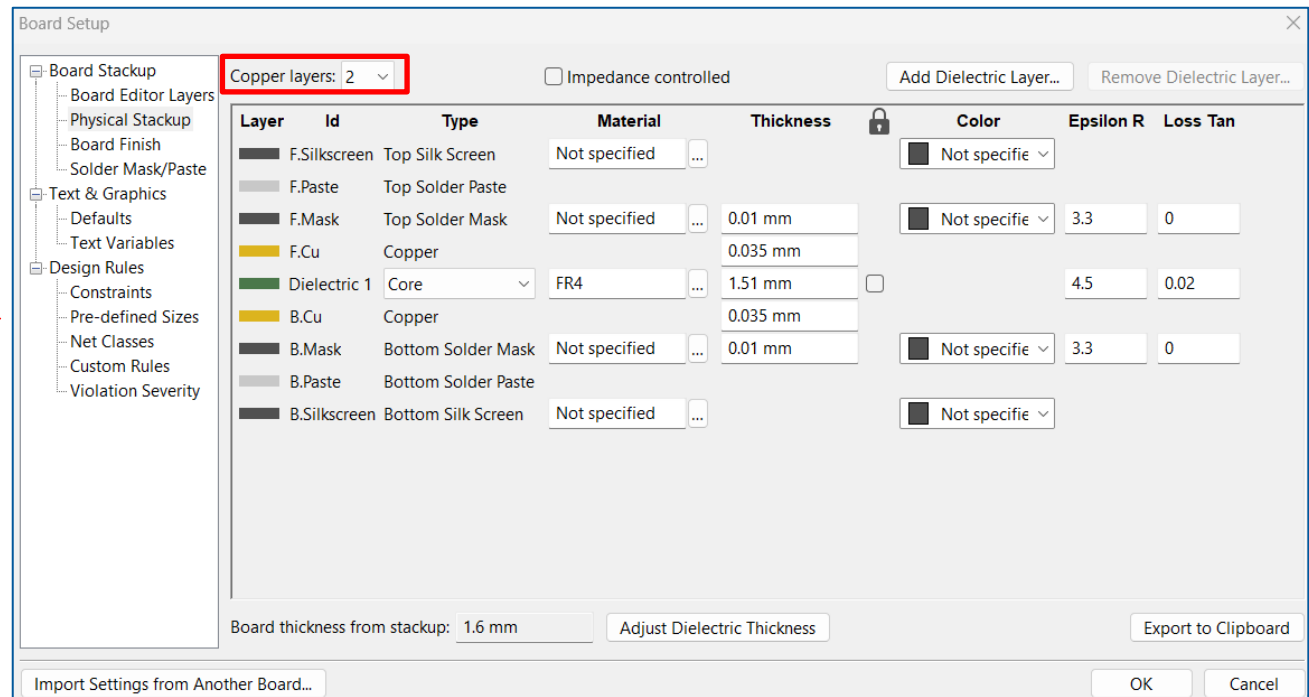
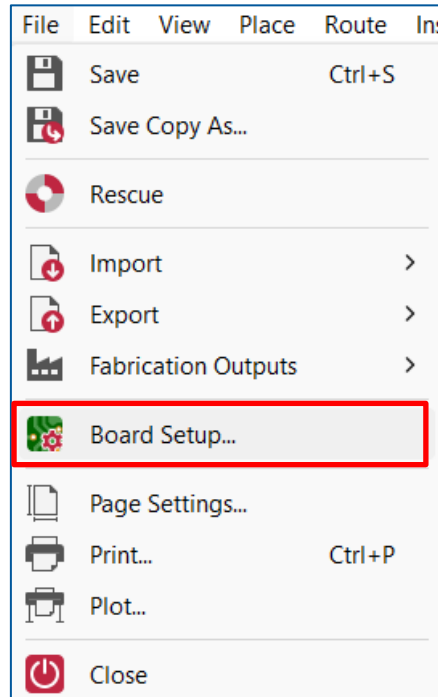
PCB Design មួយអាចមាន Zone ច្រើនទៅតាម Design ជាក់ស្តែង។



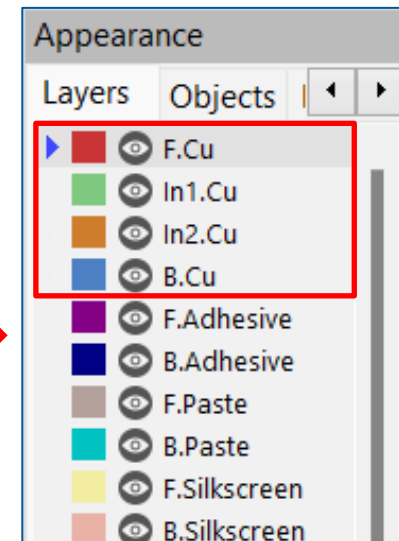
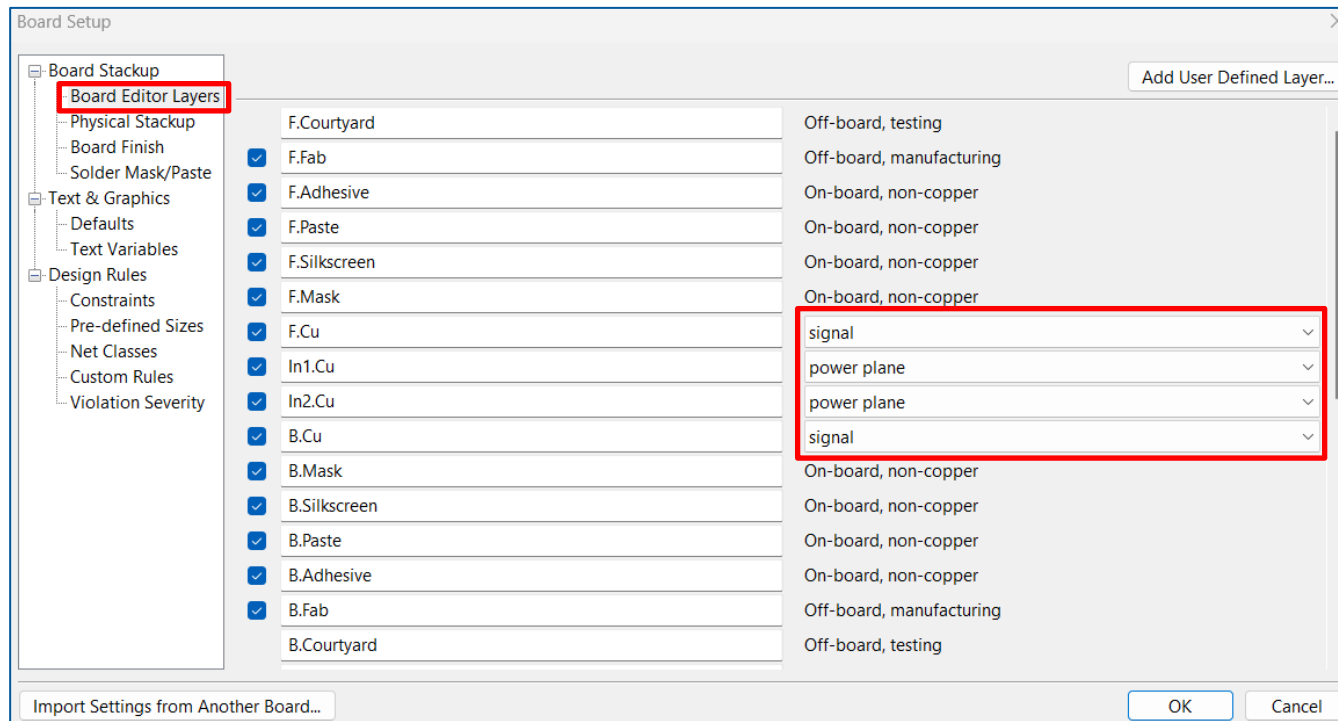


PCB បង្ហាញពីការភ្ជាប់គ្នារវាង PCB ដែលមានច្រើនស្រទាប់





ចូលទៅកាន់ File >> Board Setup >> Board Stackup >> Physical Stackup រួចធ្វើការកំណត់ចំនួនស្រទាប់នៅលើ Copper Layers

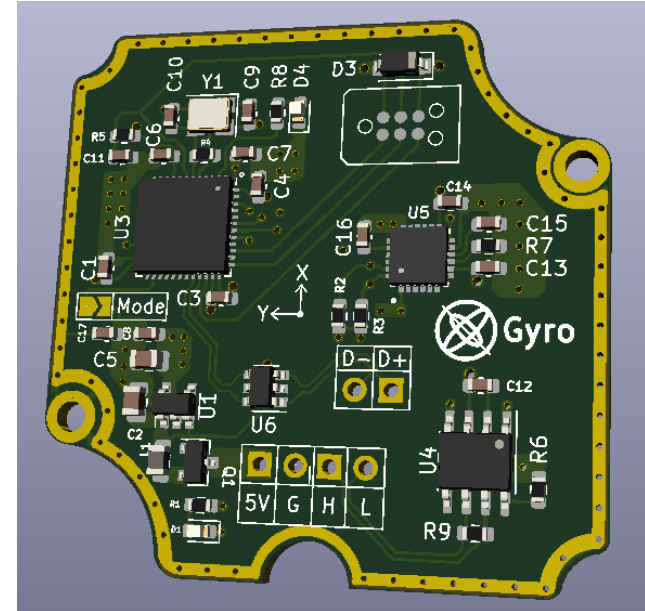


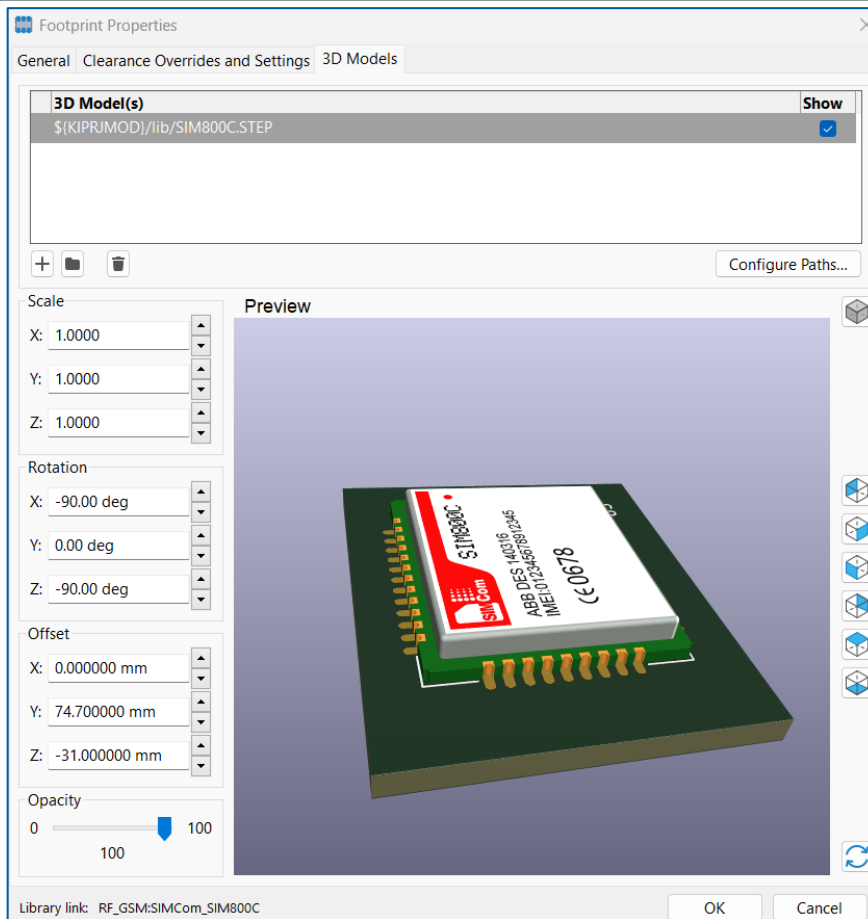
បន្ទាប់មកប្តូរទៅកាន់ Board Editor Layers រួចធ្វើការកំណត់ប្រភេទស្រទាប់ (Define Layer)

រូបរាងរបស់ PCB ត្រូវបានកំណត់ទៅតាមប្រអប់ដែលត្រូវដាក់ ទំហំដែលបានកំណត់។



- User.Drawings
- User.Comments
- User.Eco1
- User.Eco2
- Edge.Cuts
- Margin

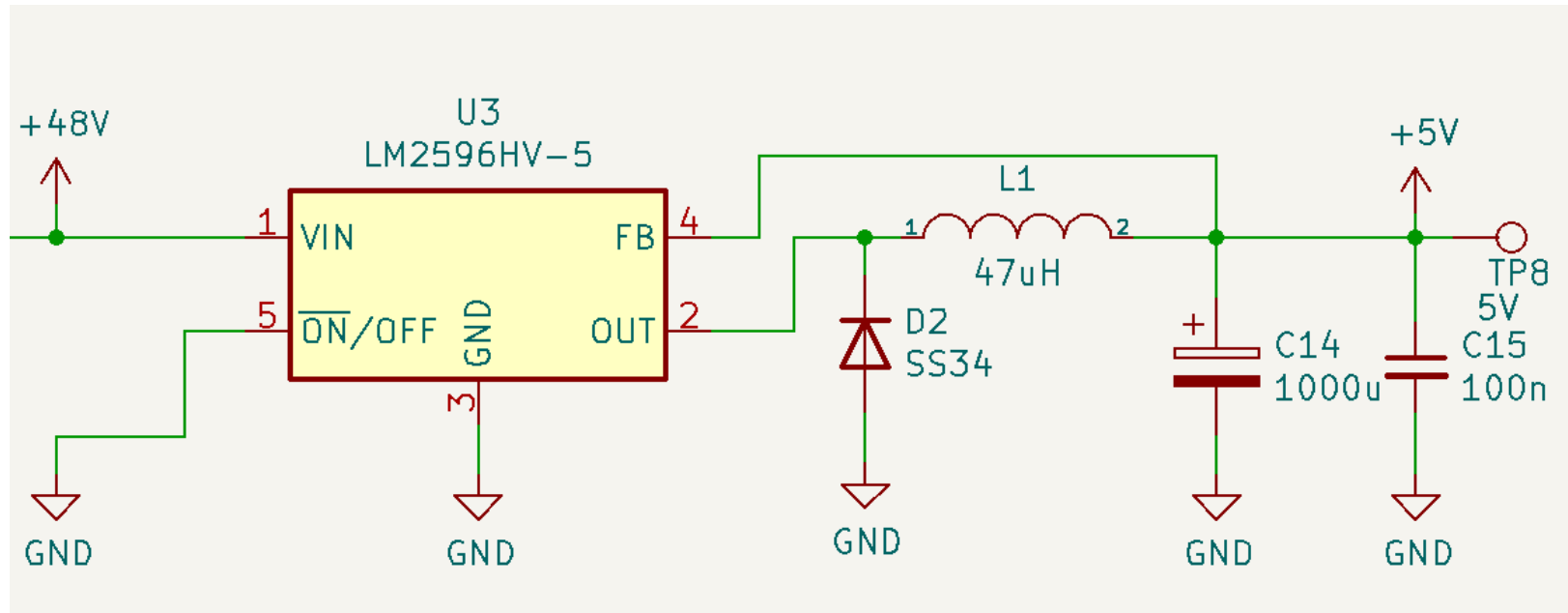


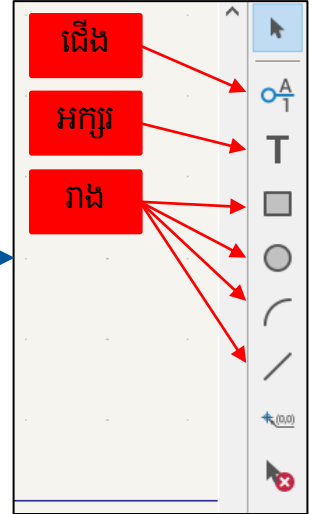
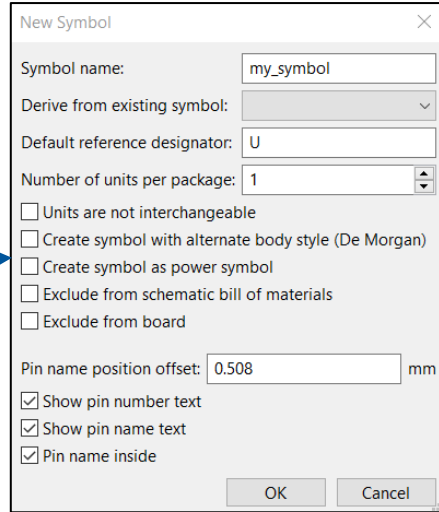
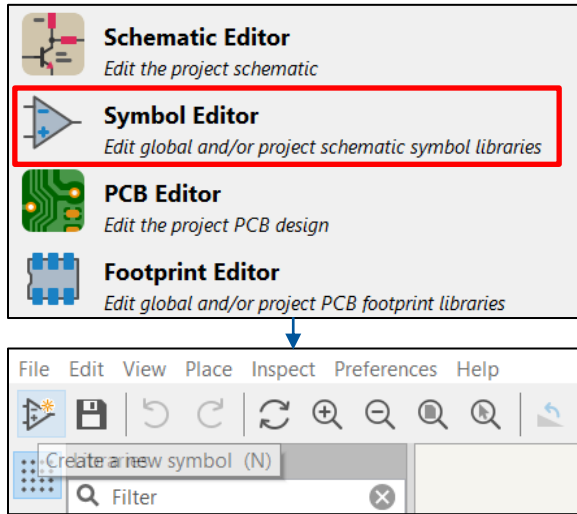


ដើម្បីងាយស្រួល ប្រើប្រាស់ File STEP
ដើម្បីបញ្ចូលទៅកាន់ 3D



Practice + Design Review

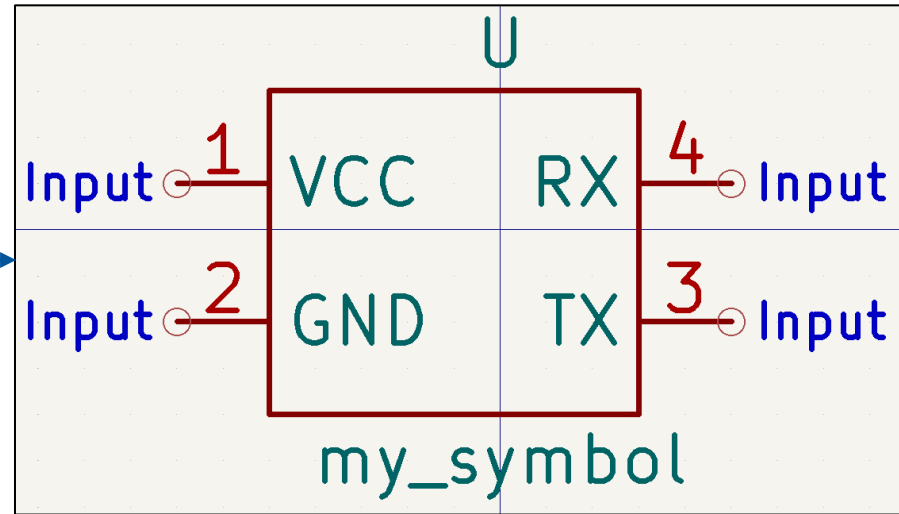
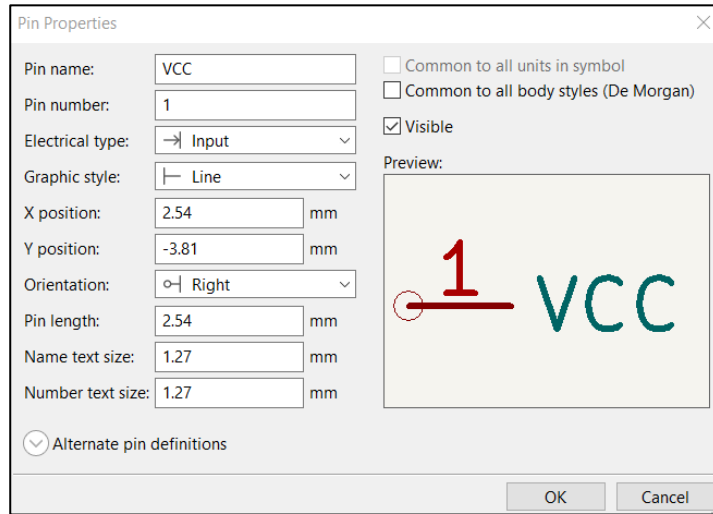




1. នៅផ្ទាំង Home ជ្រើសរើសយក Symbol Editor
2. ជ្រើសរើស Group ណាមួយ និង Create new symbol (N)

3. ដាក់ឈ្មោះ Symbol រួចចុច OK

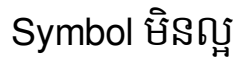
4. ចុចលើឧបករណ៍នៅខាងស្តាំដើម្បីបង្កើតជារូបរាង និងជើងរបស់ Symbol



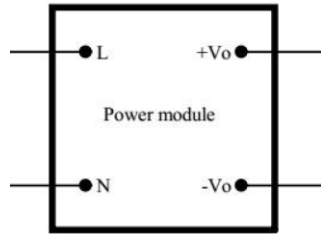
5. ដាក់ឈ្មោះជើង និងលេខរបស់
ជើង

6. បង្កើតរូបរាងរបស់ Symbol

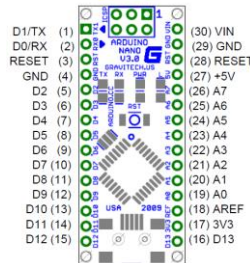
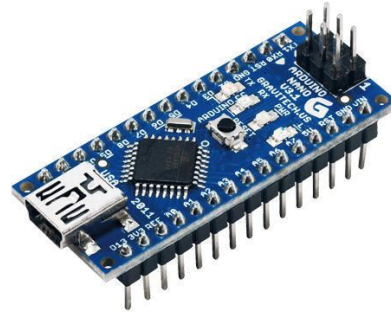
7. Export Symbol ដើម្បីទុកប្រើប្រាស់



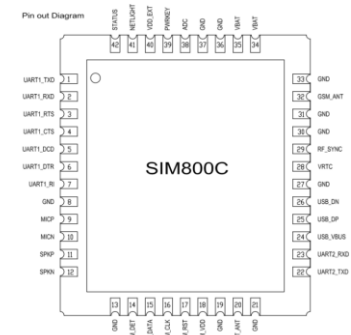
Practice + Review



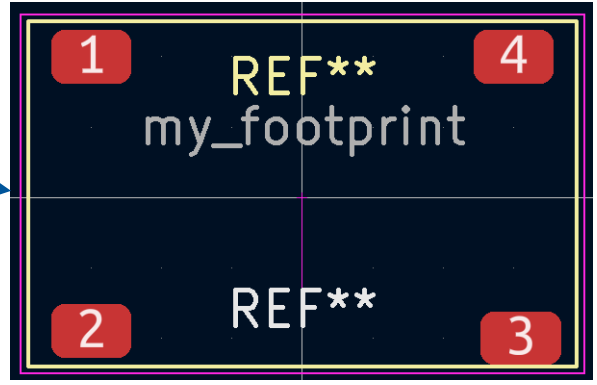
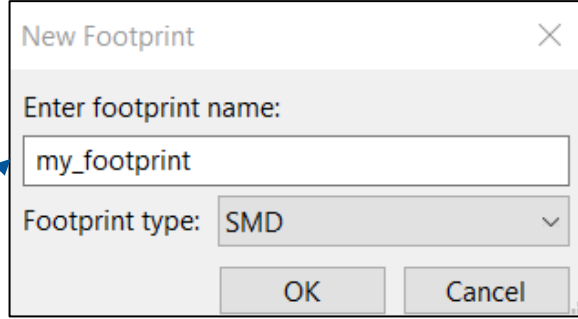
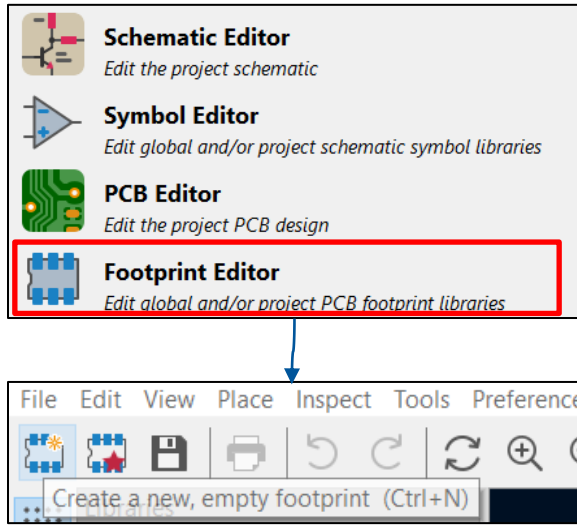
[Hi-Link](#)



[Arduino V3.0](#)



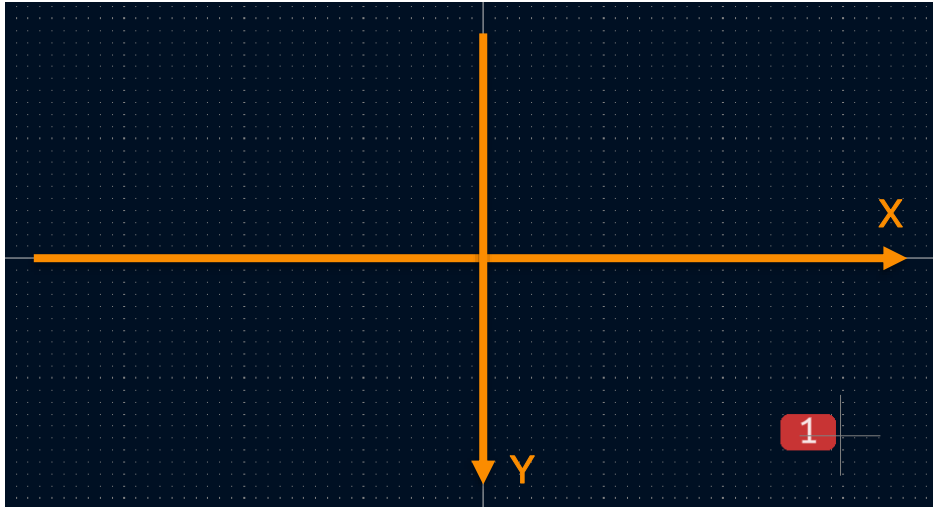
[SIM800C](#)



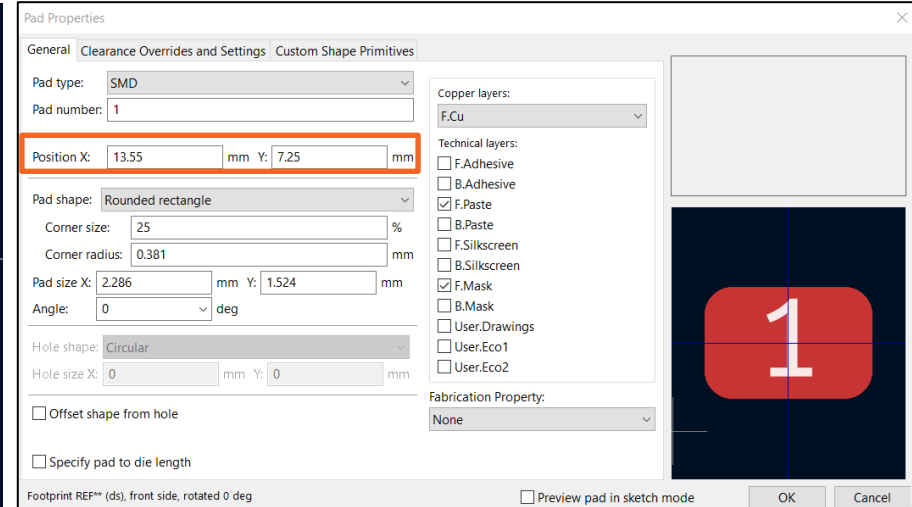
1. នៅទំព័រដើមជ្រើសរើស
Footprint Editor
2. ជ្រើសរើស Group រួចចុច
Create a new footprint

3. ដាក់ឈ្មោះ Footprint និងជ្រើស
រើស Footprint Type

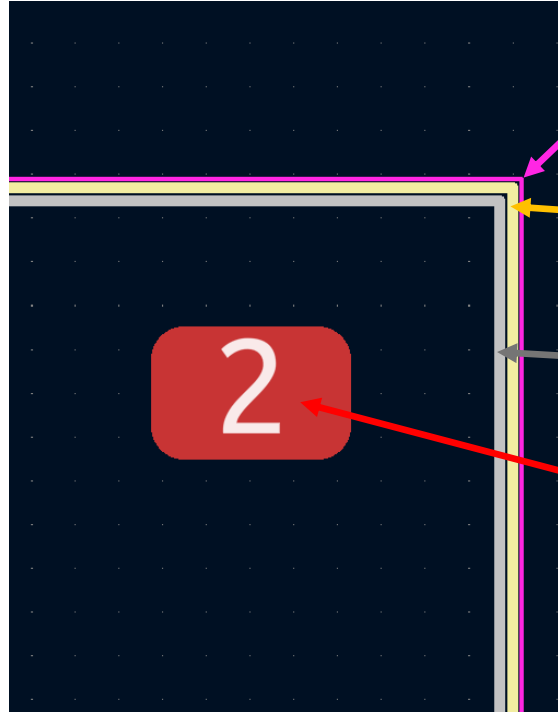
4. បង្កើតរូបរាង Footprint រួច
Export ទុកប្រើប្រាស់



កូអរដោនេ សម្រាប់គូរ



Properties របស់ជើង



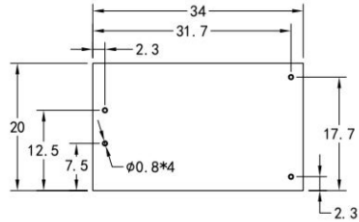
Courtyard : បញ្ជាក់ពីដែនកំណត់នៃទំហំរបស់គ្រឿង (ការពារប្រជ្រៀតគ្នា)

Silkscreen : បង្ហាញនៅលើ PCB ដើម្បីជាគំនូសចំណាំ ឬរូបរាង

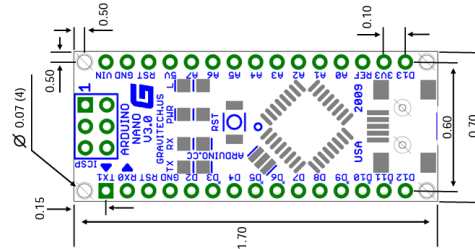
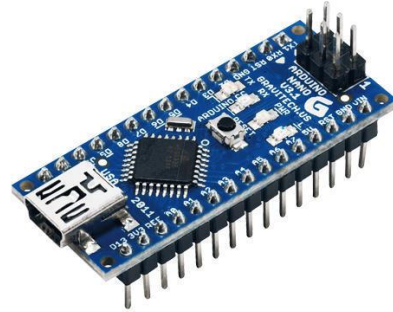
Drawing : បញ្ជាក់ពីទំហំពិតរបស់គ្រឿង (មិនរួមទាំងជើងនោះទេ)

Pad : ផ្ទៃសម្រាប់ផ្សារជើងរបស់គ្រឿង

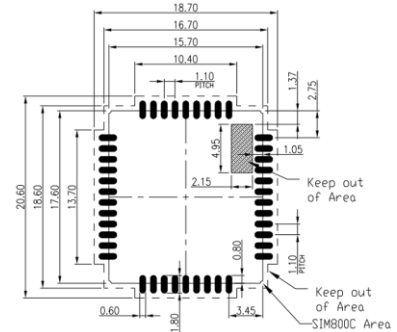
Practice + Review



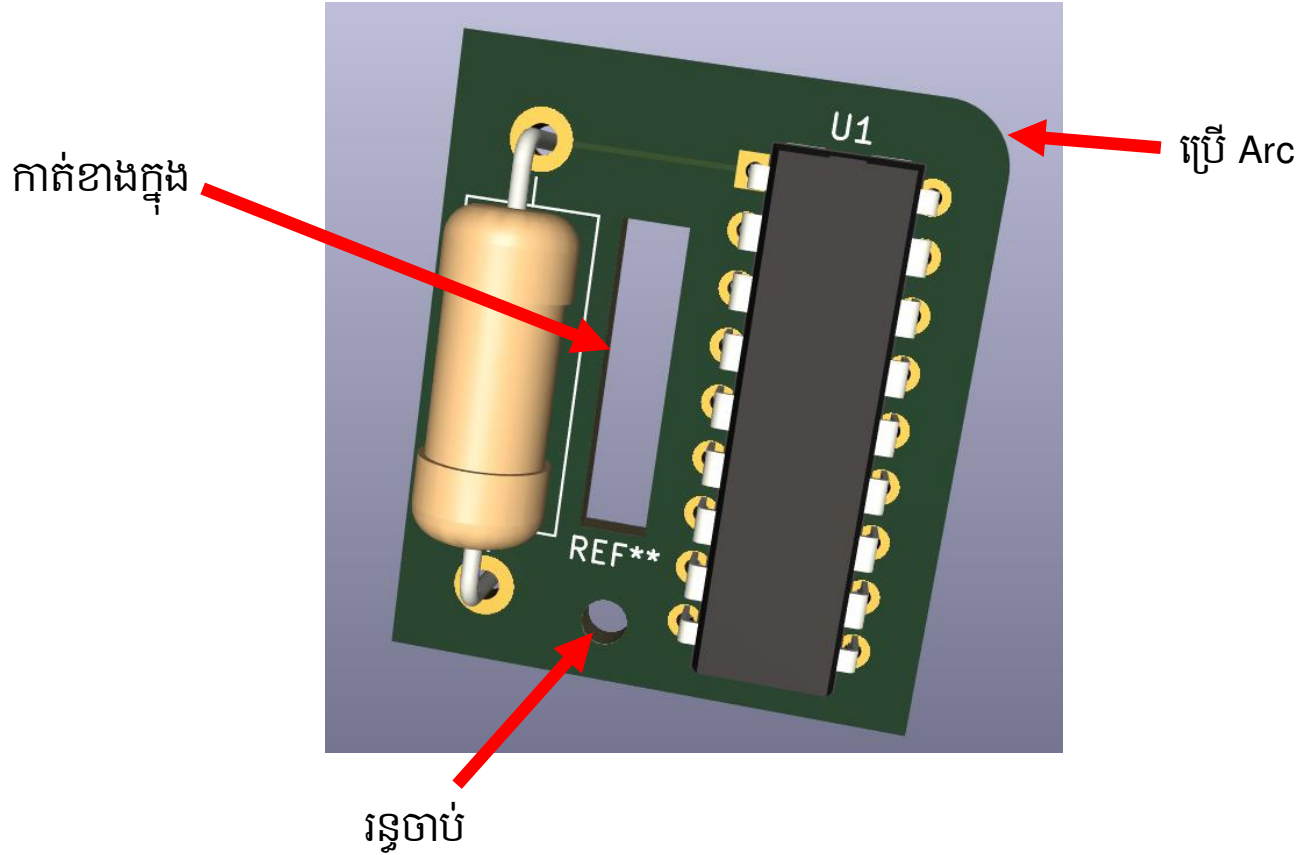
[Hi-Link](#)



[Arduino V3.0](#)



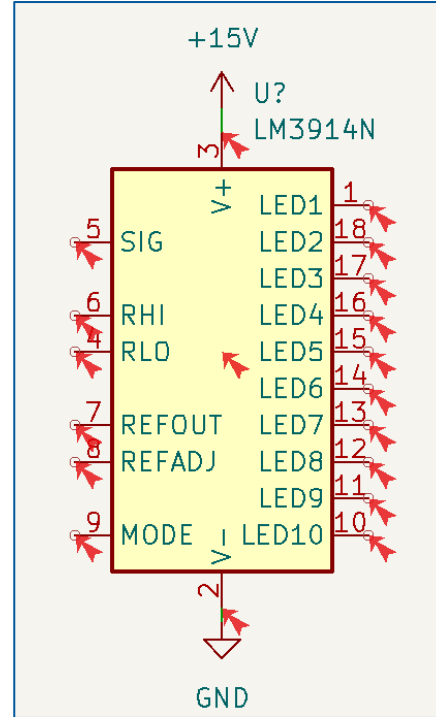
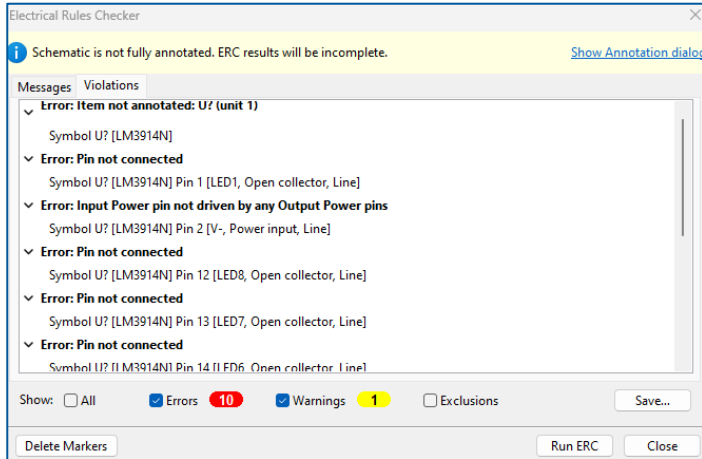
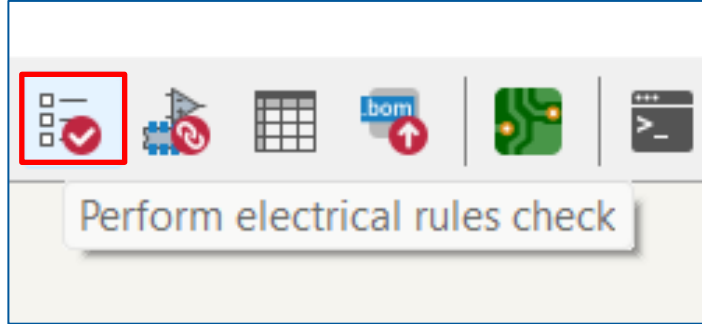
[SIM800C](#)



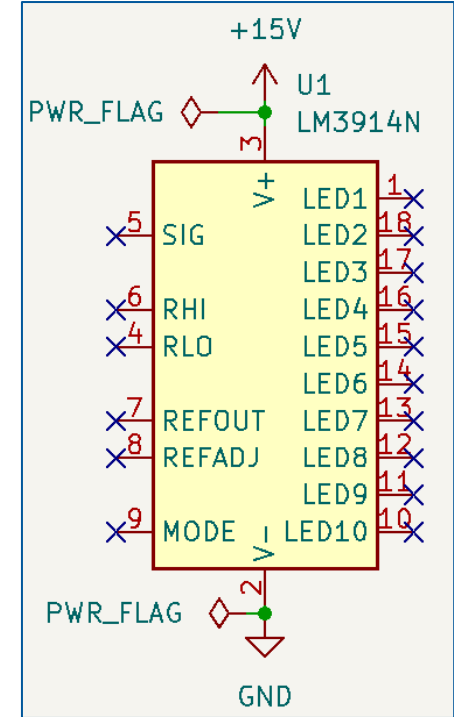
7. Design Rule

7.1 Schematic

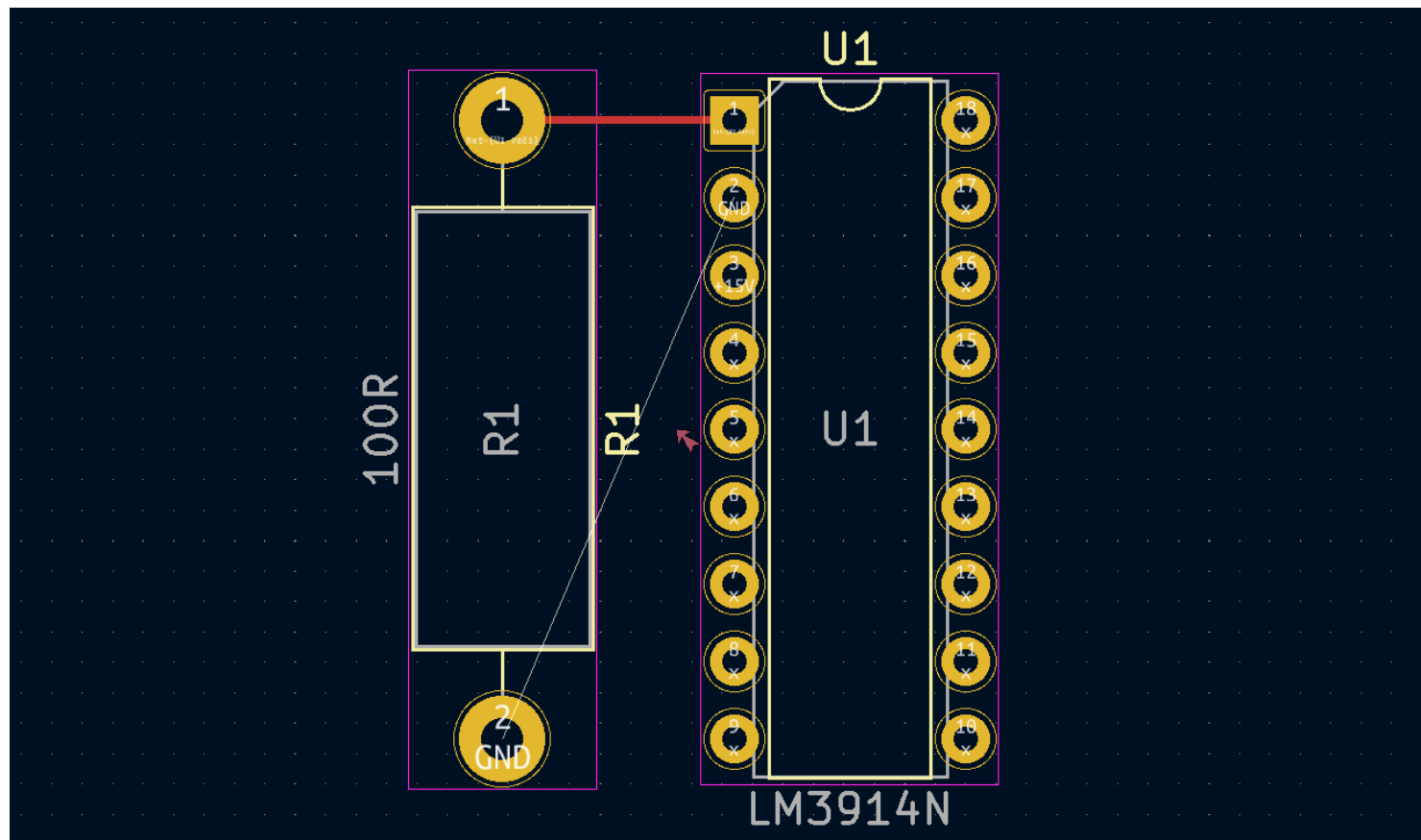
Tool សម្រាប់ពិនិត្យមើលភាពត្រឹមត្រូវ

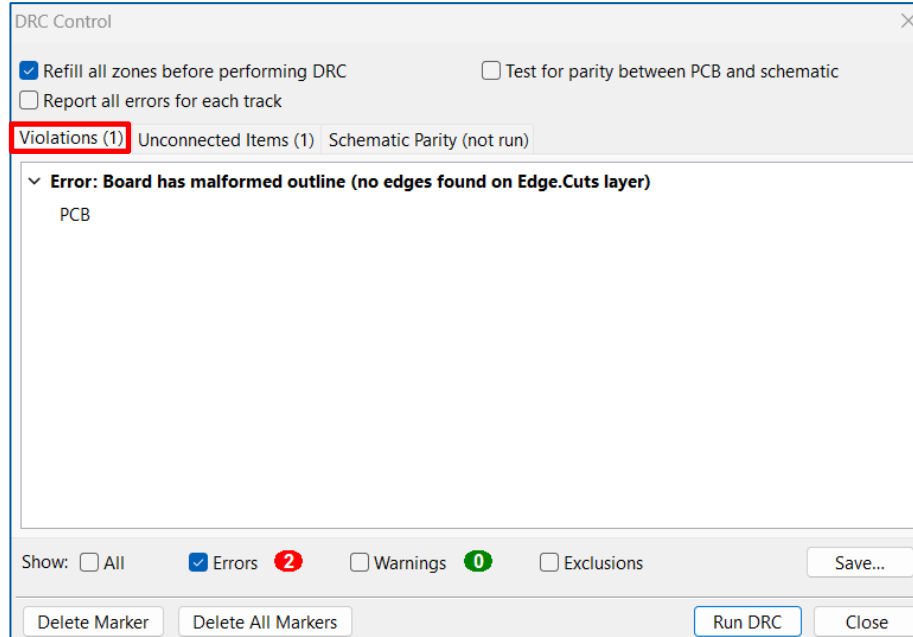


ការឌីហ្សាញមិនល្អ

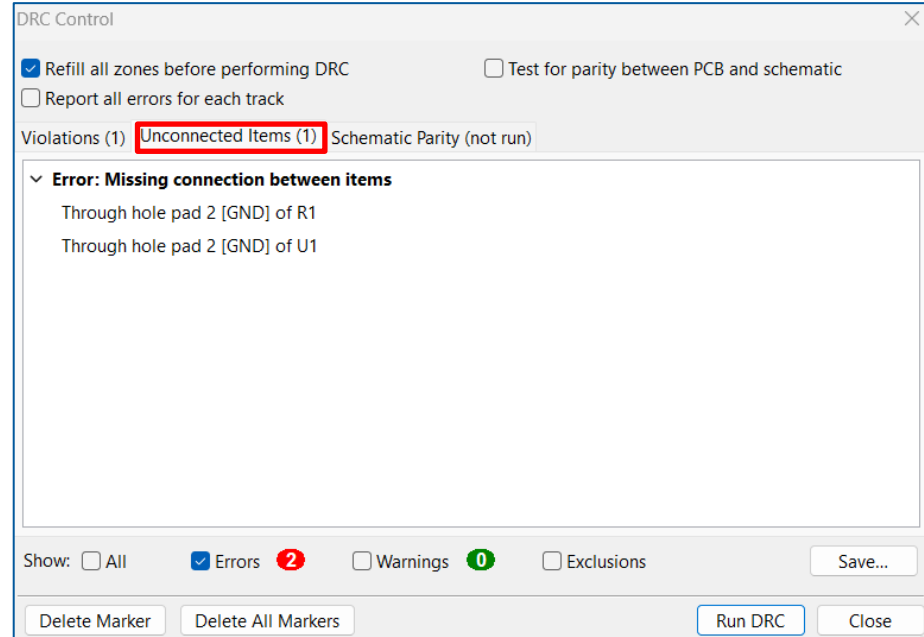


ការកំណត់ត្រឹមត្រូវ





Violations បញ្ជាក់ពី Error ដែលខុស ឬខ្វះ



Unconnected Items បង្ហាញពីខ្សែដែលមិនទាន់ភ្ជាប់