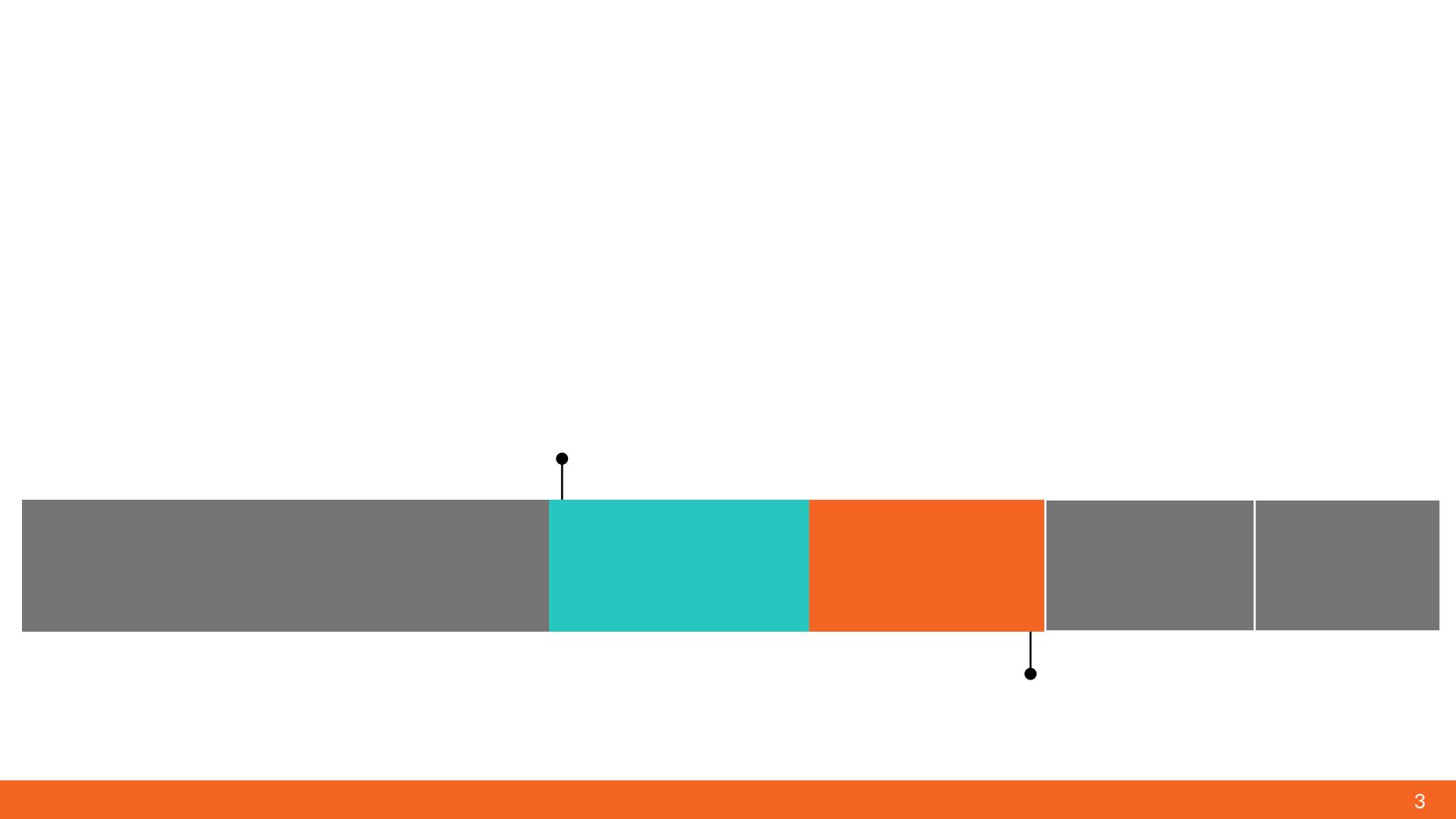


Online Course

1

មូលដ្ឋានឌីហ្មាល់បច្ចុបត ជាមួយ KiCAD **Fundamental**





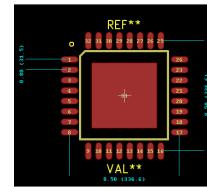
ផើម្ពីចិត្តប្រព័ន្ធបានជាបន្ទះ: PCB ត្រូវស្អងកាត់ ៣ ដំណាក់កាលខាងក្រោម

1



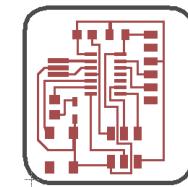
Schematic

2



Footprint

3

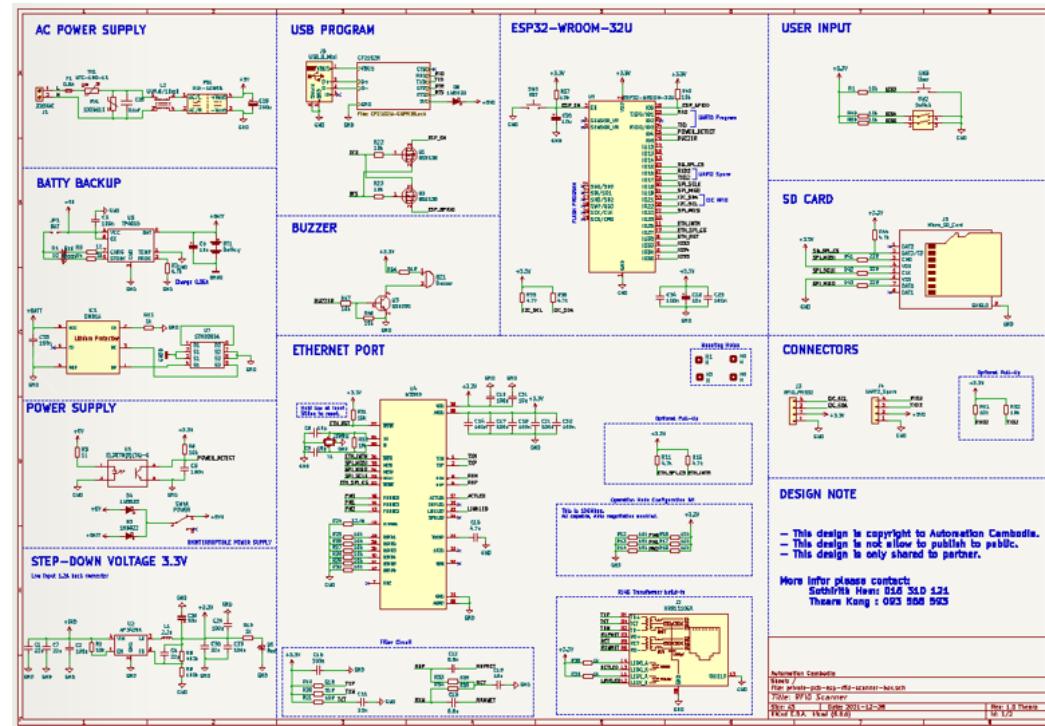
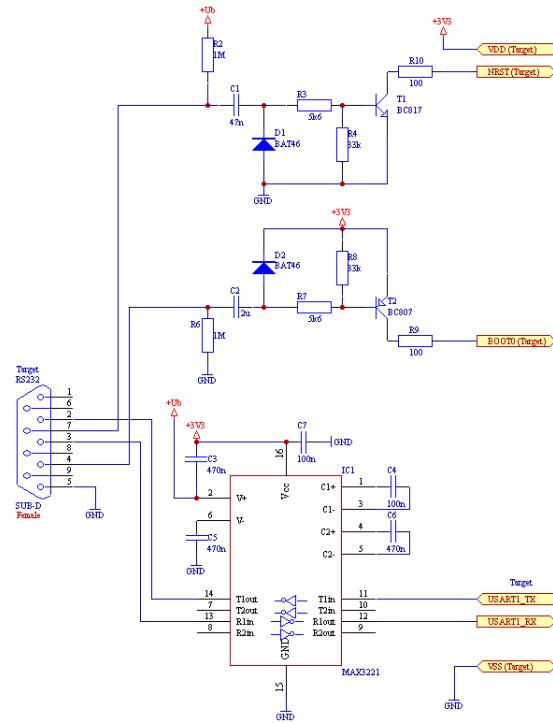


PCB

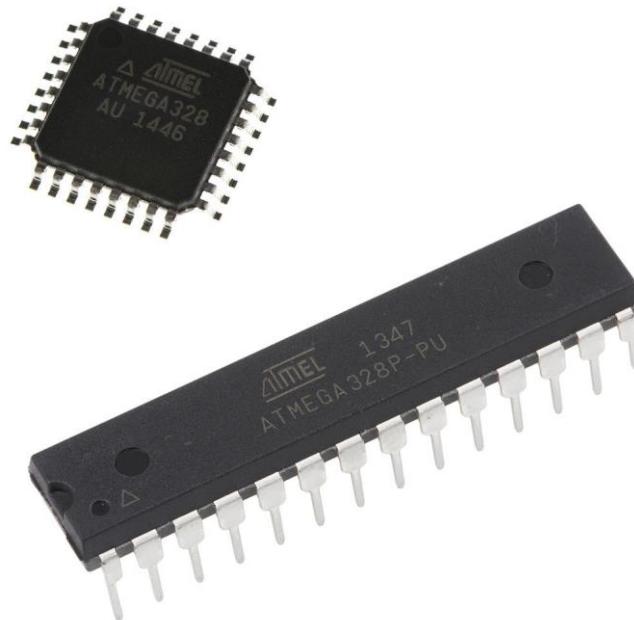
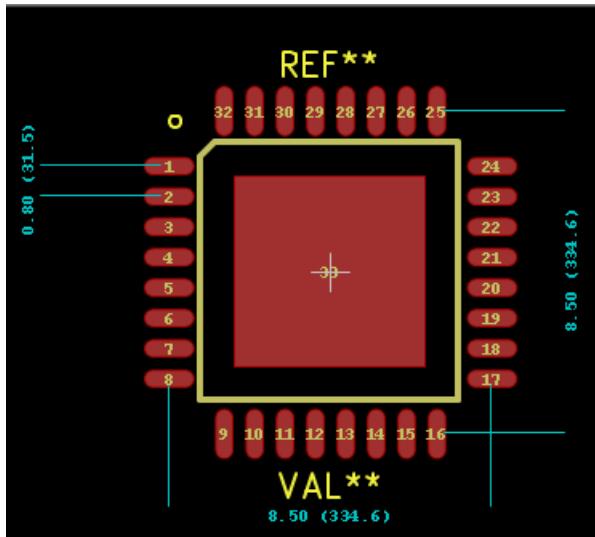
1. Introduction

ដំណាក់ការជើងប្លាសជា PCB

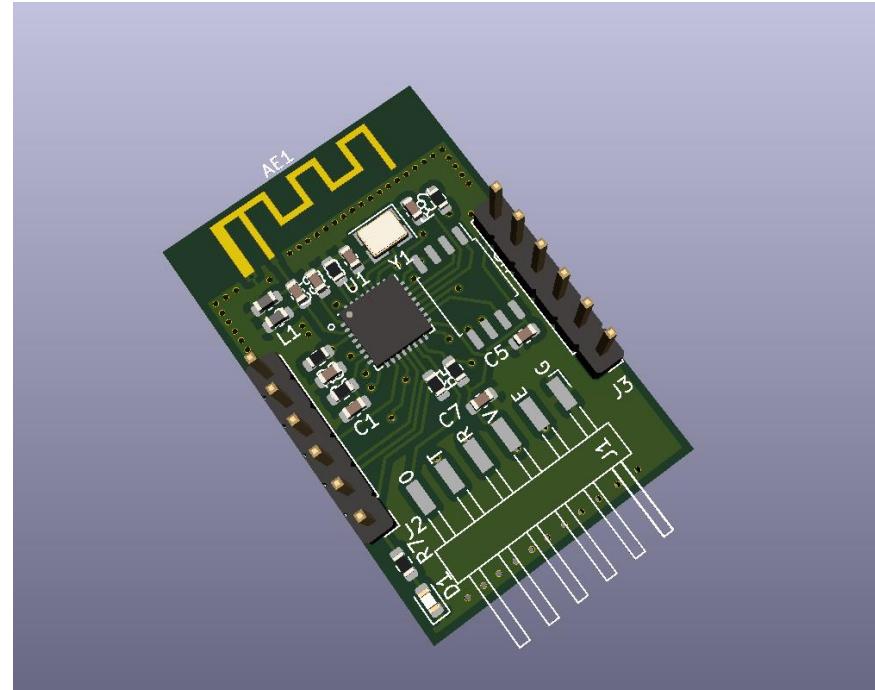
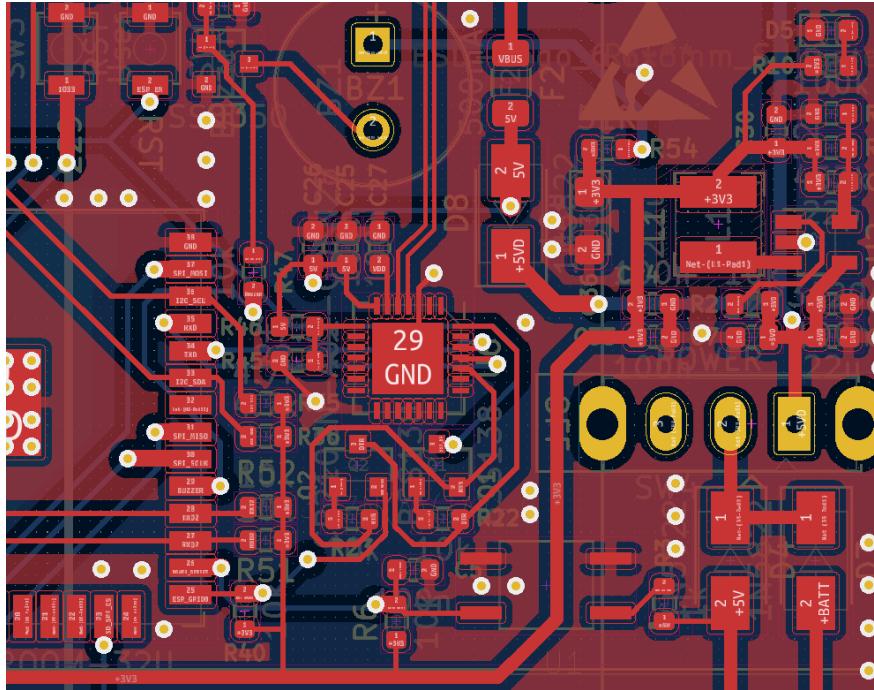
1. Schematic



2. Footprint



3. PCB (Printed Circuit Board)



៤ ចំនួចផើម្ពីគាថចេច: ខ្លួន PCB

1



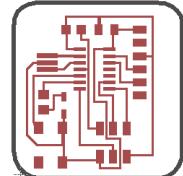
ចេច: ក្រុមហ៊ុន

2



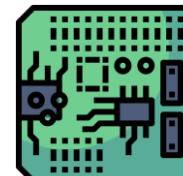
អេឡិចត្រូនិក

3



ខ្លួន

4

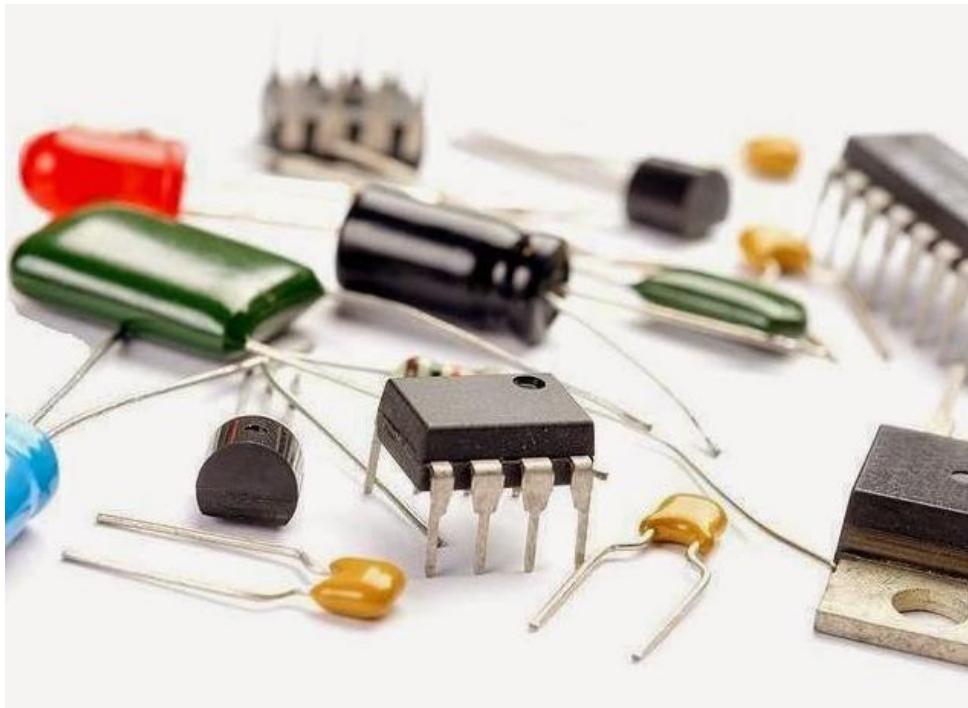


ផលិត

1. កម្មវិធី



2. អេឡិចត្រូនិច



What is datasheet ?

IRFZ44 N-Channel MOSFET Transistor

FEATURES

- Drain Current -10mA@ T_J(25°C)
- Drain Source Voltage
 $V_{DS} = 50VDC$
- Static Drain-Source On-Resistance
 $R_{DS(on)} = 0.032\Omega$ (Max)
- Fast Switching

DESCRIPTION
Designed for low-voltage, high-speed switching applications in power supplies, converters and power motor controls, these devices are particularly well-suited for bridge circuits where close speed and commutating safe operating areas are critical and offer additional safety margin against unexpected voltage transients.

ABSOLUTE MAXIMUM RATINGS (at 25°C)		
SYMBOL	PARAMETER	VALUE
V _{DS}	Drain-Source Voltage	50
V _{GDS}	Gate-Drain Voltage-Cathode	±25
I _D	Drain Current Continuous	100
I _{GS}	Gate-Source Reverse Bias Current	100 nA

P_1	Total Dissipation at T=100°	W	W
T_c	Zero Operating Junction Temperature	°C	°C
T_{bg}	Storage Temperature	°C	°C
TERMAL CHARACTERISTICS			
SYMBOL	PARAMETER	UNIT	UNIT
$R_{D,J}$	Thermal Resistance, Junction-to-Case	°C/W	°C/W
$R_{D,B}$	Thermal Resistance, Junction-to-Board	°C/W	°C/W

See www.Wolfram.com

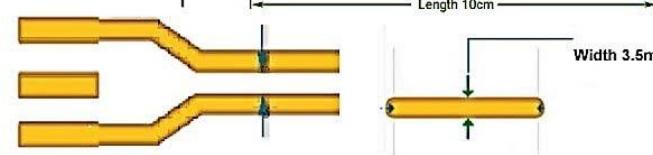
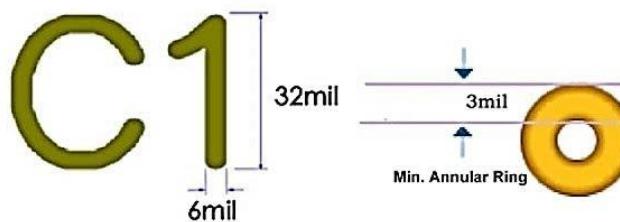
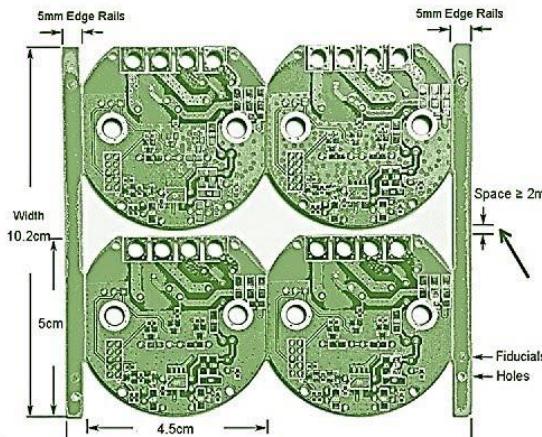
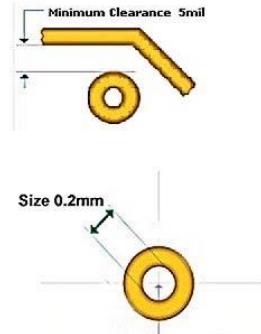
MOS TMS 25132-38 JL, TMS 25133-36 JL
LSI TMS 25132-45 JL, TMS 25133-45 JL
32768-BIT ERASABLE PROGRAMMABLE READ ONLY MEMORIES

第十一章 | 项目管理与组织行为

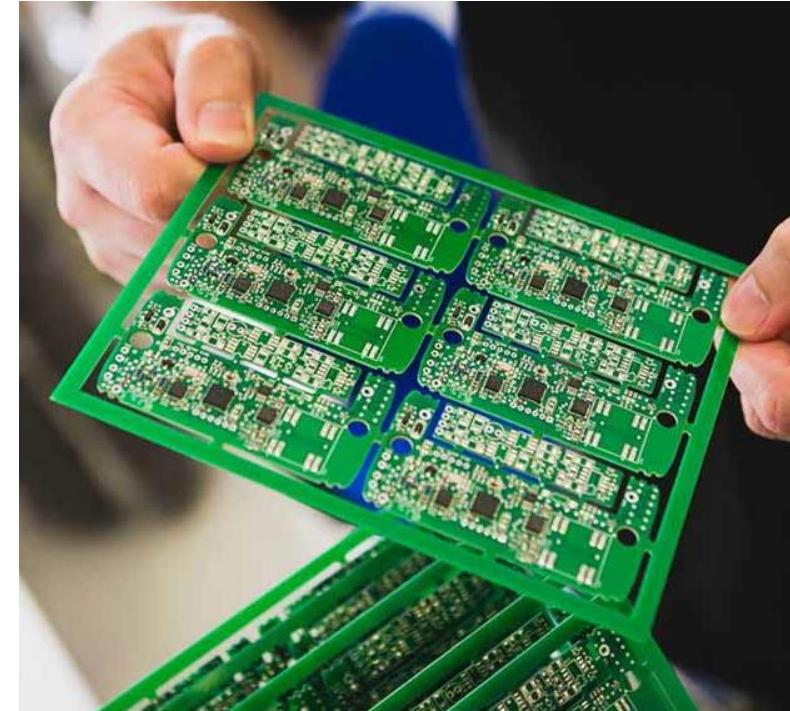
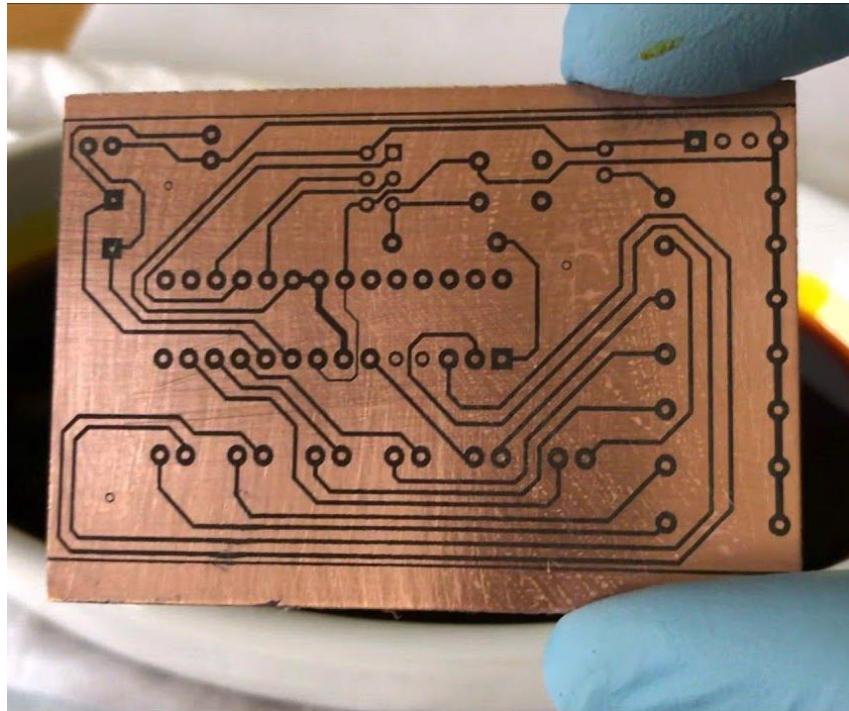
3. ខ្លួន PCB

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



4. ផលិតបន្ទះ PCB





- មិនគិតថ្លែង (មិនជាប់ License)
- មានស្ថើត្រប់ OS
- ក្រុមហ៊ុនដំឡើងក្នុង និងក្រោមស្ថុកប្រើប្រើន
- សំឡុះអ្នកប្រើប្រាស់ការនៃពេលវេលា

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.

Ubuntu	macOS	Windows
Debian	Flatpak	Arch Linux
Fedora	openSUSE	Linux Mint
Gentoo	FreeBSD	Source Code

The screenshot shows the KiCAD website's download section. At the top, there is a navigation bar with links for BLOG, DISCOVER, COMMUNITY, HELP, CONTRIBUTE, SPONSORS, LIBRARIES, DOWNLOAD, ABOUT, and a blue 'Donate' button. Below the navigation is a decorative banner featuring a colorful circuit board design. Underneath the banner, a breadcrumb navigation shows 'Home / Download'. The main title 'Download' is displayed in large, bold letters. Below it, the text 'Select your operating system or distribution' is followed by a grid of nine boxes, each containing an icon and the name of a supported operating system: Ubuntu (Ubuntu logo), macOS (Apple logo), Windows (Windows logo), Debian (Debian logo), Flatpak (Flatpak logo), Arch Linux (Arch Linux logo), Fedora (Fedora logo), openSUSE (openSUSE logo), and Linux Mint (Linux Mint logo).

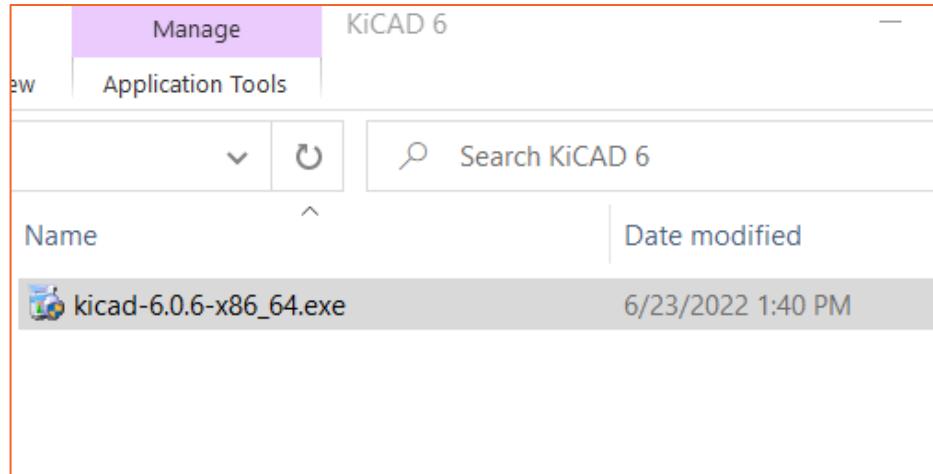
1. ទាញយកកម្មវិធី [Download](#) ដ្ឋីសិសតាម OS

The screenshot shows the KiCAD website's "Windows Downloads" section. At the top, there's a navigation bar with links for BLOG, DISCOVER, COMMUNITY, HELP, CONTRIBUTE, SPONSORS, LIBRARIES, DOWNLOAD, ABOUT, and a blue "Donate" button. Below the navigation is a breadcrumb trail: Home / Download / Windows Downloads. To the right, a sidebar titled "All Platforms" lists various operating systems with corresponding icons: Ubuntu (checkbox), macOS (apple icon), Windows (checkbox, highlighted in blue), Debian (circle icon), Flatpak (square icon), Arch Linux (triangle icon), Fedora (globe icon), openSUSE (square icon), Linux Mint (circle icon), Gentoo (circle icon), FreeBSD (circle icon), and Source Code (code icon). The main content area is titled "Windows Downloads" and notes that KiCAD supports Windows 8.1, 10, and 11, with a link to "System Requirements". It highlights the "Stable Release" version 6.0.6. A dropdown menu for "64-bit (recommended)" is open. The page is organized by region: "Worldwide" (OSDN, GitHub), "Europe" (CERN - Switzerland, Futureware - Austria), and "China" (AlibabaCloud, Chongqing University).

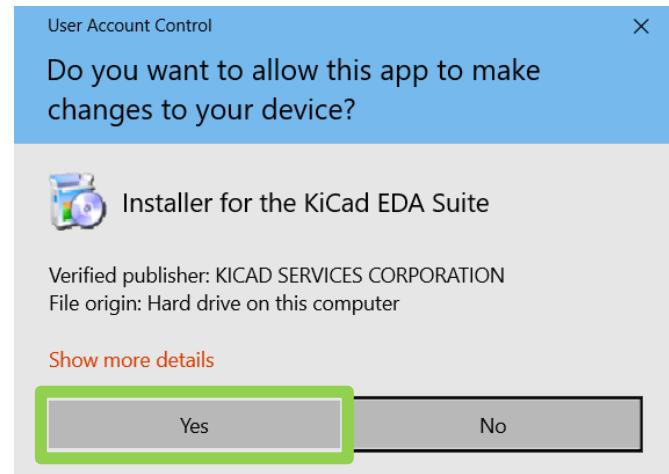
2. ផ្សេសនឹស Server ទាញយកដែលស្ថិតនៅជិតម្លក

2. Design Software

2.2 កម្មិែងកម្មវិធី KiCAD



3. Double Click លើកម្មវិធីដែលបានទាញយក



4. ចូច Yes ដើម្បីដំណឹក

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



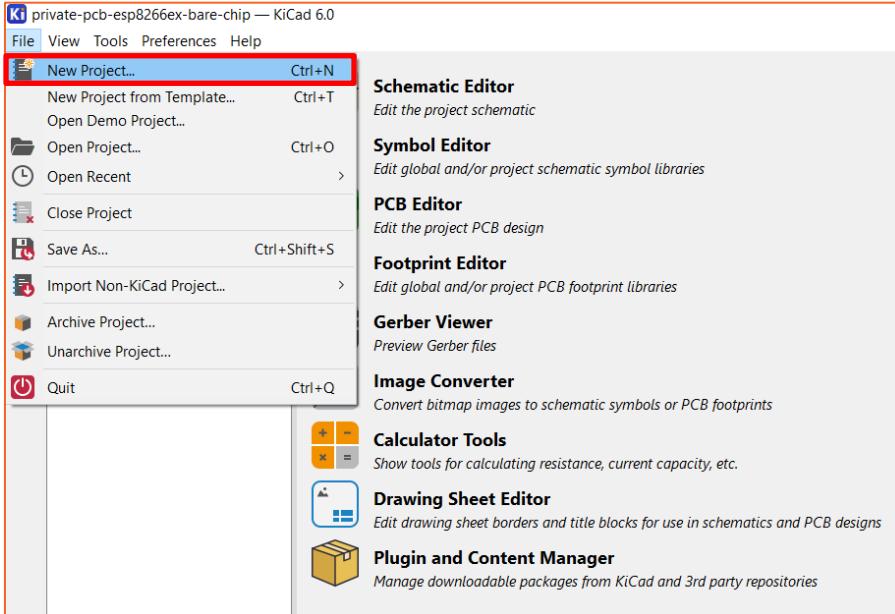
Plugin and Content Manager

Manage downloadable packages from KiCad and 3rd party repositories

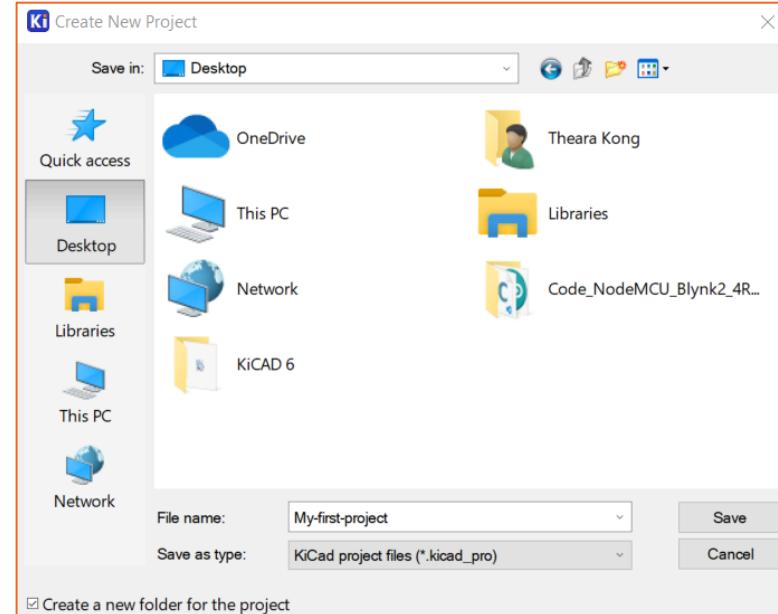
ចូលទៅក្នុង Symbol Editor ដើម្បីធ្វើការកែ ប្រហែត
Symbol ថ្មី

2. Design Software

2.3 មគ្គន Project ផ្ទុក



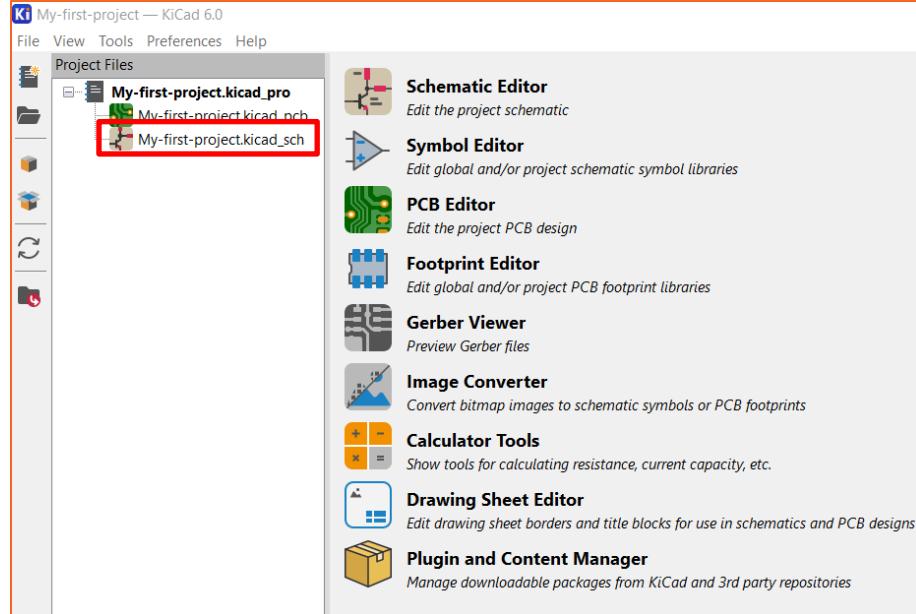
1. ចូចលើ File រួចរាលឹស New Project (Ctrl+N)



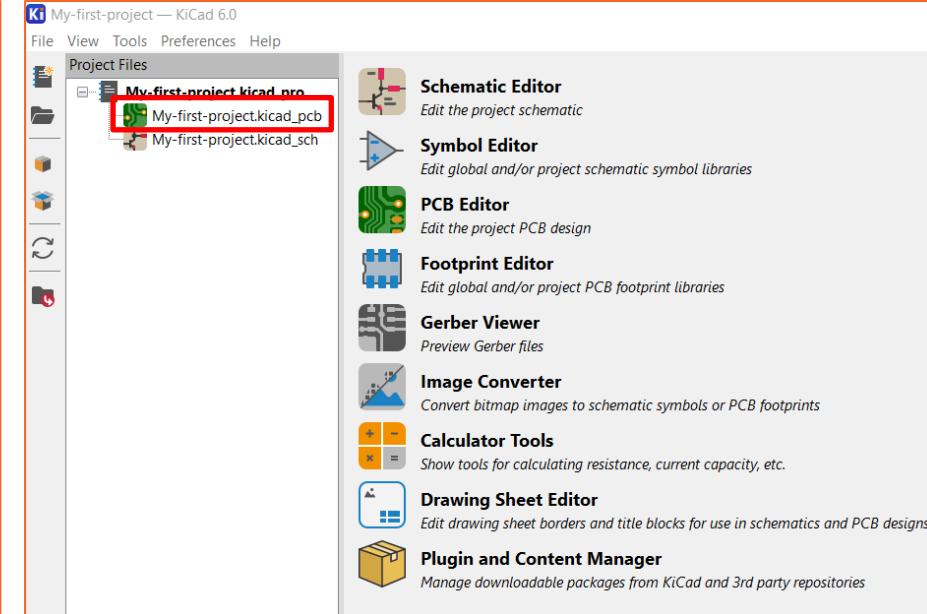
2. ជាក់លើកទីតាំង Project របស់អ្នក

2. Design Software

2.3 មគ្គន Project ផ្ទុក



3. ចូចលើ .kicard_sch ដើម្បីបើក Schematic



4. ចូចលើ .kicard_pcb ដើម្បីបើក PCB

2. Design Software

2.3 Tools សម្រាប់ផ្សេងៗ Schematic

The screenshot displays the user interface of a schematic design application, likely Altium Designer, with several toolbars and menus visible.

File (highlighted with a red border):

- Save (Ctrl+S)
- Save Current Sheet Copy As...
- Insert Schematic Sheet Content...
- Import
- Export
- Schematic Setup...
- Page Settings...
- Print... (Ctrl+P)
- Plot...
- Close

Place (highlighted with a red border):

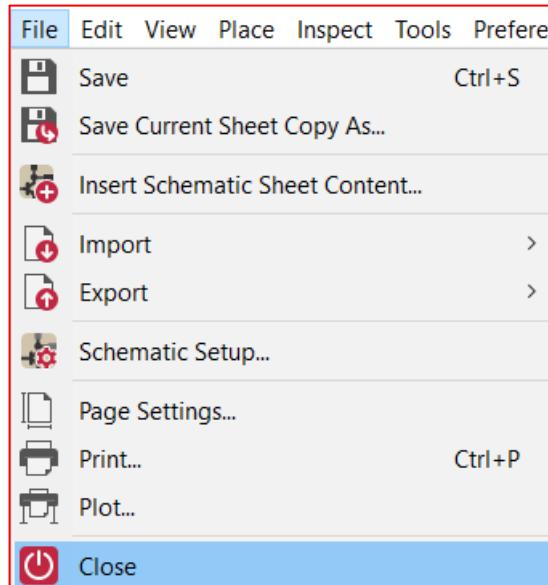
- Add Symbol
- Add Power
- Add Wire
- Add Bus
- Add Wire to Bus Entry
- Add No Connect Flag
- Add Junction
- Add Label
- Add Global Label (Ctrl+L)
- Add Hierarchical Label
- Add Sheet
- Import Sheet Pin
- Add Lines
- Add Text
- Add Image

Inspect (highlighted with a red border):

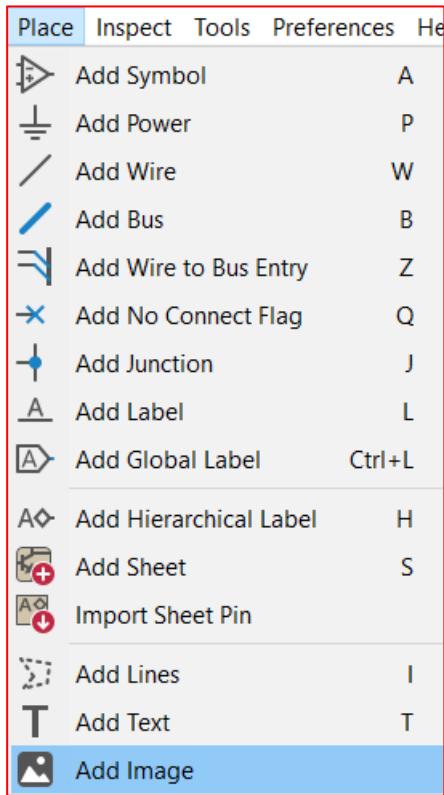
- Electrical Rules Checker
- Previous Marker
- Next Marker
- Exclude Marker
- Simulator...

Tools (highlighted with a red border):

- 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...

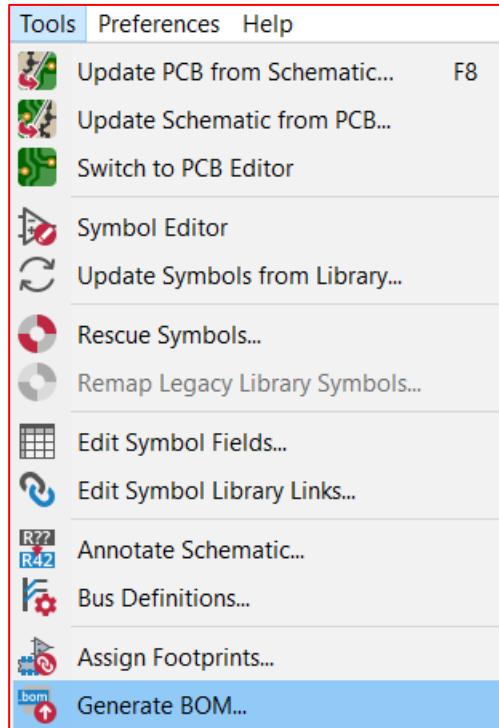


- **Save** រក្សាទុកនៅលើ File ដីផែល
- **Save Current Sheet Copy As** រក្សាភ្លាហាតស់កំពុង Edit ទៅកន្លែងដោយ
- **Insert Schematic Sheet Content** បញ្ចូលក្រដាស់ផែលមានស្រាប់
- **Import** ដាក់បញ្ចូល File
- **Export** បញ្ចូល File
- **Schematic Setup** កំណត់ពីសេច្ញា
- **Page Setting** កំណត់ពីទំនួរបស់សេច្ញា
- **Print** ព្រឹនសេច្ញា
- **Plot** បញ្ចូលសេច្ញាដោយ File ដោយ
- **Close** បិទសេច្ញា



អក្សរនៃខាងស្តាំនៃ Shortcut នៃការគម្រោង (Tools) នេះ:

- **Add Symbol** បញ្ចូលទេសដីសម្រាប់ជួយដាក់លើសម្រាប់ការគម្រោង
- **Add Power** បញ្ចូលទេសដីសម្រាប់ជួយដាក់លើសម្រាប់ការគម្រោង
- **Add Wire** គ្រប់គ្រងផ្ទាល់រំភាព
- **Add Bus** គ្រប់គ្រងផ្ទាល់បុស
- **Add Wire to Bus Entry** គ្រប់គ្រងផ្ទាល់ទៅបុស
- **Add No Connect Flag** ជាក់និមិត្តសញ្ញាបញ្ជាក់ថាមិនមានភាព
- **Add Junction** ជាក់ចំនួចភាព
- **Add Label** ជាក់អក្សរសម្គាល់ខ្លួយ (ស្ថាល់ពេកកុងក្រដាស់កំពុងធ្វើការ)
- **Add Global Label** ជាក់អក្សរសម្គាល់ខ្លួយ (ស្ថាល់ពីគ្រប់ក្រដាស់)
- **Add Hierarchical label** ជាក់អក្សរសម្គាល់របស់ក្រដាស់
- **Add Sheet** បង្កើតក្រដាស់ថ្មី
- **Add Line** គ្រប់គ្រងផ្ទាល់បន្ទាត់

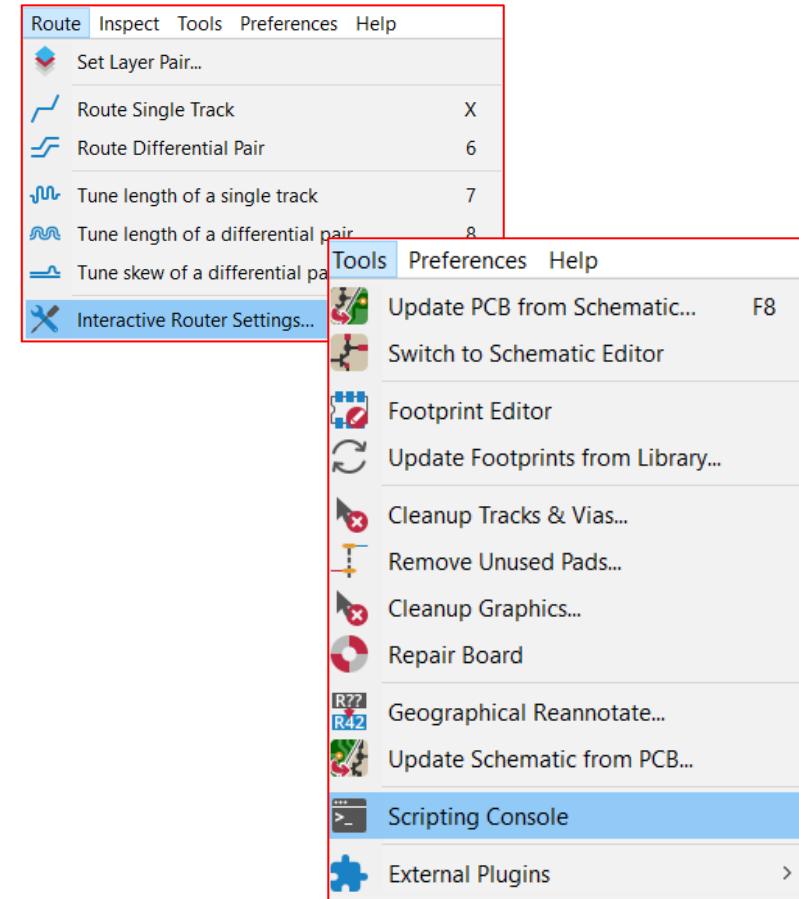
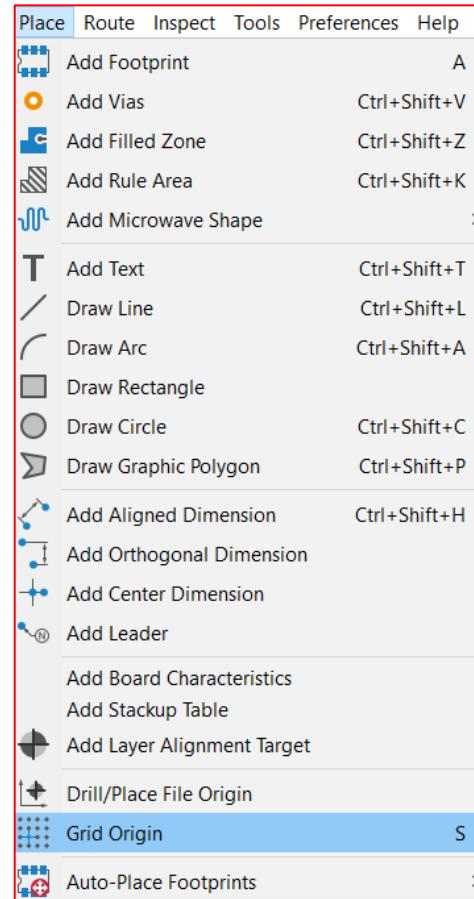
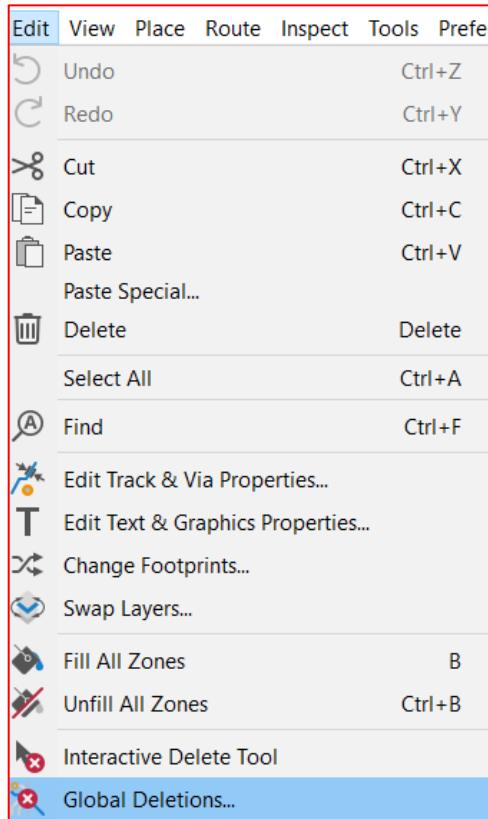


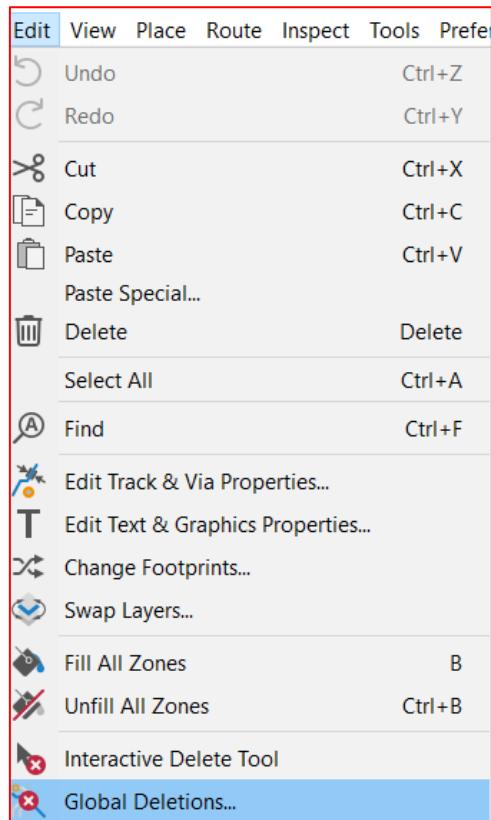
អក្សរនៅខាងស្តាំជា Shortcut ទៅកាន់ Tools នេះ:

- **Update PCB from Schematic** Update PCB ពីសេចក្តី
- **Update Schematic from PCB** Update សេចក្តី ពី PCB វិញ
- **Switch to PCB Editor** ប្រភែផ្ទាំងកែ PCB
- **Symbol Editor** កែសម្រួល Symbol
- **Annotate Schematic** ផ្គត់ផ្គង់សេចក្តី
- **Bus Definition** ដាក់លេខាជាយខ្លួយ Bus
- **Assign Footprints** កំណត់ Footprints
- **Generate BOM** បង្កើតតារាងការបង្រៀនគ្រឿង Build Of Material

2. Design Software

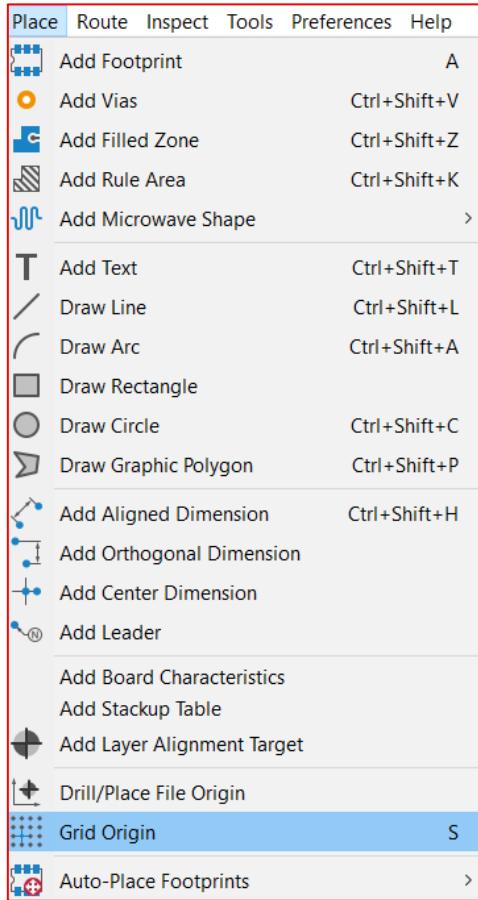
2.4 Tools សម្រាប់ផ្សេងៗ PCB





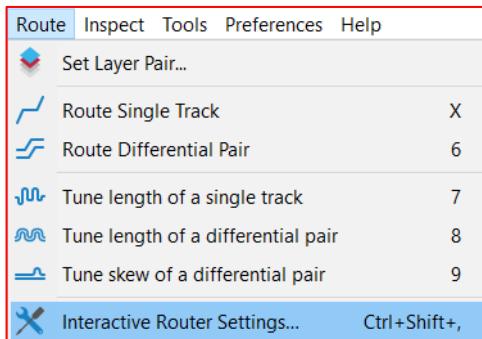
អក្សរនៃខាងស្តាំជា Shortcut ទៅកាន់ Tools នេះ:

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



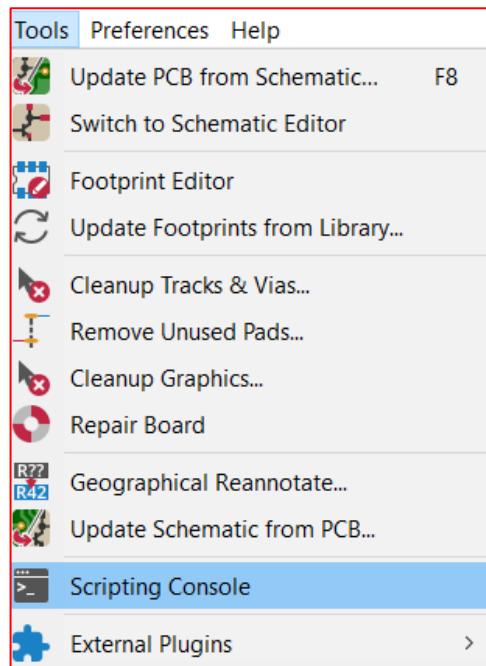
អក្សរនៅខាងស្តាំជាក់ Shortcut ទៅកាន់ Tools នេះ:

- **Add Footprint** ទាញយក Footprint ពី Library
- **Add Vias** ដាក់វានូចម្បង
- **Add Filled Zone** ដាក់ផ្ទៃទែងដែង
- **Add Rule Area** ដាក់កំបន់ដែលមានការកំណត់
- **Add Microwave Shape** ប្រើប្រាយរួមចំណាំ Microwave
- **Text** ដាក់អក្សរ
- **Draw Line** គូសបន្ទាត់
- **Draw Arc** គូសខ្សោយការង
- **Draw Rectangle** គូសចត្តកាលកំណង
- **Draw Circle** គូសរដ្ឋង់
- **Add Aligned Dimension** ដាក់រដ្ឋាភិបាលដែលត្រួត
- **Add Leader** ដាក់កំណត់ចំណាំ



អក្សរនៅខាងស្តាំជា Shortcut ទៅកាន់ Tools នេះ:

- Set Layer Pair កំណត់ Layer
- Route Single Track រត្តខ្សែមួយសំរែស
- Route Differential Pair រត្តខ្សែគុណ
- Tune length of a single track ធ្វើការកែសម្រួលប្រដៃងខ្សែមួយ
- Tune length of a differential pair កែសម្រួលប្រដៃងខ្សែគុណ
- Tune Skew of a differential pair កែសម្រួលប្រដៃងគុរបស់ខ្សែគុណ
- Interactive Router Setting គ្នាកំណត់របៀបនៃការរត្តខ្សែ

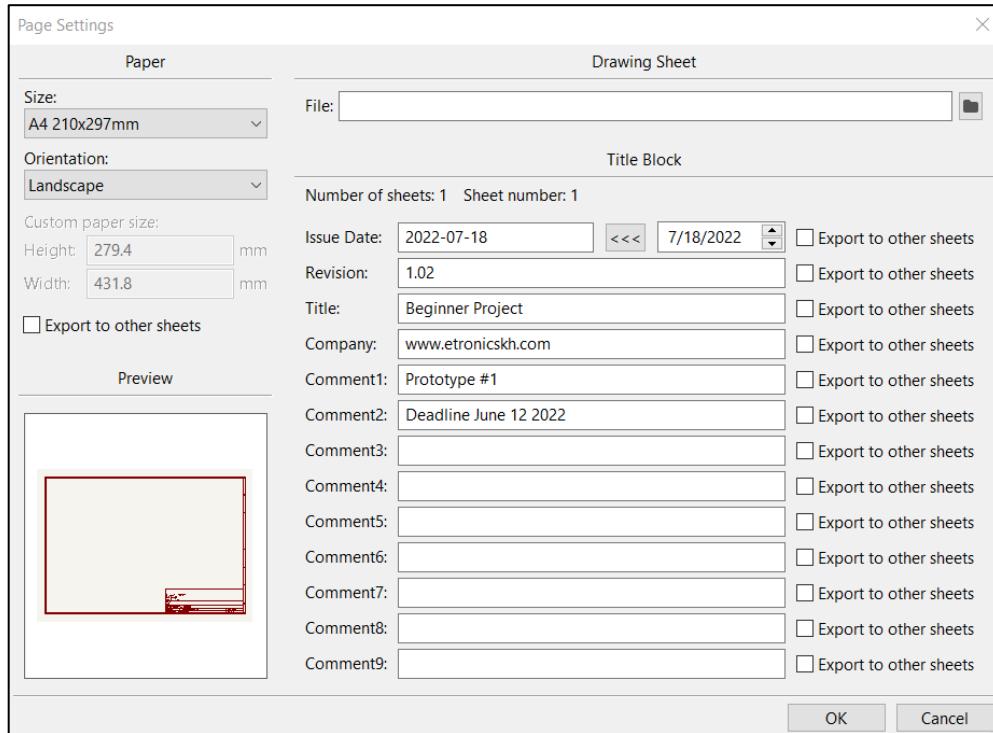


អក្សរនេះខាងស្តាំជា Shortcut ទៅកាន់ Tools នេះ:

- **Update PCB from Schematic** Update PCB ពី Schematic
- **Switch to Schematic Editor** ចូរទៅធ្វើការលើ Schematic
- **Footprint Editor** កែសម្រួល ប្រហែតិត 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 ផ្លូវដំឡើង

3. Schematics Design

3.1 ការរួមចំពោន់ (Sheet)



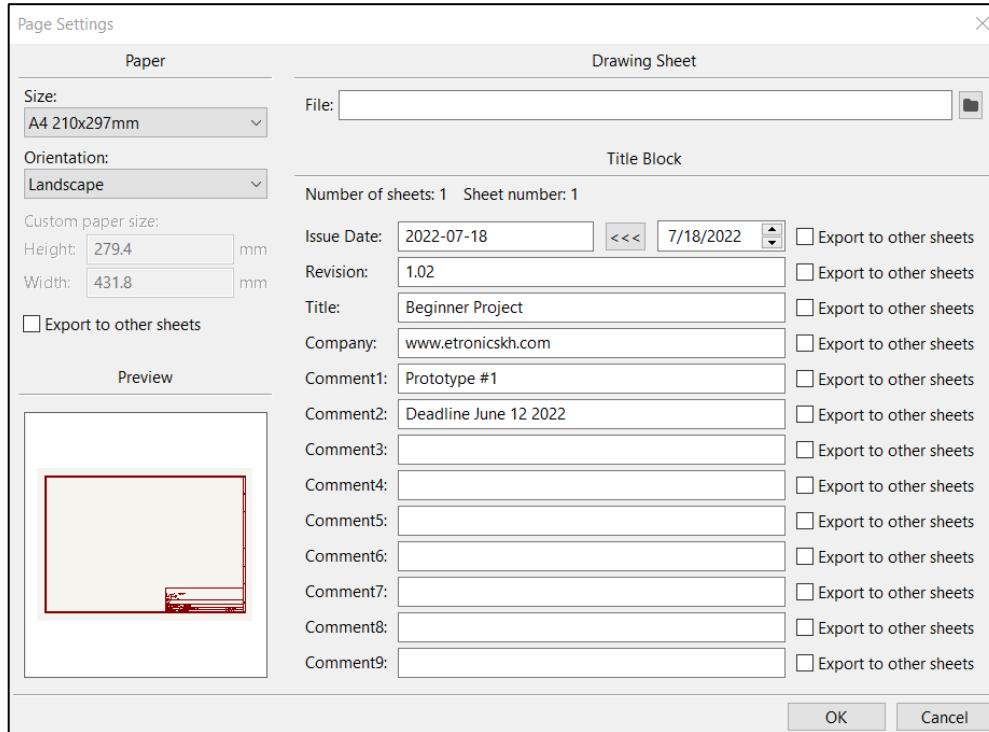
Deadline June 12 2022	Prototype #1	www.etronicskh.com
Sheet: /	File: My-first-project.kicad_sch	
Title: Beginner Project		
Size: A4	Date: 2022-07-18	Rev: 1.02
KiCad E.D.A. kicad (6.0.6)		Id: 1/1

Designe 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 សម្រាប់បង្ហាញព័ត៌មានពីខ្លួន
របស់គ្មាន រួមទាំងព័ត៌មានរបស់អ្នកខ្លួន។

3. Schematics Design

3.1 ការរួមចំព្រផស (Sheet Setting)



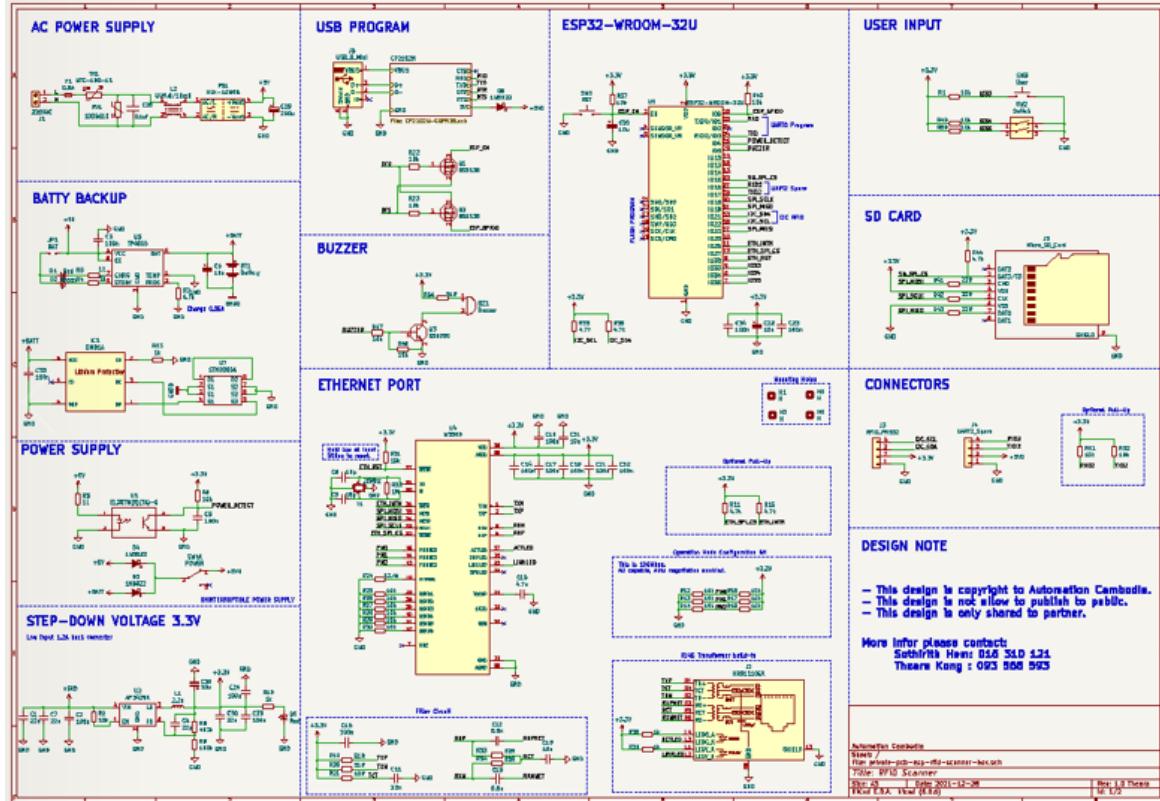
Deadline June 12 2022	Prototype #1	www.etronicskh.com
Sheet: /	File: My-first-project.kicad_sch	
Title: Beginner Project		
Size: A4	Date: 2022-07-18	Rev: 1.02
KiCad E.D.A. kicad (6.0.6)		Id: 1/1

Designe 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 សម្រាប់បង្ហាញព័ត៌មានពីខ្លួន
របស់គ្មាន រួមទាំងព័ត៌មានរបស់អ្នកខ្លួន។

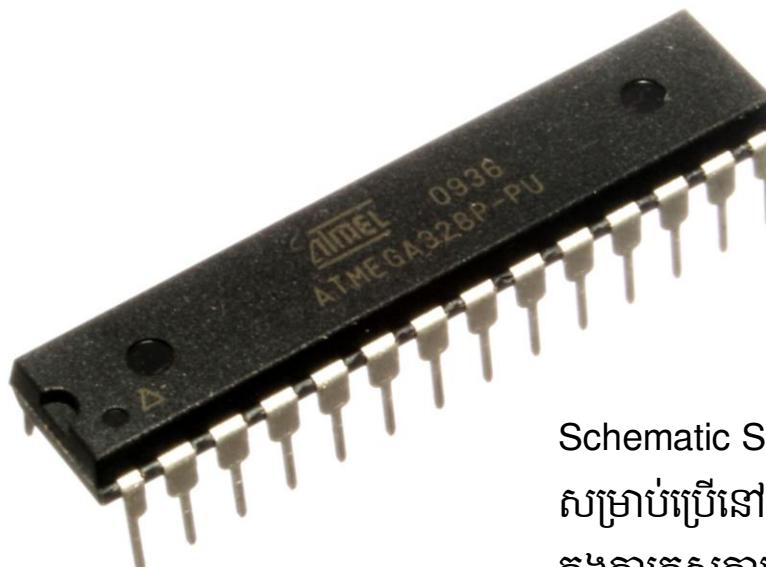
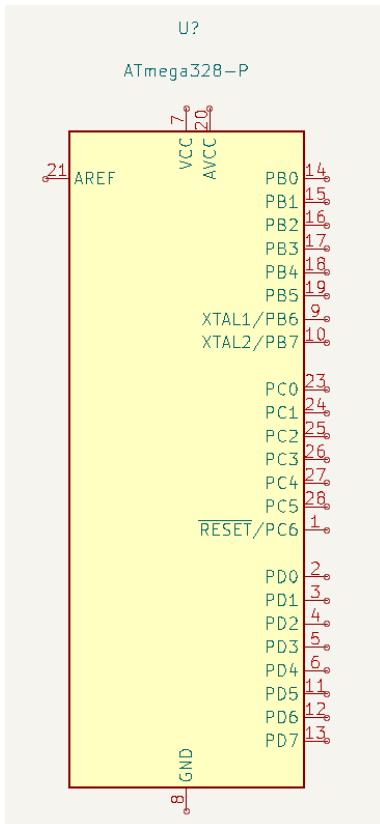
3. Schematics Design

3.1 ការរួចចំក្រដាស (Sheet Layout)



Page Layout ជាការរួចចំក្រដាសសំឡើង
ដោយស្នូលមេីន។ នេះការរួចចំទ្វាមាន
តំបន់ដូចជា៖

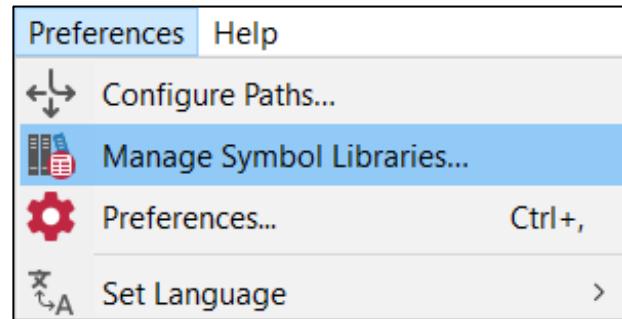
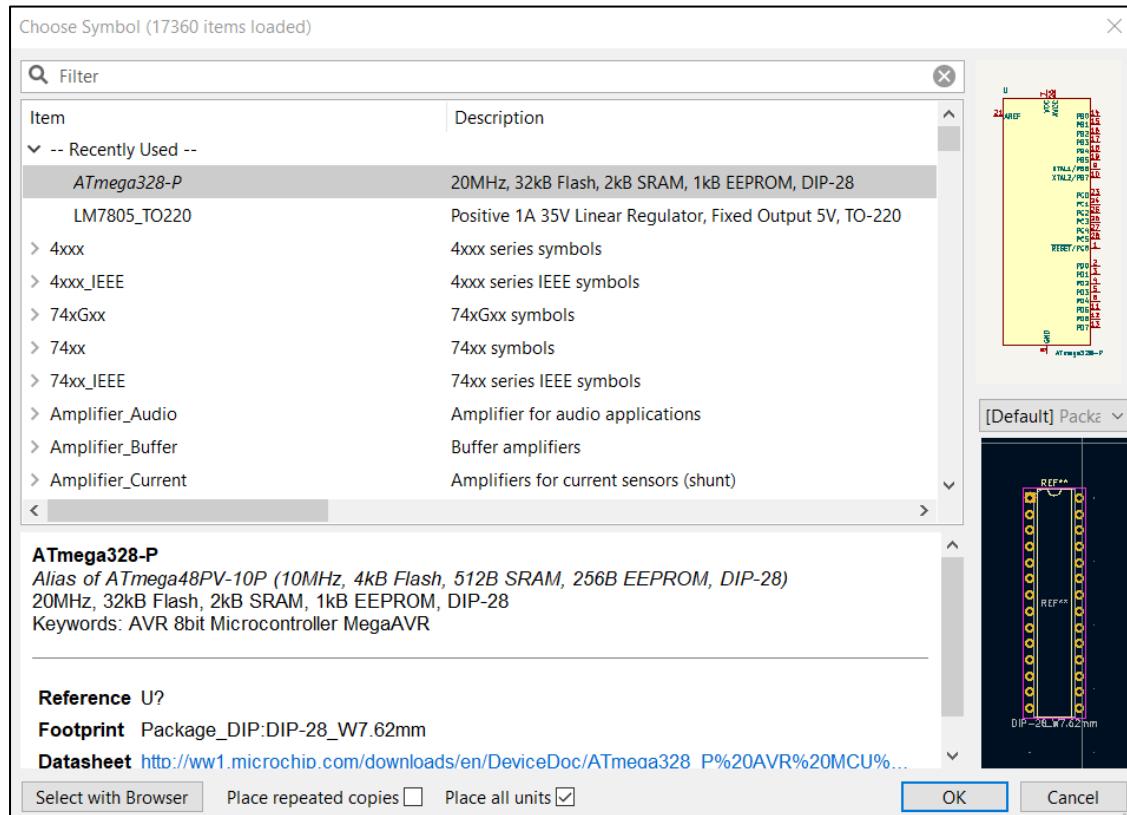
- Input Terminal
- Power Supply
- Voltage regulation
- Main circuit
- Additional circuit
- User Input
- Indicators
- Output Terminal



Schematic Symbol ជានិមិត្តសញ្ញា
សម្រាប់ប្រើនៅលើសេចក្តី ដើម្បីអាយក្រុណ
ក្នុងការគូសភ្លាប់។ ស្មើកែទាំងអស់នេះនិមិត្ត
សញ្ញាមិនធ្វើឡើនឹងរួចរាល់ពីការបស់គ្រឿង
នោះទេ។

3. Schematics Design

3.3 Manage Symbol Libraries

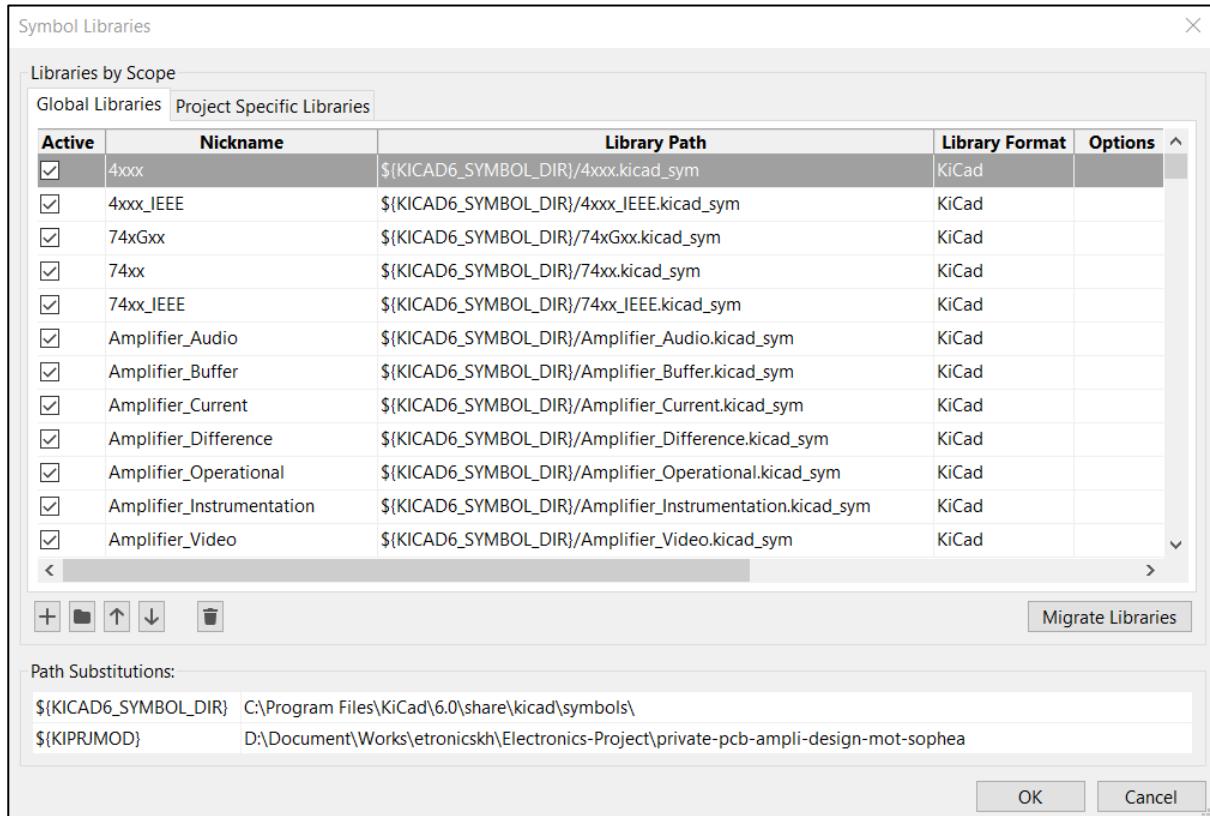


Libraries ជាតារាង Symbol សម្រាប់កម្មវិធីទាំងអស់ដែលត្រូវបានប្រើប្រាស់ក្នុងគម្រោង។
Libraries ផ្ទេរជាប្រព័ន្ធឌែលត្រូវបានប្រើប្រាស់ក្នុងគម្រោង។

- Global ប្រព័ន្ធឌែលត្រូវបានប្រើប្រាស់ក្នុងគម្រោងនៅលើក្នុងគម្រោង។
- Project ប្រព័ន្ធឌែលត្រូវបានប្រើប្រាស់ក្នុងគម្រោងដែលបានជាក់បញ្ចប់។

3. Schematics Design

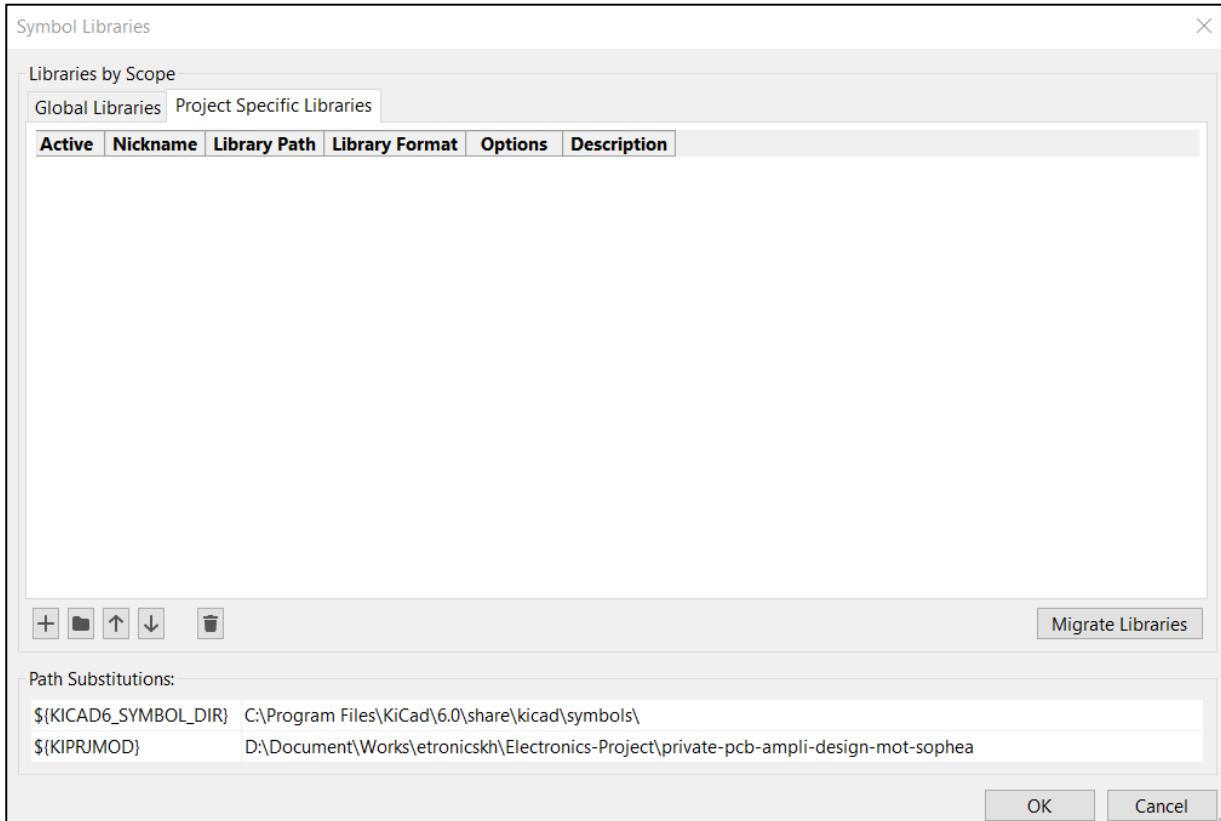
3.3.1 Global Libraries



នៅពេលដែល Libraries ត្រូវបាន
បញ្ចប់ទៅក្នុង Global Libraries នៅ:
វានឹងអាចបោរយកទៅប្រើបានពីគ្រប់
គម្រោងនៅក្នុងកំព្យចំរ។

3. Schematics Design

3.3.2 Project Specific Libraries



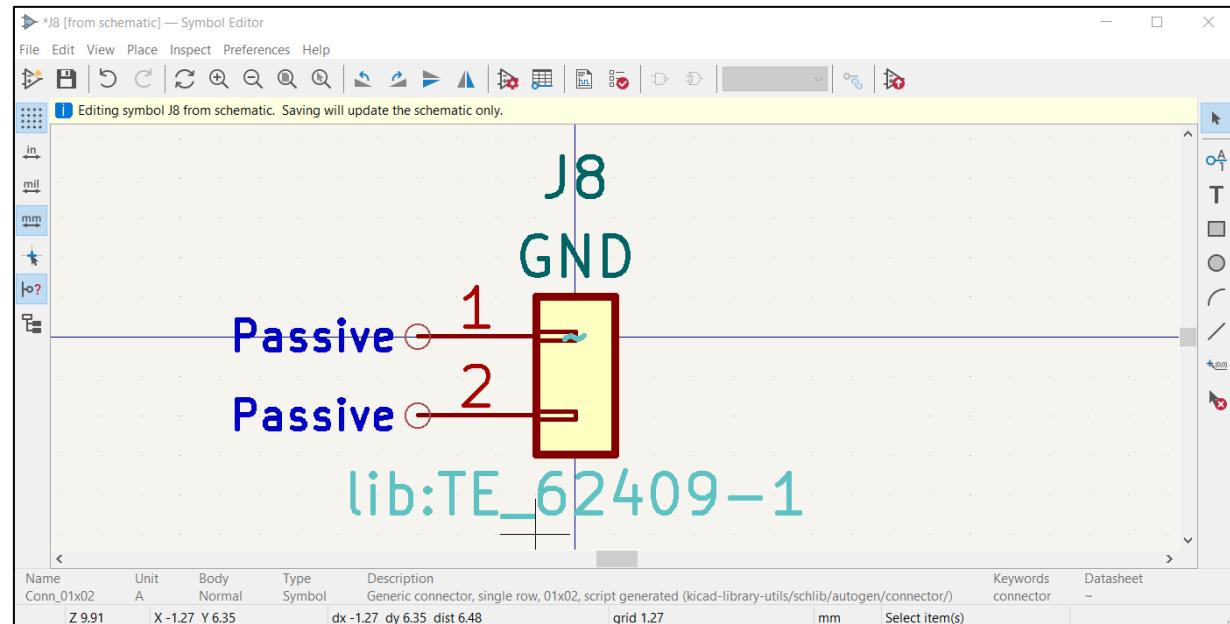
Symbol ដែលបានជា Project Libraries ប្រើបានតែគម្រោងដែលបានជាក់វាចូលជា Libraries ។ ភាគធ្វើនេះផ្តល់ជាបញ្ហាលទៅក្នុង Directory របស់គម្រោង ដើម្បីងាយស្មែលក្នុងការរចកវិប័យ។

3. Schematics Design

3.3.3 Edit Symbol

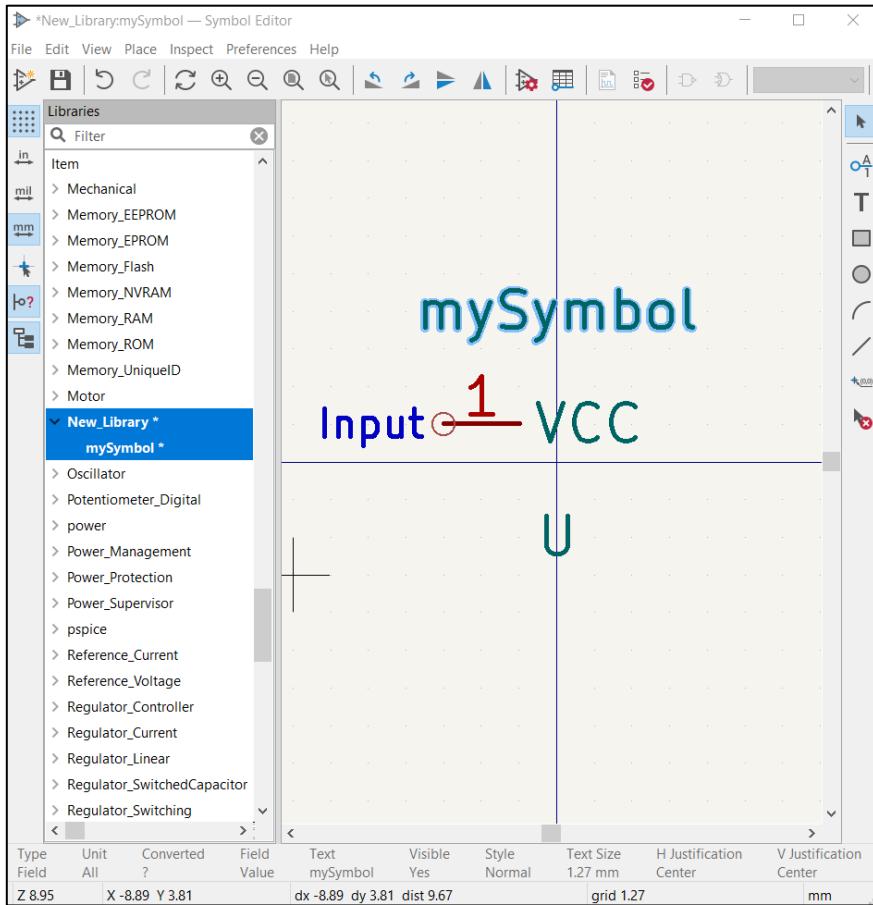
Move	M
Drag	G
Align Elements to Grid	
Rotate Counterclockwise	R
Rotate Clockwise	
Mirror Vertically	Y
Mirror Horizontally	X
Properties...	E
Edit Reference Designator...	U
Edit Value...	V
Edit Footprint...	F
Autoplace Fields	O
De Morgan Conversion	
Edit with Symbol Editor	Ctrl+E
Change Symbol...	
Update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

ធ្វើរីកកិច្ច Symbol ដែលស្ថិតនៅលើ Sheet ត្រូវបានកែចង្វារដោយគ្មានតំបន់ Right Click នូវក្រុមហ៊ុនកិច្ចដើម្បី "Edit With Symbol Editor"

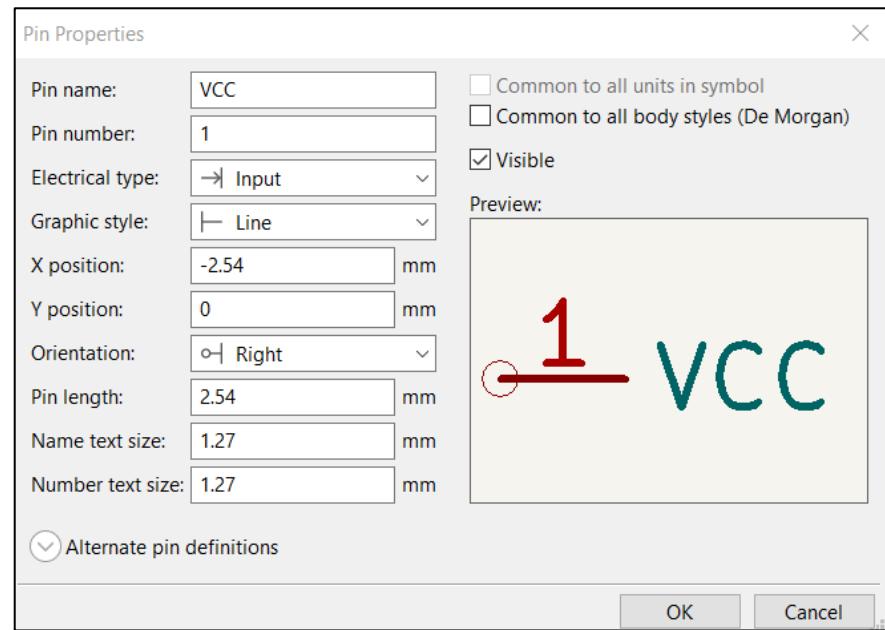


3. Schematics Design

3.3.3 Create Symbol

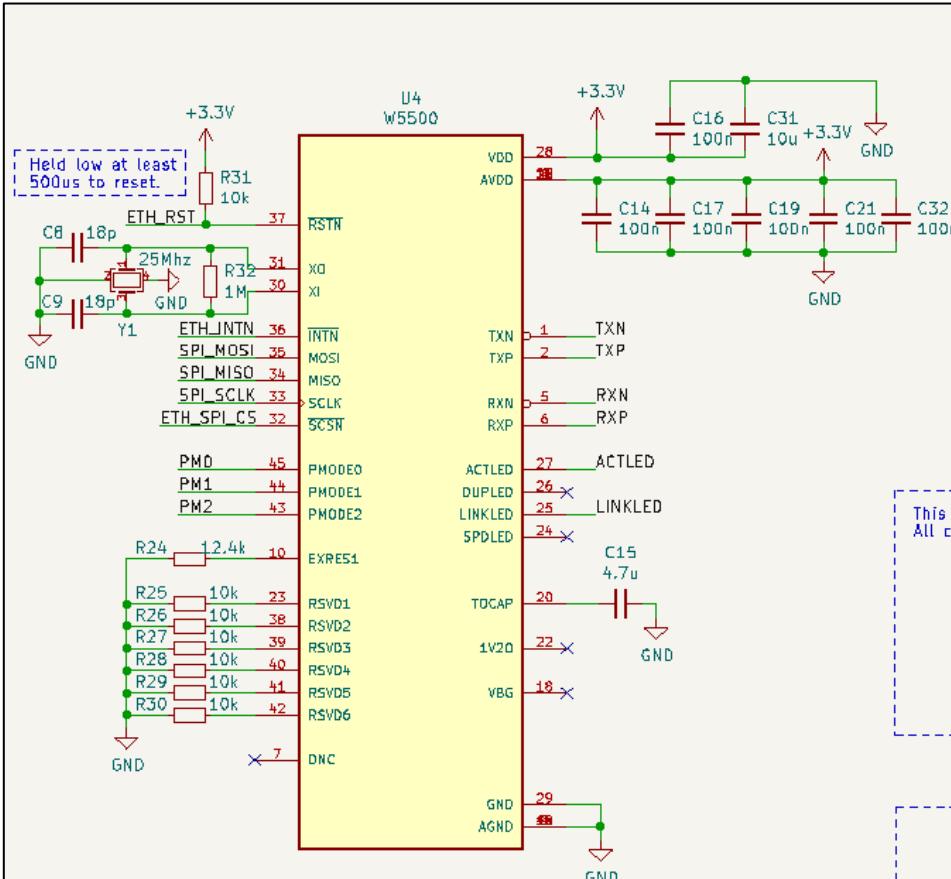


ធ្វើការបង្កើត Library ដើម្បី ដោយជាក់លេខា៖ និងអក្សរតាង និមិត្តសញ្ញា វិចធ្វើការគ្នាប់ Symbol ដោយប្រើប្រាស់ ឧបករណ៍នៅផ្ទៃកខាងស្តាំនៃផ្ទាំងបង្កើត Library ។



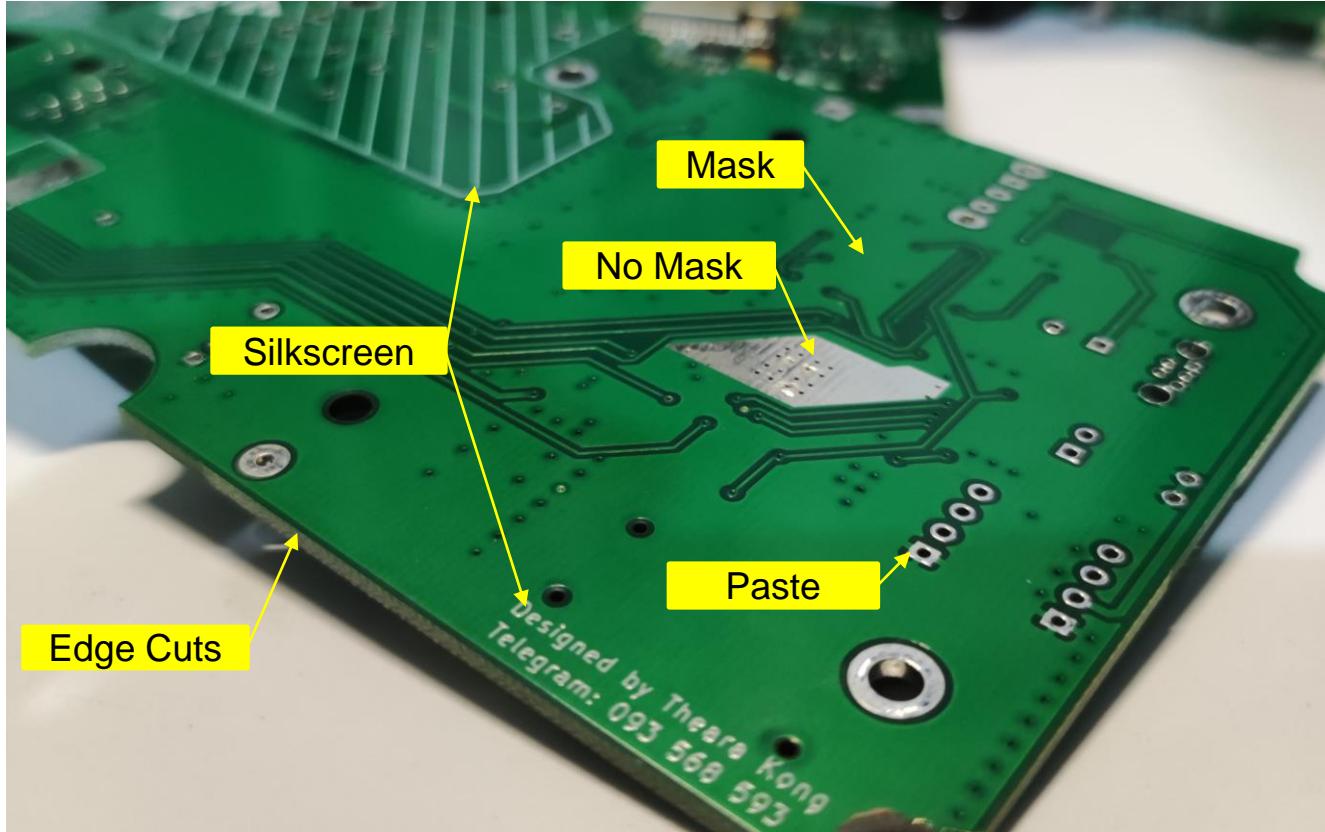
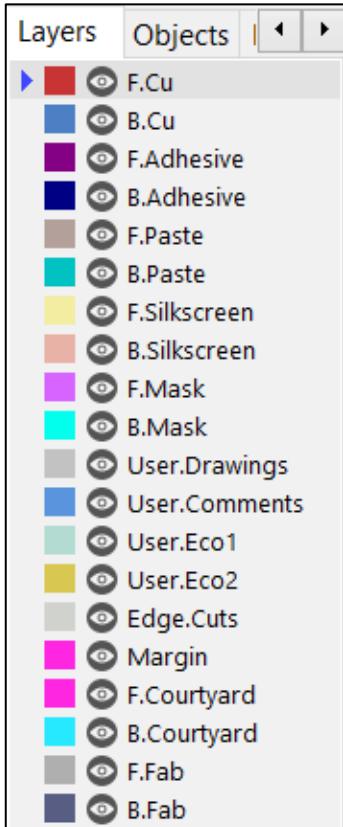
3. Schematics Design

3.4 Schematic Diagram



ដើម្បីបានដោក្រាមលូ ដោក្រាមត្រូវមានដូចខាងក្រោម៖

- តំបន់ផ្លួងគ្នា
- អាចមើលត្រូវអក្សរបានច្បាស់
- មិនមានខ្សោយកាត់គ្នាថ្វីន
- មានអក្សរកត់ចំណាំ
- ថែកជាថ្មី Power Symbol
- បញ្ជាក់ពីដើម្បីដែលមិនប្រើ
- ប្រើប្រាស់ Symbol ដូចខាងក្រោម



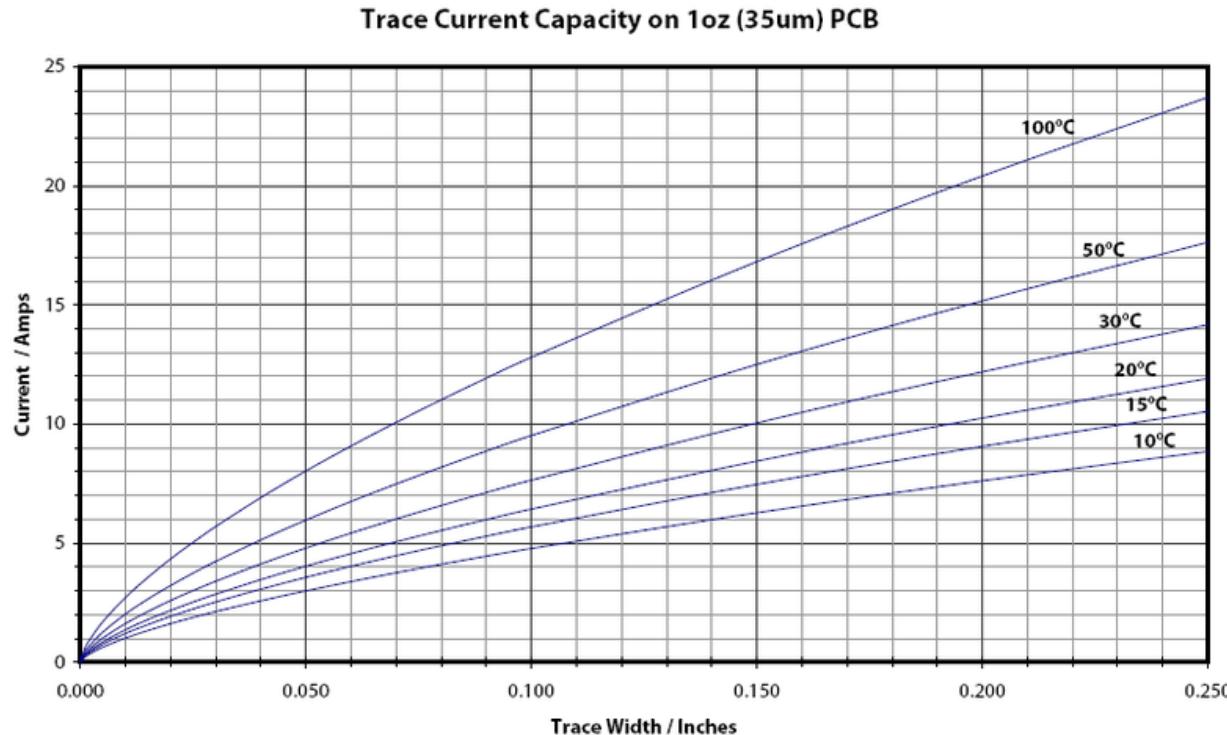
4. PCB Design

4.2 Trace (and Current Rate)

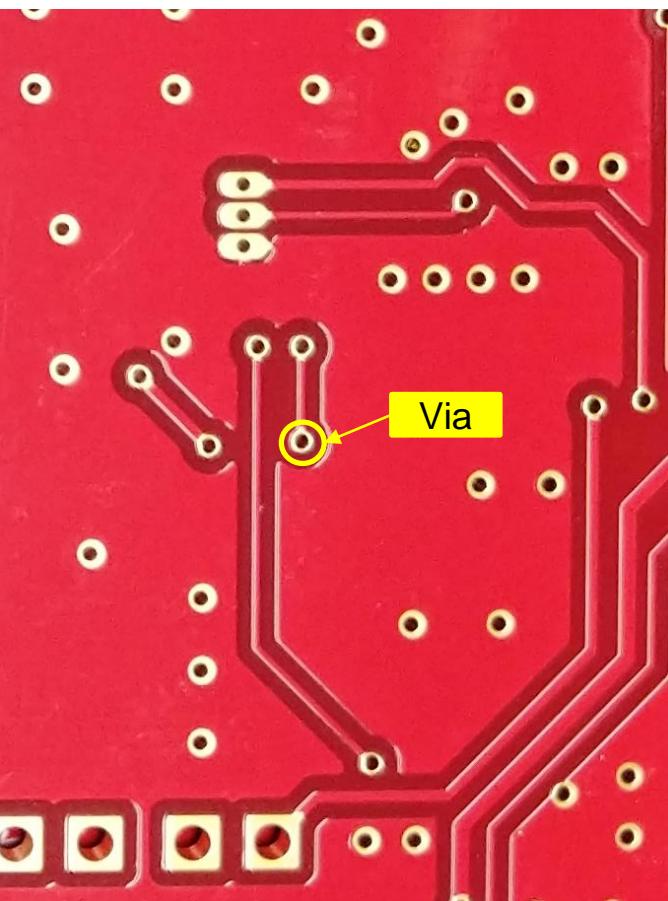


IPC Recommended Track Width For 1 oz cooper PCB and 10 °C Temperature Rise

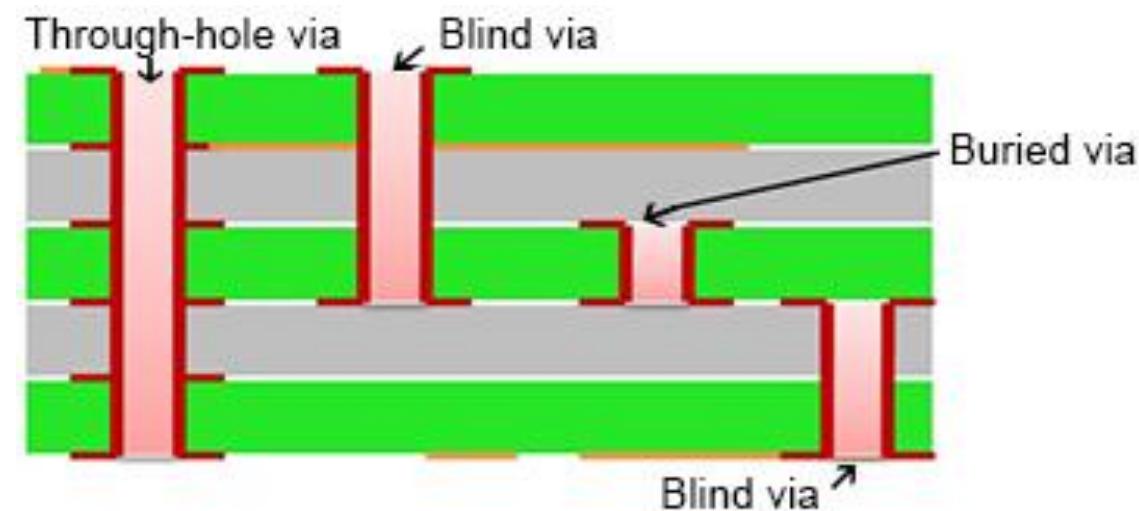
Current/A	Track Width(mil)	Track Width(mm)
1	10	0.25
2	30	0.76
3	50	1.27
4	80	2.03
5	110	2.79
6	150	3.81
7	180	4.57
8	220	5.59
9	260	6.60
10	300	7.62

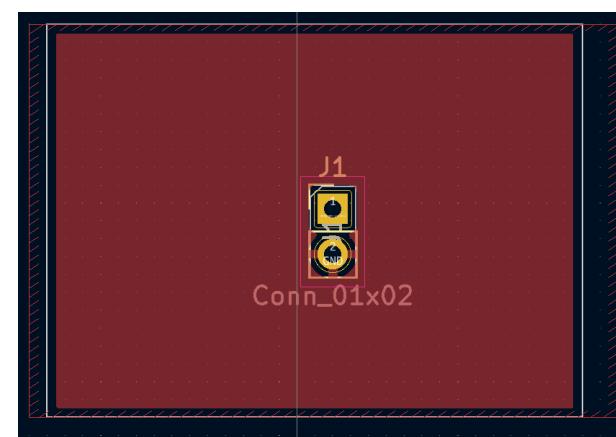
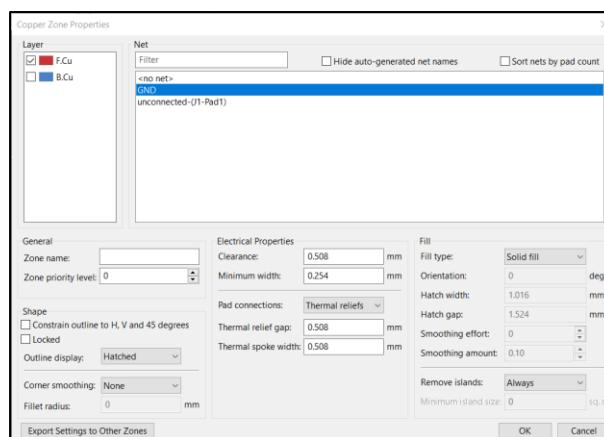
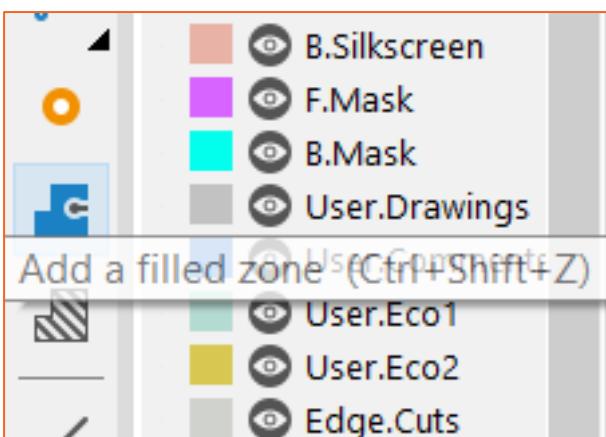
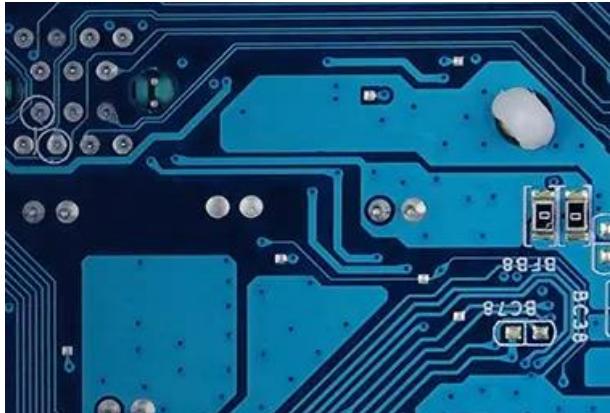


[Online Calculator](#)



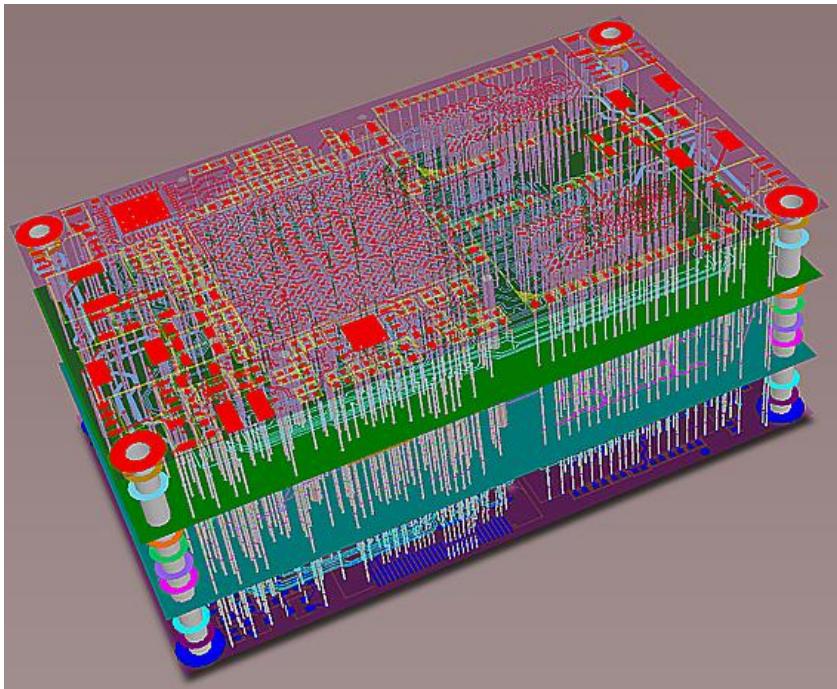
Via ត្រូវដើរឲ្យចុះងារនៅពី Layer មួយទៅ Layer មួយឡើត។
Via មានច្បាស់ប្រភេទ និងអាចធ្វើបានទៅតាមការកំណត់របស់រាងចក្រ។



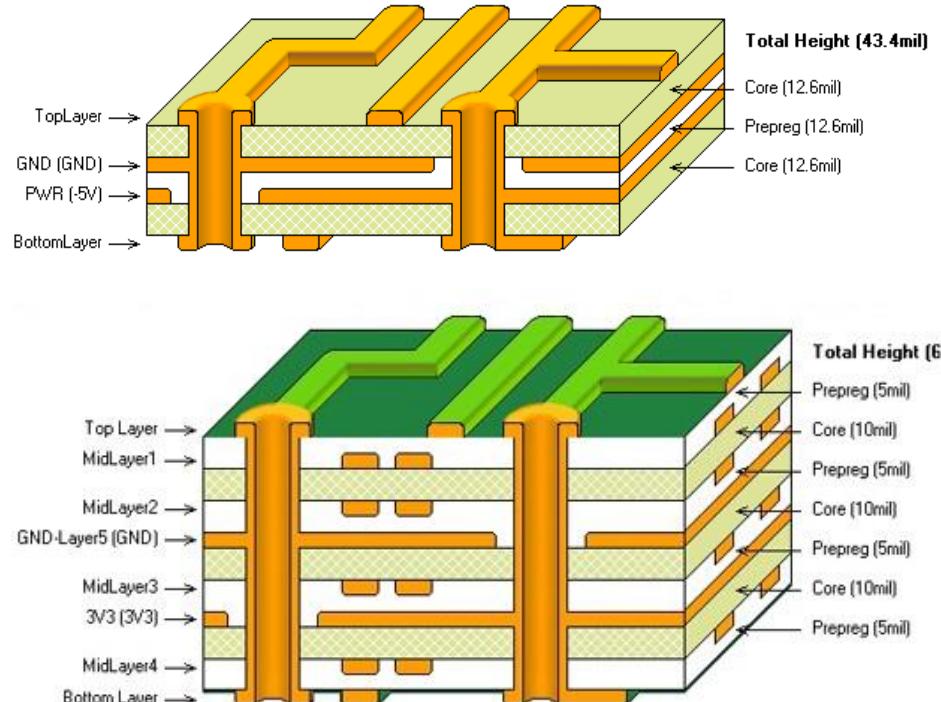


Zone ប្រ Plane ក្នុងសម្រាប់ជាដៃខ្លះថែកចាយ Ground ប្រ Power ដែលអាចថែកចាយចន្ទនាន់។

PCB Design មួយអាជមាន Zone ត្រឹមទៅតាម Design ជាក់ស្តីង។

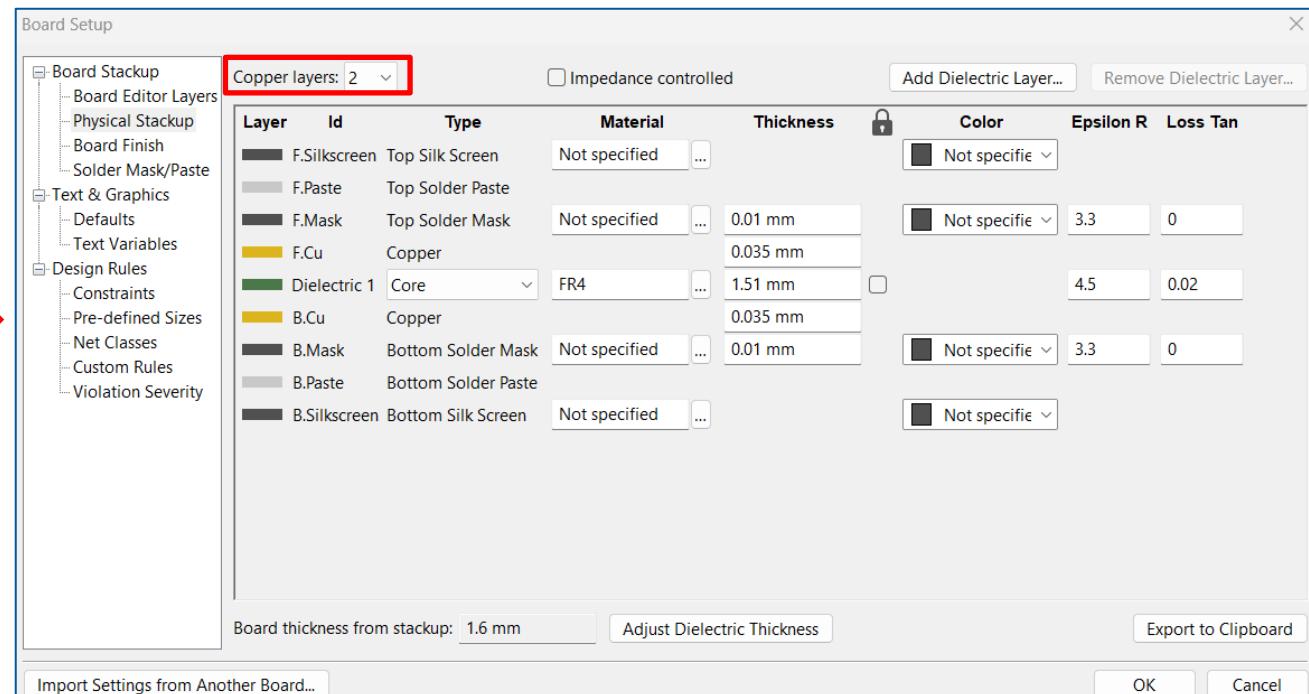
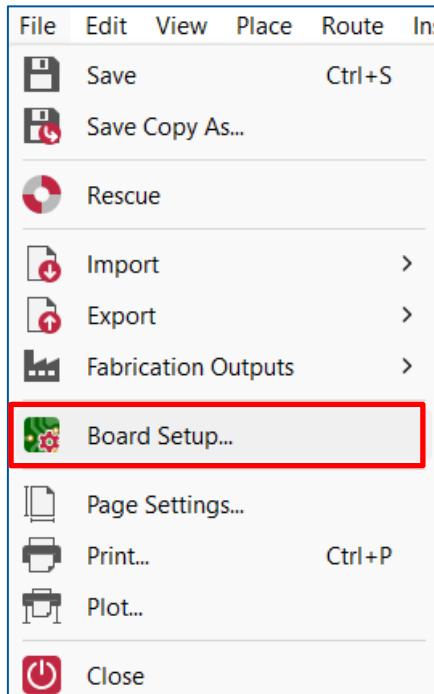


PCB បង្ហាញពីការគ្វាប់គ្វាបស់ PCB ដែលមានប្រើប្រាស់ប្រចាំបែង

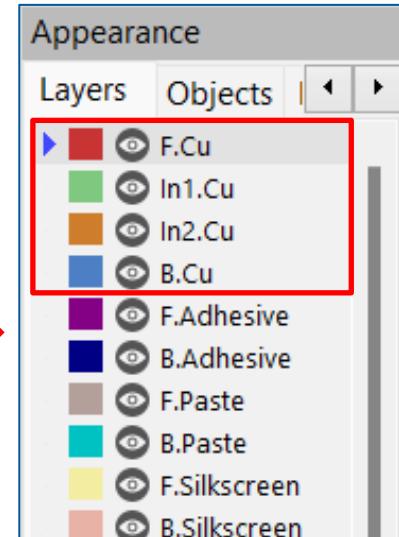
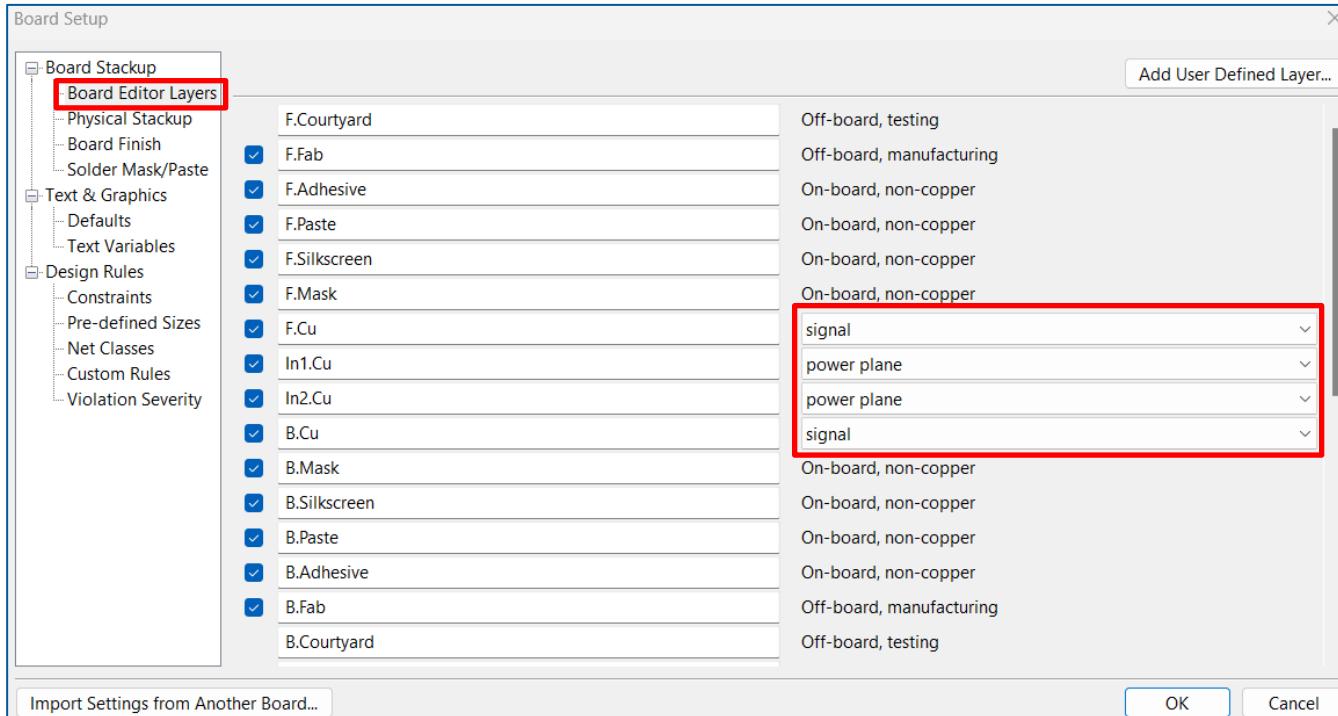


4. PCB Design

4.5.1 កំណត់ស្របតាម (Setup Multiple Layers)

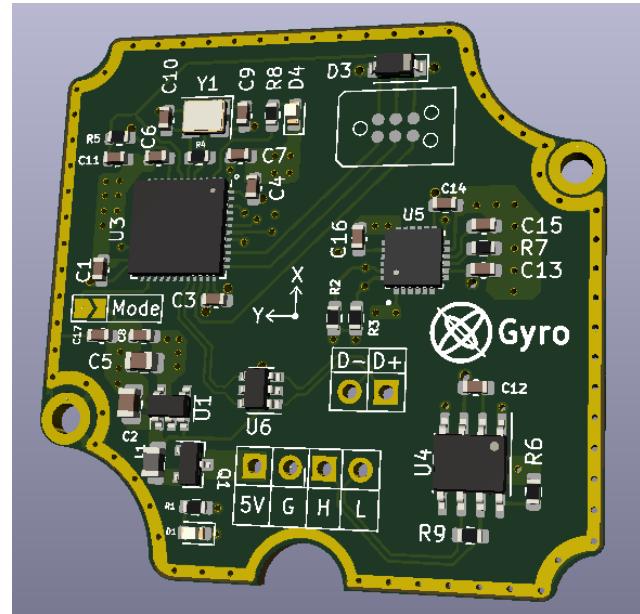
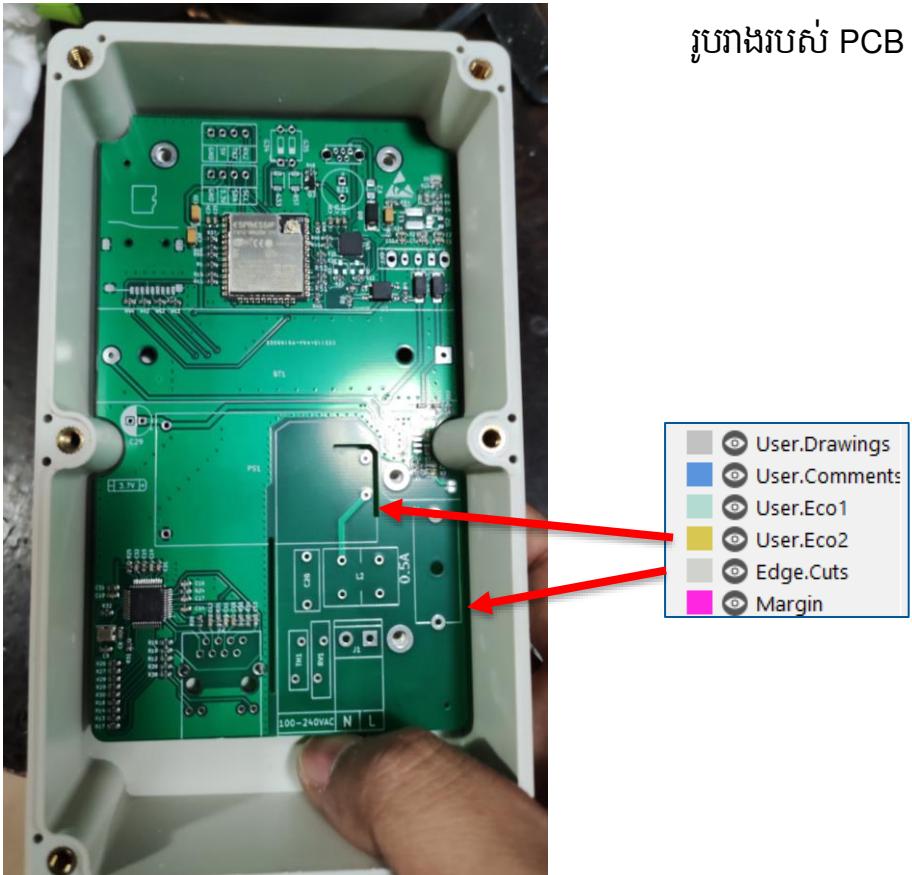


ចូលទៅកាន់ File >> Board Setup >> Board Stackup >> Physical Stackup រួចធ្វើការកំណត់ចំណាំនៃស្របតាម Copper Layers



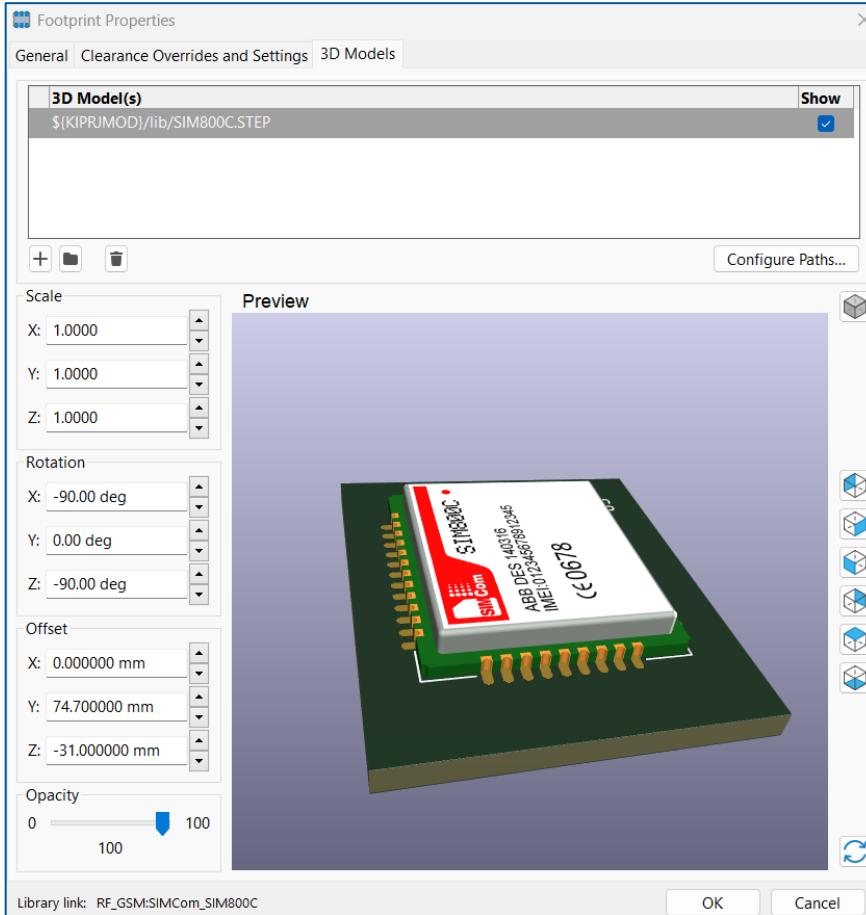
បន្ទាប់មកបញ្ជីកាន់ Board Editor Layers គួរពីការកំណត់ប្រភេទស្រសាប់ (Define Layer)

រូបរាងរបស់ PCB ត្រូវបានកំណត់ទៅតាមប្រអប់ដែលត្រូវដាក់ ទំហំដែលបានកំណត់។



4. PCB Design

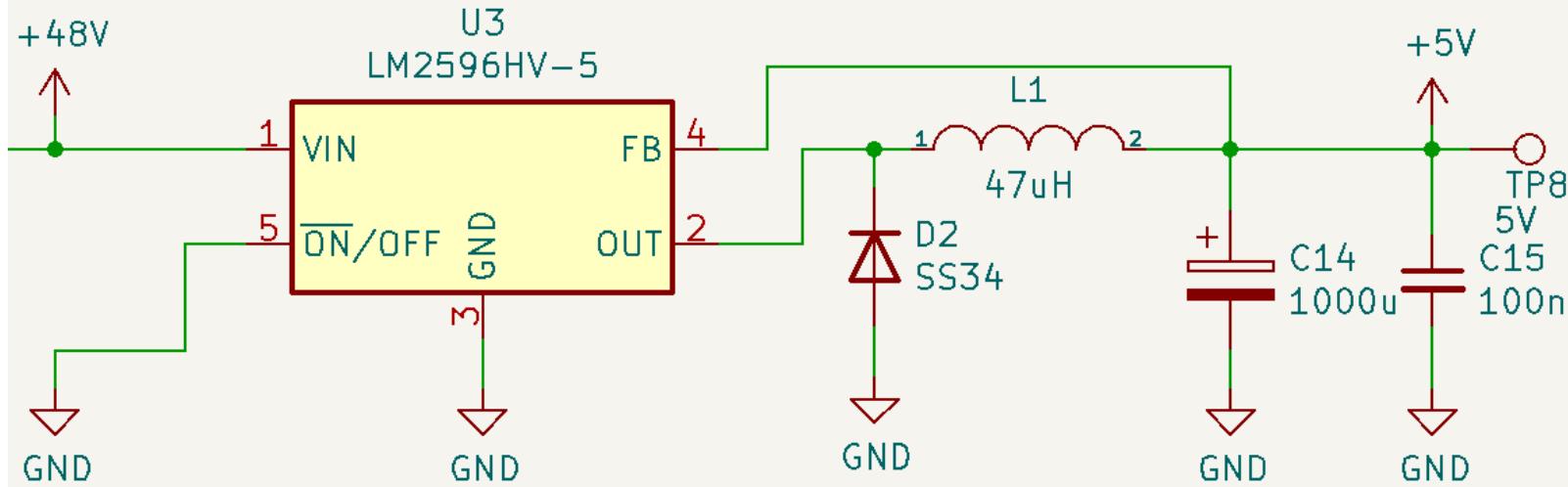
4.7 កំណត់ 3D View



ធើម្នូនាយស្ថុល ប្រើប្រាស់ File STEP
ធើម្នូបញ្ចប់ទៅកាន់ 3D

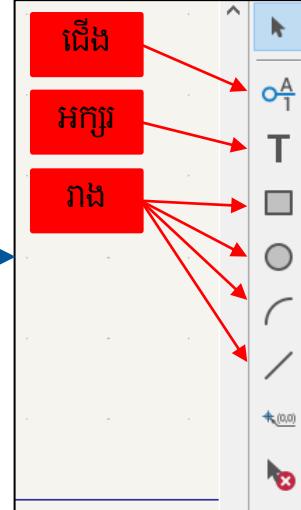
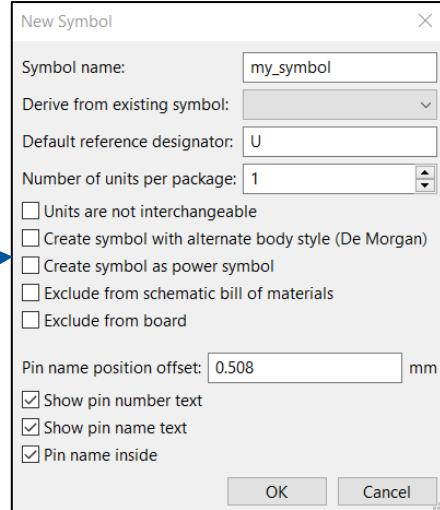
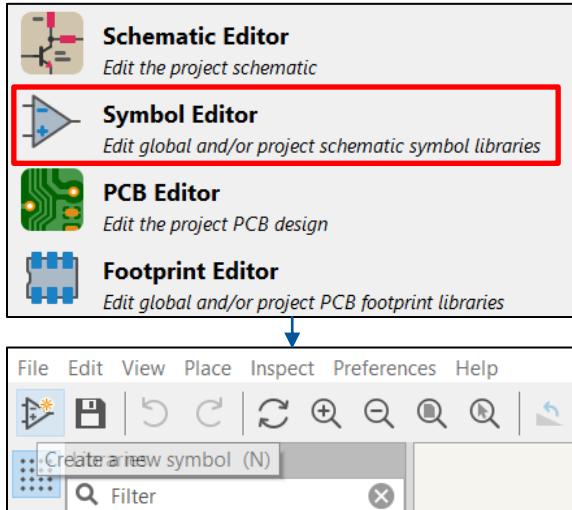


Practice + Design Review



5. Custom Library

5.1 Create Symbol



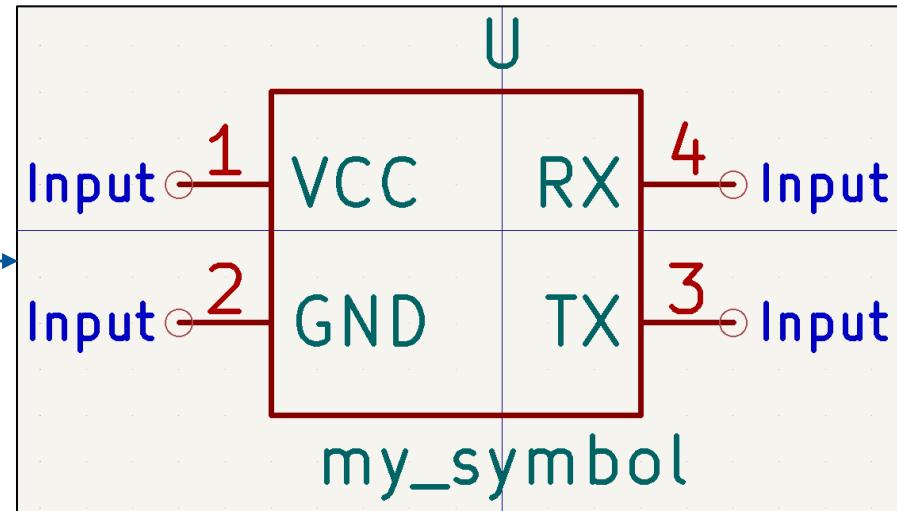
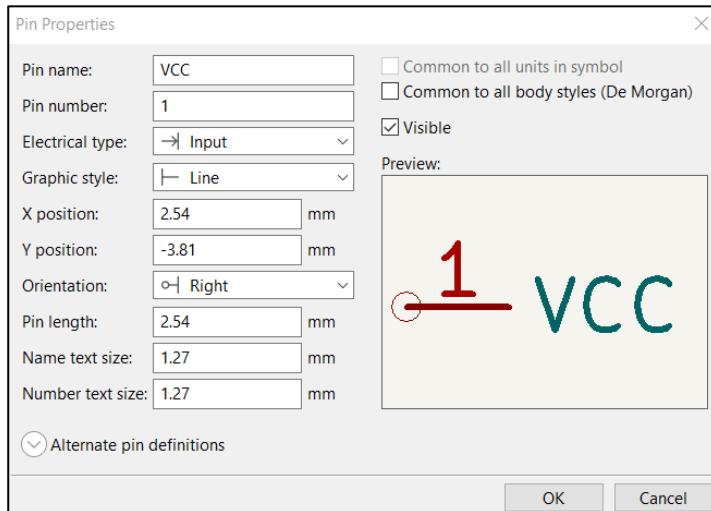
- នៅផ្ទាំង Home ធ្វើសម្រួលយក
Symbol Editor
- ធ្វើសម្រួល Group ណាមួយ
និង Create new symbol (N)

- ជាក់ឡើង Symbol ត្រចប់ច
OK

- ចូចលើខបករណ៍នៅខាង
ស្តាំដើម្បីបង្កើតជាយុបាង និង
ធ្វើសម្រួល Symbol

5. Custom Library

5.1 Create Symbol (Continue)

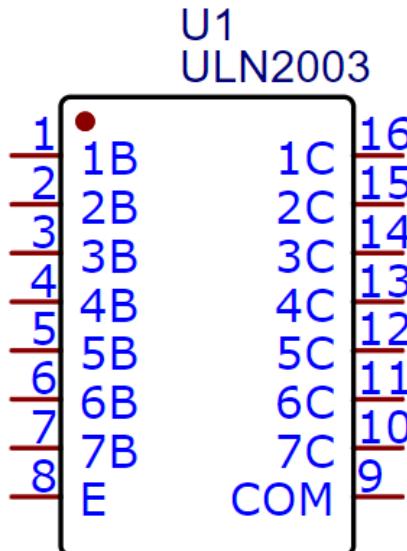


5. ជាក់លេខ: ដើរ និងលេខរបស់
ដើរ

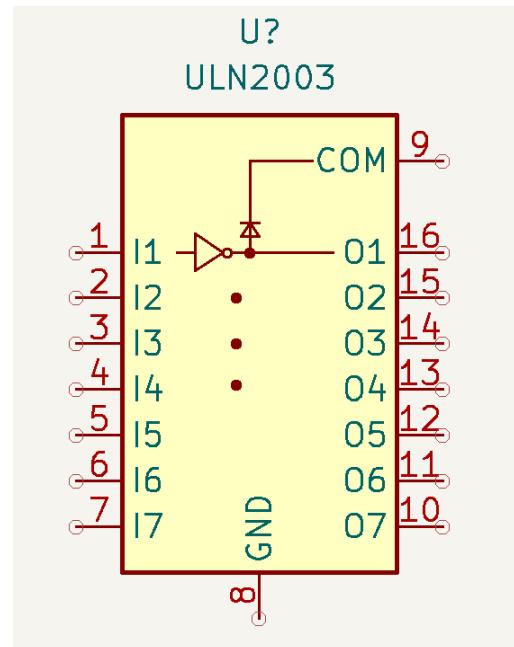
6. បង្កើតរបស់ Symbol
7. Export Symbol ដើម្បីទូកប្រើប្រាស់

5. Custom Library

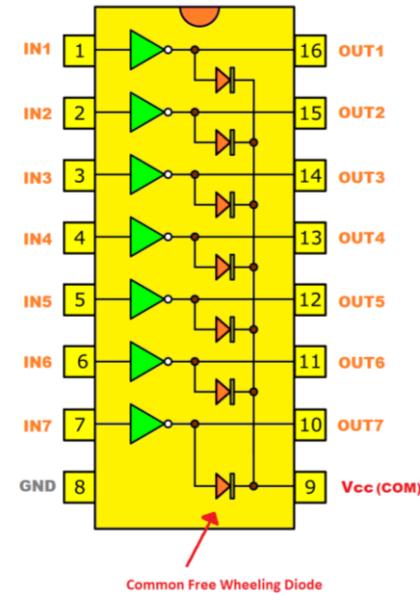
5.1 Create Symbol (Continue)



Symbol မြင်လွှာ

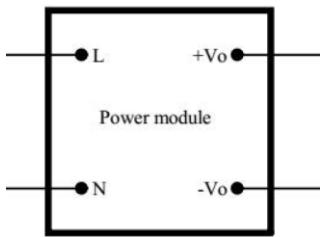


Symbol ပြု

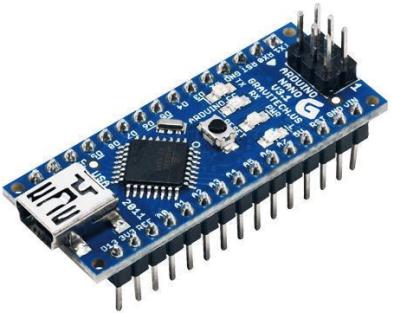


Symbol တော်ဝယ်

Practice + Review



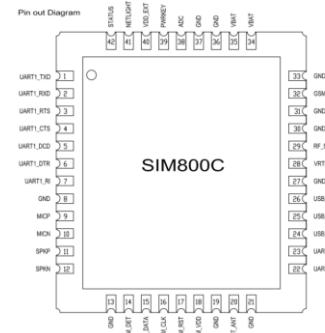
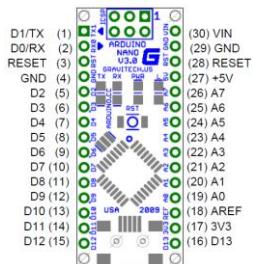
Hi-Link



Arduino V3.0

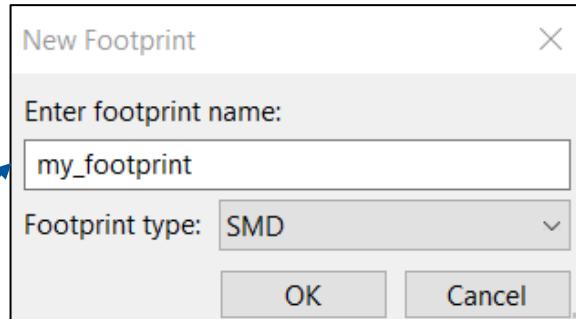
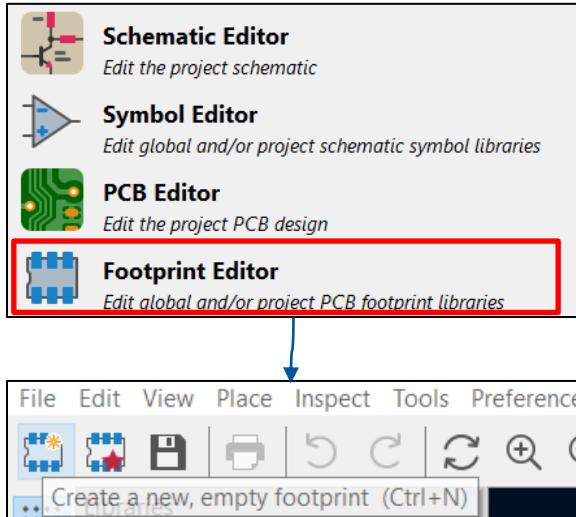


SIM800C



5. Custom Library

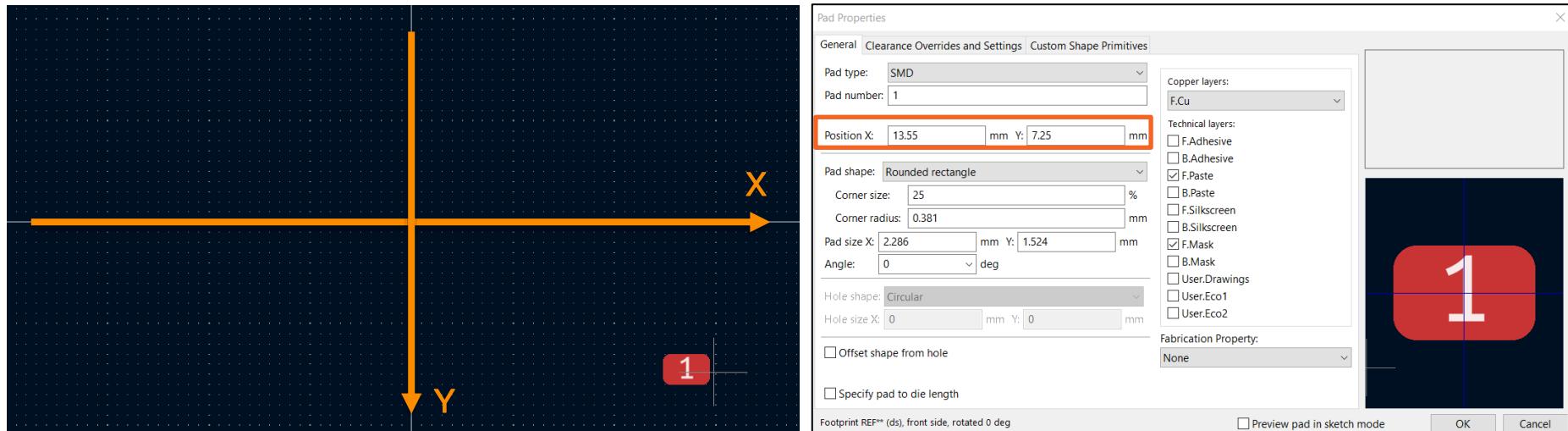
5.2 Create Footprint



- នៅទីតាំងដើម្បីសរើស
Footprint Editor
- ផ្តល់ជូន Group ថ្មី
Create a new footprint

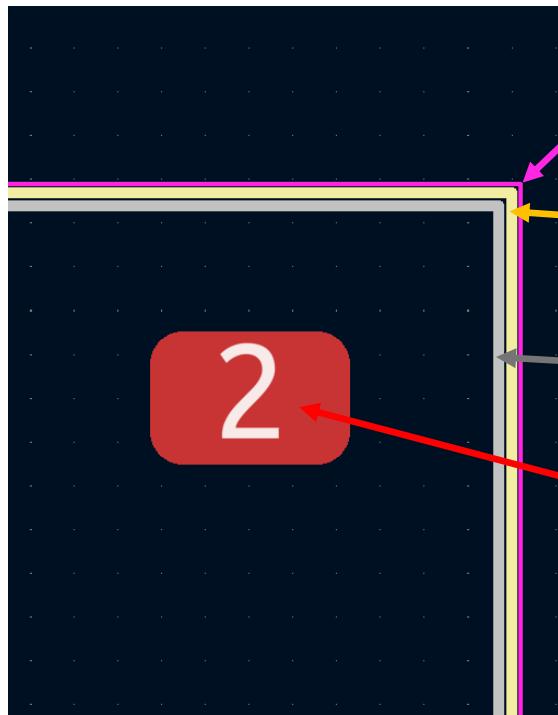
- ជាក់លេខា: Footprint និងផ្តល់ជូន
ផ្តល់ជូន Footprint Type

- បង្កើតរបាយការងារ Footprint ឬ
Export ទូកប្រើប្រាស់



ក្រអែវិធោ សម្រាប់គ្នា

Properties របស់ដើង



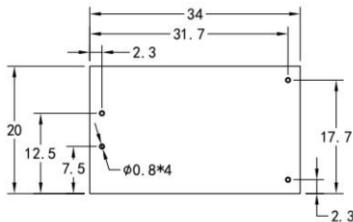
Courtyard : បញ្ជាក់ពីដែនកំណត់នៃទំហំរបស់គ្រឿង (ការពារប្រយោជន៍តាម)

Silkscreen : បង្ហាញនៅលើ PCB ដើម្បីជាតាំងសច្ខាតំណុលប្រភេទ

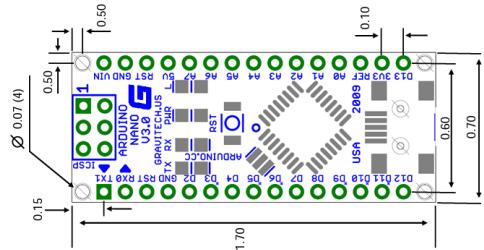
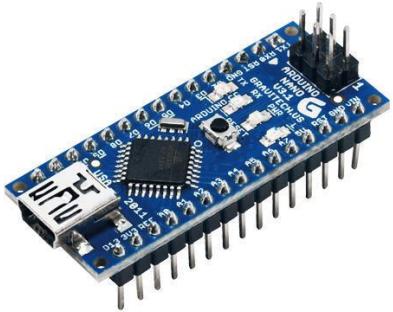
Drawing : បញ្ជាក់ពីទំហំពិតរបស់គ្រឿង (មិនរួមទាំងដើម្បីនៅឡើង)

Pad : ផ្ទៃសម្រាប់ផ្សាយដើររបស់គ្រឿង

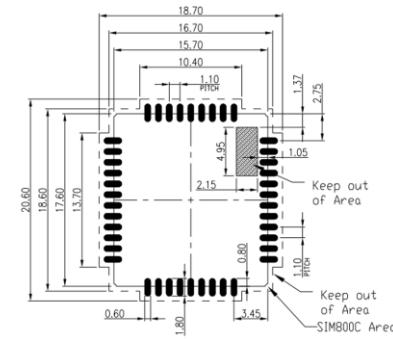
Practice + Review



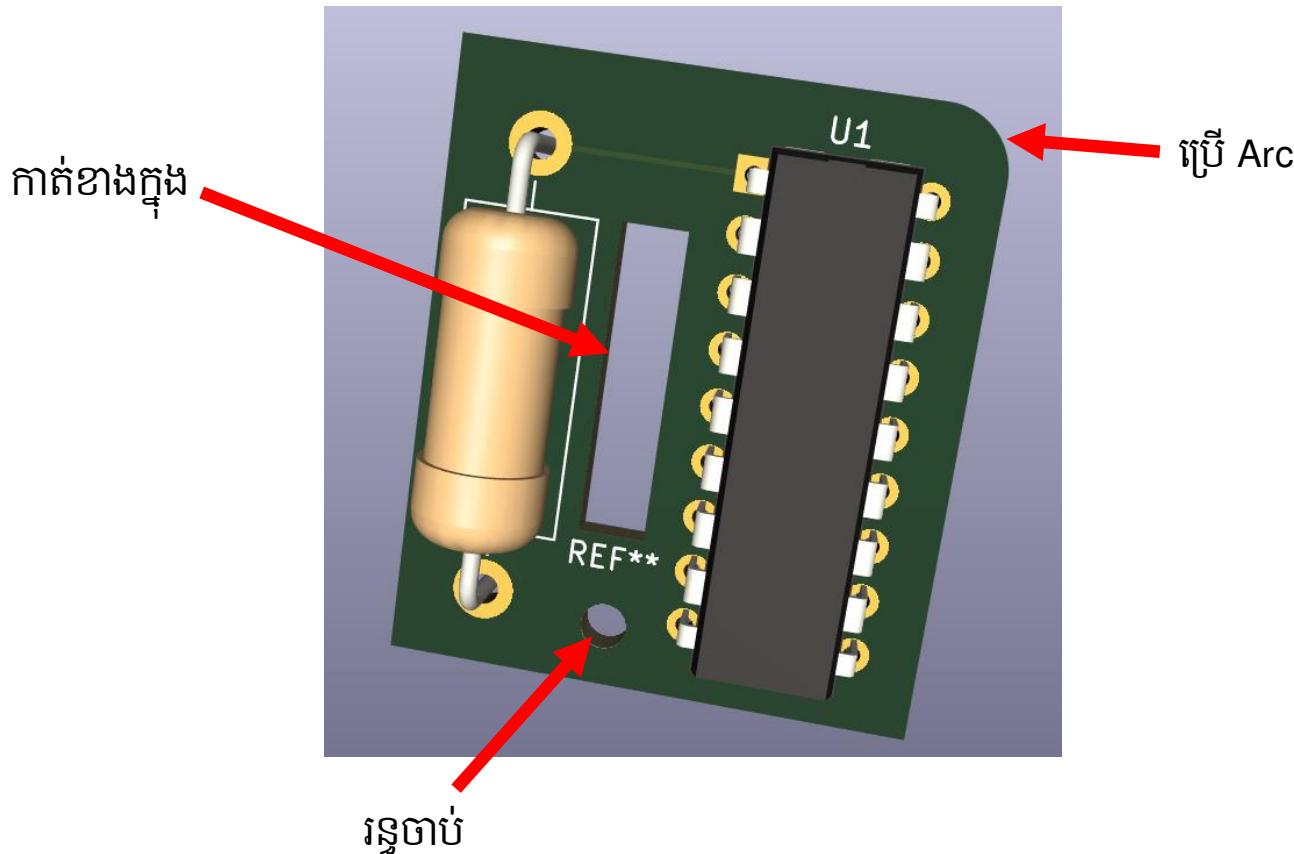
[Hi-Link](#)



[Arduino V3.0](#)



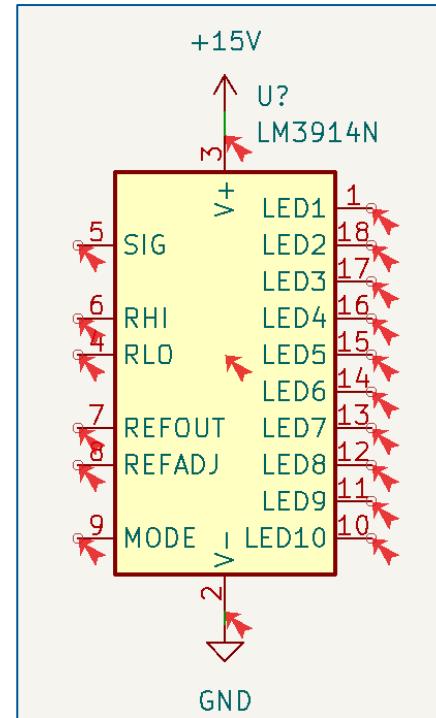
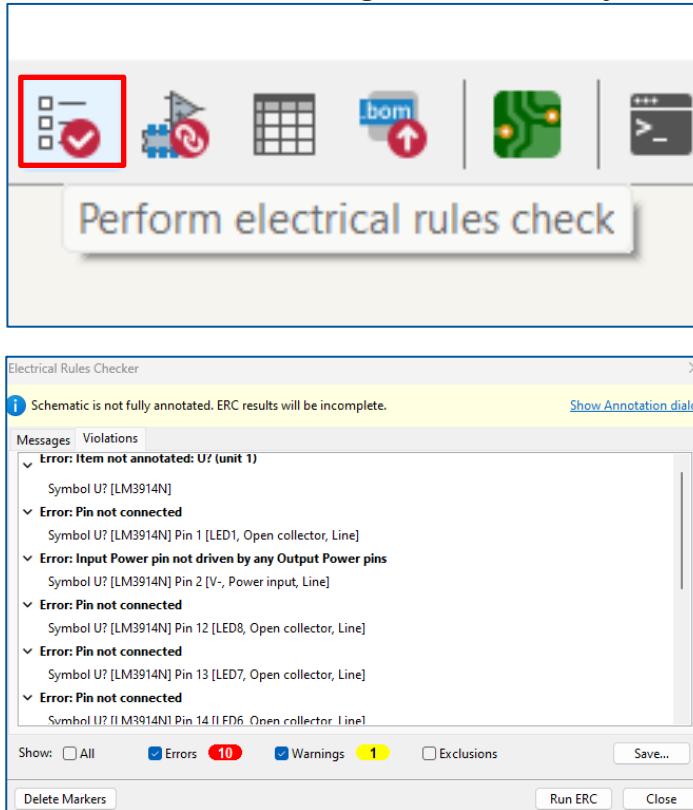
[SIM800C](#)



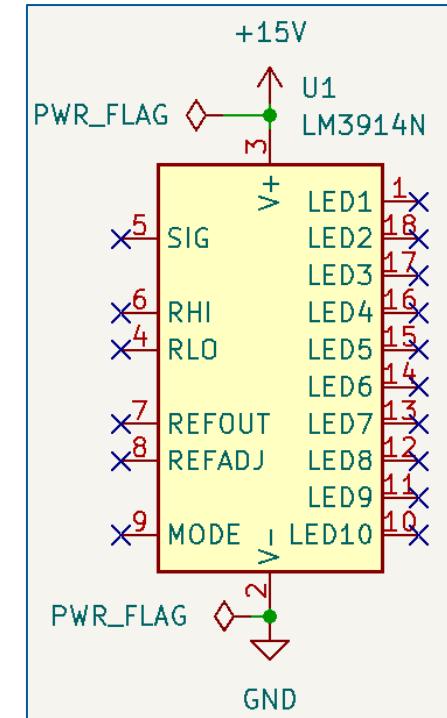
7. Design Rule

7.1 Schematic

Tool សម្រាប់ពិនិត្យមេនូវភាពត្រឹមត្រូវ



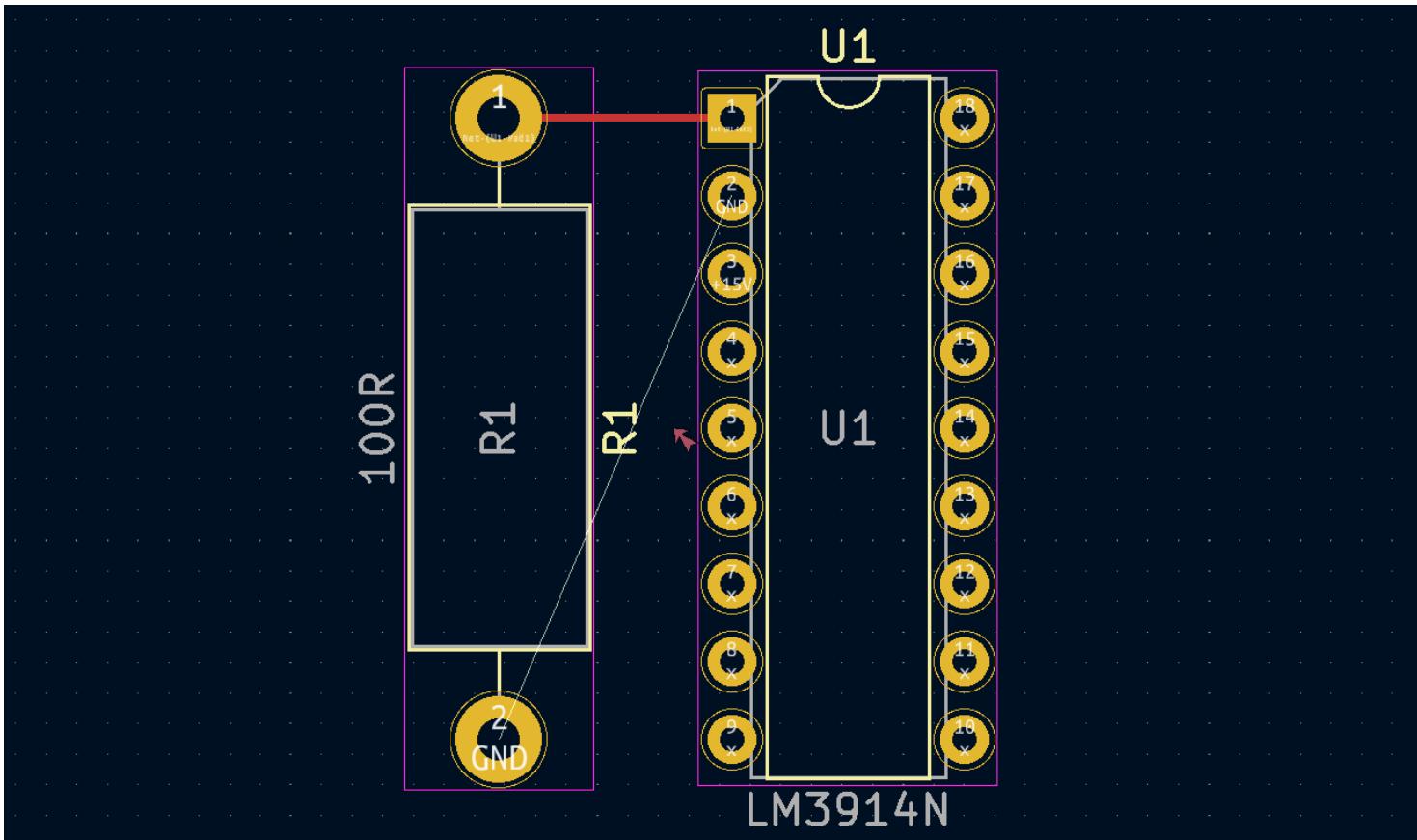
ការខ្សោយត្រូវមិនល្អ



ការកំណត់ត្រឹមត្រូវ

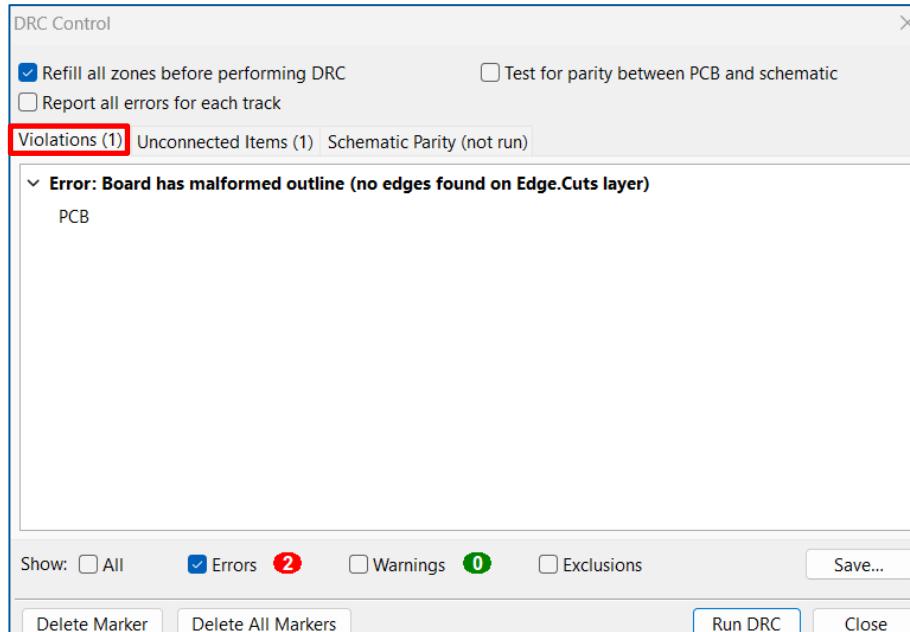
7. Design Rule

7.2 PCB

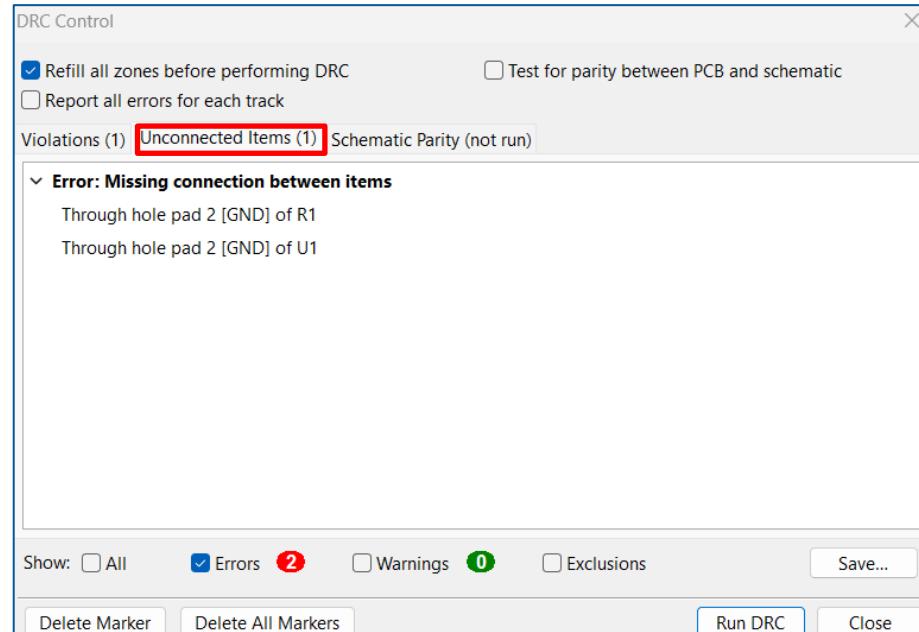


7. Design Rule

7.2 PCB



Violations បញ្ជាក់ពី Error ដែលខុស ប្រឡេ:



Unconnected Items បង្ហាញពីខ្សោយដែលមិនទាន់ភាប់