



เอกสารประกอบการบรรยาย

ดาวน์โหลดได้ที่

<https://github.com/lamloei/present2>

ติดตามได้ที่

<https://www.facebook.com/lamloeicom>



Lamloei ทำเกี่ยวกับอะไร?

PCB/AC Form Factory

เอกสารกำกับโรงงาน PCB PCBA PCBC

- หมวดหมู่ PCB/AC
- สินค้า
 - NodeWIFI
 - Node32s, Node32s Plus
 - Node32Pico
 - Pico01



PCB/AC คืออะไร?

PCB (Printed Circuit Board) - แผ่นลายทองแดง

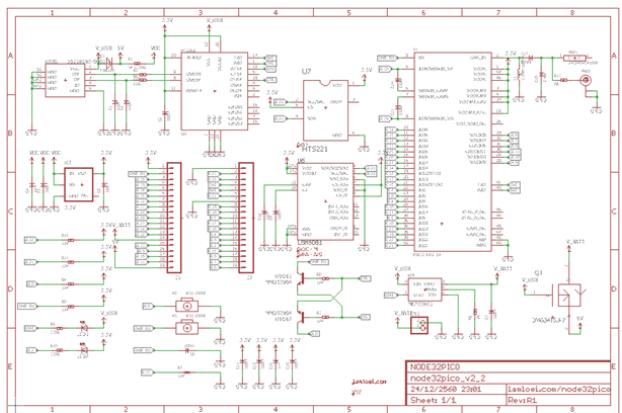
PCBA (Printed Circuit Board Assembly) – แผ่น
ลายทองแดงที่บัดกรีอุปกรณ์แล้ว

PCBC (Printed Circuit Board Casing) - กล่องใส่อุปกรณ์

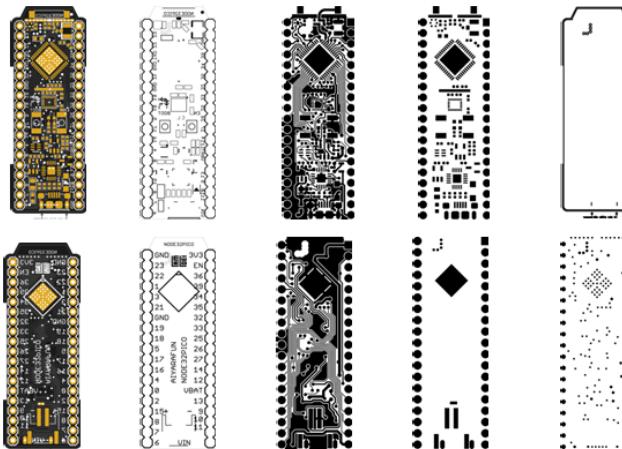


เอกสารกำกับโรงงาน มีอะไรบ้าง?

Schematic Gerber BOM OTHER



Schematic



Gerber

Node32Pico			v2.2	24/12/2017			
Item	Quantity	Reference	Part	Footprint	Mfg	Mfg P/N	Vendor
1	1	ANT1	ANTENNA CHIP UWB 2.3 - 2.7 GHZ		Johanson	2500AT44M0400E	Mouser
2	1	ANT2	U.FL				
3	2	C1, C2	CAP CER 47PF 50V COG 2%	0402	Murata	GRM155C1H470GA01D	Mouser
4	9	C3, C5, C6, C7, C8, C9, C11, C12, C16	CAP CER 0.1UF 16V X7R	0402	Murata	GRM155R71C104KA88D	Mouser

BOM

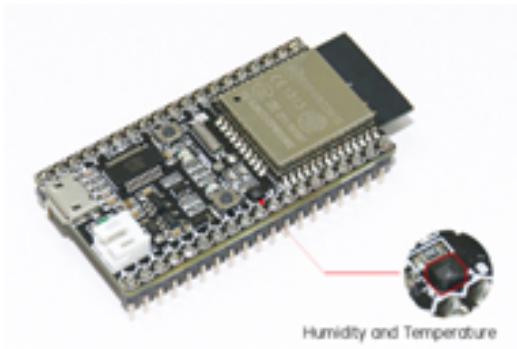
OTHER



Products



Node32Pico



Node32s Plus



Node32s

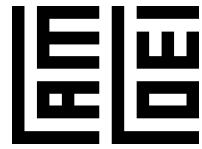


NodeWiFi

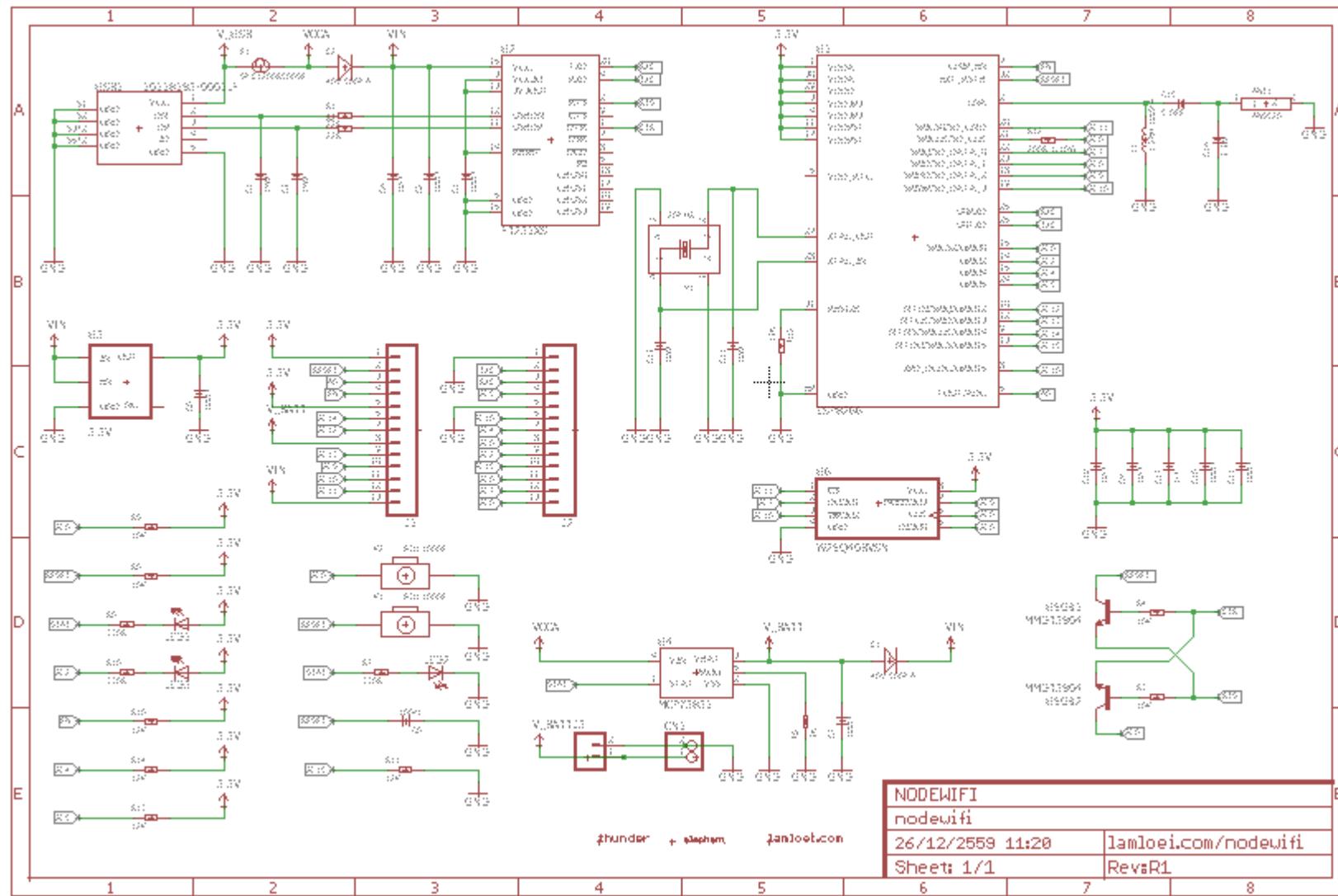


ตามสไตล์ Maker คืออะไร?

ผู้ประดิษฐ์ผลิตภัณฑ์ ตอบโจทย์ตัวเอง

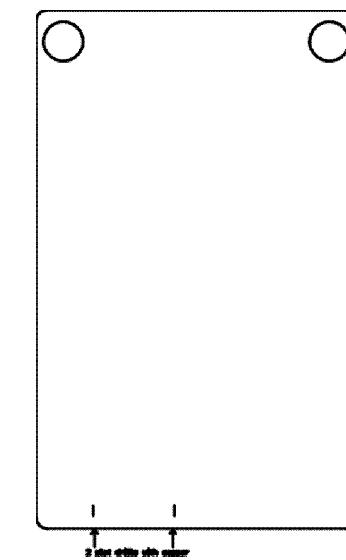
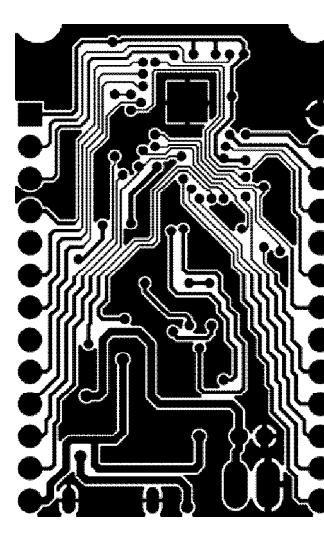
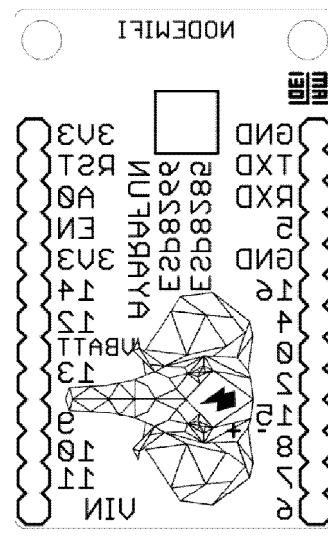
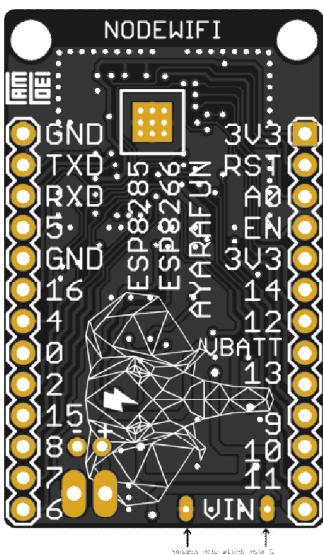
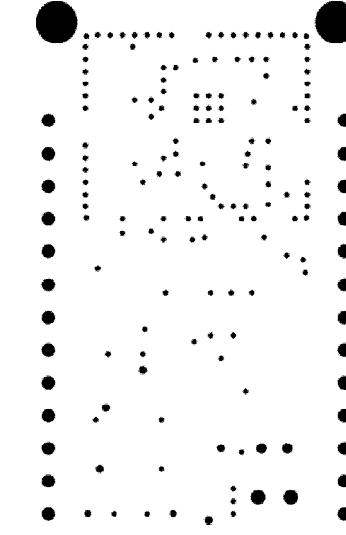
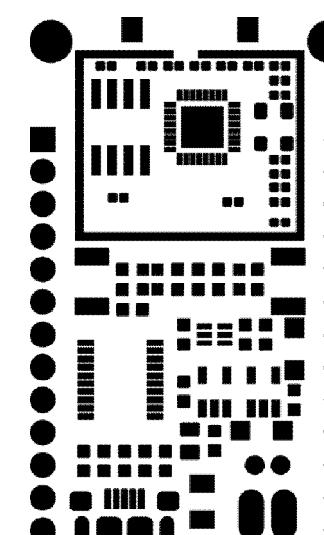
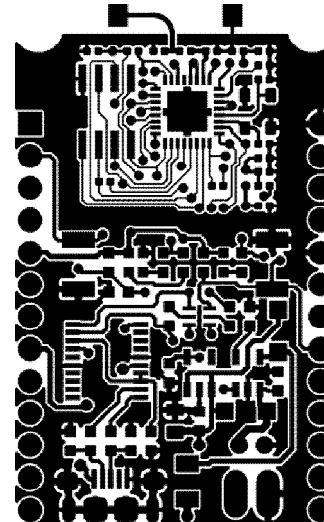
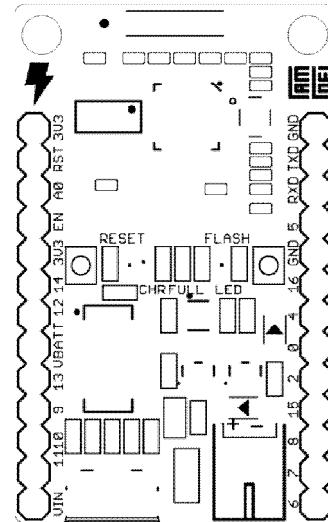
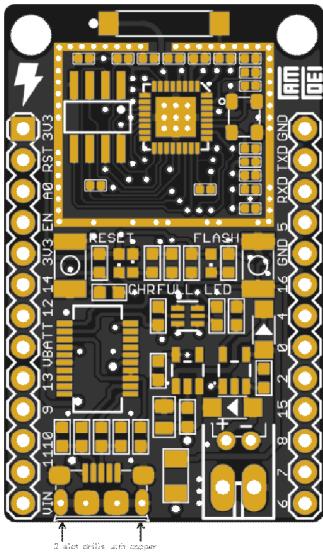


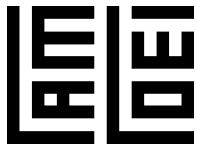
NodeWiFi Schematic





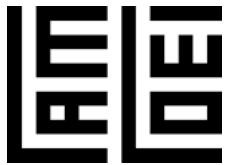
NodeWiFi Gerber





NodeWiFi BOM

		Node WiFi		v1.0	23/1/2017					
Item	Quantity	Reference	Part	Footprint	Mfg	Mfg P/N	Vendor	Vendor P/N	Gtech P/N	Notes
1	1	ANT1	ANTENNA CHIP UWB 2.3 - 2.7 GHZ		Johanson	2500AT44M0400E	Mouser	609-2500AT44M0400E	PL01	
2	2	C1, C2	CAP CER 47PF 50V NPO 5%	0603	Samsung	CL10C470JB8NNNC	Digikey	1276-1037-1-ND	PT022A	
3	5	C3, C5, C6, C7, C8	CAP CER 0.1UF +/-10% 50V X7R	0603	Samsung	CL10B104KB8NNNC	Digikey	1276-1000-1-ND	C542B	
4	1	C4	CAP CER 10UF 10V X5R	0805	Murata	GRM21BR61A106KE19L	Mouser	81-GRM21BR61A106KE19	ND02	
5	2	C9, C10	CAP CER 10UF 6.3V X6S	0402	Murata	GRM155C80J106ME1D	Mouser	81-GRM155C80J106ME1D		
6	2	C11, C12	CAP CER 9PF 50V NPO	0402	Murata	GRM155C1H9R0DA01D	Mouser	81-GRM1885C1H9R0CA1D	PL05	
7	2	C13, C16	CAP CER 0.1UF 16V X7R	0402	Murata	GRM155R71C104KA88D	Mouser	81-GRM155R71C104KA88	PL04	
8	1	C14	CAP CER 2.4PF 50V NPO	0402	Murata	GRM155C1H2R4BA01D	Mouser	81-GRM155C1H2R4BA1D	PT088	
9	1	C15	CAP CER 6.8PF 50V NPO	0402	Murata	GRM155C1H6R8CA01D	Mouser	81-GRM155C1H6R8CA1D	PT087	
10	1	C17	CAP CER 1UF 25V X5R	0402	Murata	GRM155R61E105KA12D	Mouser	81-GRM155R61E105KA2D	PL03	
11	1	CN1	2mm PTH connector							Not installed
12	2	D1, D2	DIODE SCHOTTKY 40V 500mA	SOD-123	Taiwan Semi	B0540W	Mouser	821-B0540W	ND05	
13	1	F1	PTC RESTTBLLE 0.25A 24V CHIP	1206	Bel Fuse	OZCJ0025AF2E	Mouser	530-0ZCJ0025AF2E	ND06	
14	2	J1, J2	13-PIN Male Header	0.1"						
15	1	J3	JST 2.0mm Battery Connector							
16	2	K1, K2	Switch, Momentary Tact SPST 160gf 3.9x2.9x2.0mm	SMD	ALPS	SKRKAEE010	Mouser	688-SKRKAE	PT012	
17	1	L1	FIXED IND 1.6NH 1A 120 MOHM	0402	TDK	MLG1005S1N6BTD25	Mouser	810-MLG1005S1N6BTD25	PT095	
18	1	LED1	LED RED 2.0V 622nm 110mcd	0603		WB-0603QRC	China		C200A	
19	1	LED2	LED GREEN 2.0V 572nm 40mcd	0603		WB-0603QGC	China		C202C	
20	1	LED3	LED YELLOW 2.0V 589nm 145mcd	0603		WB-0603QYC	China		C201A	
21	2	R1, R2	RES 22 OHM 1% 1/10W	0603	Yageo	AC0603FR-0722RL	Mouser	603-AC0603FR-0722RL	PT020	
22	4	R3, R4, R6, R9	RES 10K OHM 1% 1/10W	0603	Yageo	RC0603FR-0710KL	Mouser	603-RC0603FR-0710KL	PT014	
23	1	R5	RES 3K OHM 1% 1/10W	0603	Yageo	RC0603FR-073KL	Mouser	603-RC0603FR-073KL		
24	3	R7, R8, R10	RES 330 OHM 1% 1/10W	0603	Panasonic	ERJ-3EKF3300V	Mouser	667-ERJ-3EKF3300V	PT015	
25	5	R11, R13, R14, R15, R16	RES 12K OHM 1% 1/16W	0402	Yageo	RC0402FR-0712KL	Mouser	603-RC0402FR-0712KL	PL10	
26	1	R12	RES SMD 0.00OHM JUMPER 1/16W	0402	Yageo	RC0402JR-070RL	Mouser	603-RC0402JR-070RL	PL13	
27	1	U1	ESP8285		Espressif		Espressif			
28	1	U2	IC USB SERIAL FULL UART	20SSOP	FTDI	FT231XS-R	Mouser	895-FT231XS-R	ND08	



Testrun

```
∞ COM29 Send

11: iuvinc1 (-72)*
12: NSTDA-WIFI (-72)
13: eduroam (-73)*
14: NSTDA-GUEST (-74)
15: NSTDA-WIFI (-88)

***CHIP ID***
ESP32 Chip ID = 940E001DA0D8
Chip Revision (official version): 0

***I2C SCANNING***
I2C device found at address 0x1C !
I2C device found at address 0x5F !
I2C device found at address 0x6A !
done

***HALL SENSOR***
sensor = 9

***TIME TEMPERATURE SENSOR***
scan start 00:00:00 Temp onBoard 167°F 75.00°C

***HTS221 SENSOR***
Humidity : 20.00 %
Temperature: 38.60 celsius

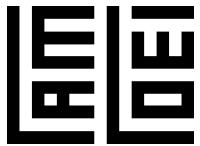
***LSM9DS1 SENSOR***
G: -0.10, -2.93, -0.39 deg/s
A: -0.03, 0.03, 0.98 g
M: 0.61, -0.74, -0.82 gauss
Pitch, Roll: 1.54, 1.55
Heading: 137.85

***WIFI SCAN***
scan done

 Autoscroll No line ending 115200 baud Clear output
```

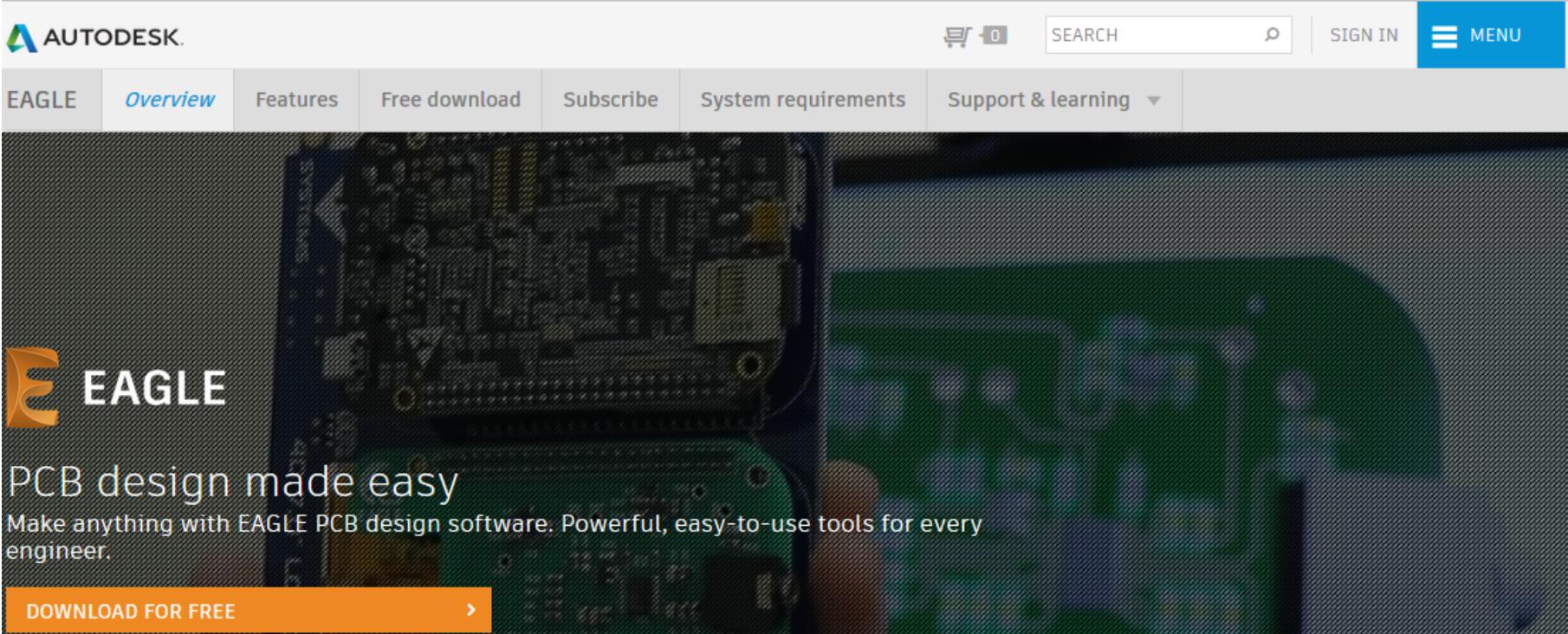


เราจะเขียนเอกสารกำกับน้ำอย่างไร?

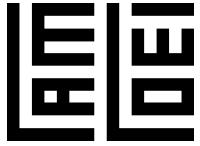


<https://www.autodesk.com/products/eagle/>

overview



The screenshot shows the Autodesk EAGLE product page. At the top, there's a navigation bar with the Autodesk logo, a shopping cart icon (0 items), a search bar, a sign-in link, and a menu icon. Below the navigation bar, there's a horizontal menu with tabs: EAGLE (selected), Overview (highlighted in blue), Features, Free download, Subscribe, System requirements, and Support & learning. The main content area features a large image of a printed circuit board (PCB) with the Autodesk EAGLE logo overlaid. Below the image, the text "PCB design made easy" is displayed, followed by the subtext "Make anything with EAGLE PCB design software. Powerful, easy-to-use tools for every engineer." At the bottom left, there's a prominent orange button with the text "DOWNLOAD FOR FREE" and a right-pointing arrow.



Download

Present Eagle

- <https://github.com/lamloei/present2>

Gerber bv

- <https://sourceforge.net/projects/gerbv/files/>

Eagle 7

- <ftp://ftp.cadsoft.de/eagle/program/7.7/>

หรือ Eagle 9

<https://www.autodesk.com/products/eagle/free-download>



ໄດອະແກຣມ

sch



bom

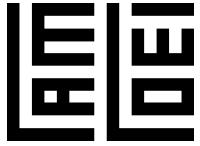
brd



gerber



ทำไมถึงเลือกใช้ Eagle?



PCB FAB

<https://oshpark.com/>

<https://pcbs.io/>

https://www.seeedstudio.com/fusion_pcb.html

<https://www.elecrow.com/pcb-prototyping.html>

<https://www.pcbway.com/>

<http://www.seegate15.com>

<https://www.facebook.com/LayerCircuit>

[http://www.allpcb.com/setinvite.aspx?inviteid=27397
&url=https://www.allpcb.com/online_quote.html](http://www.allpcb.com/setinvite.aspx?inviteid=27397&url=https://www.allpcb.com/online_quote.html)



PCB Software

<https://www.autodesk.com/products/eagle>

<http://www.altium.com/>

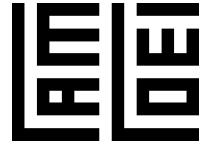
<http://kicad-pcb.org/>

<https://circuits.io/pcb>

<https://easyeda.com/>

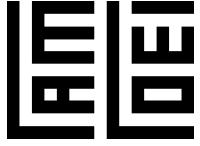
<https://www.rs-online.com/designspark/pcb-software>

และโปรแกรมอื่นๆ



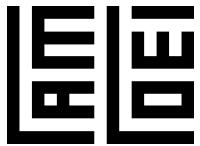
Vendor BOM

- Mouser
- Digikey
- RS
- ES



โปรแกรมที่ต้องใช้

1. Eagle
2. Gerbv
3. Notepad
4. Excel



Subscribe

Subscribe

EAGLE Standard

EAGLE Premium

EAGLE Standard

Includes 99 schematic sheets, 4 signal layers, and 160 cm² board area.

- Access to the latest software releases
- Get 1-on-1 online support ([see all subscriber benefits](#))
- Available for Windows, Mac, and Linux ([see system requirements](#))

[SEE SUBSCRIBER BENEFITS](#)

<input checked="" type="radio"/>	Monthly	\$15
----------------------------------	---------	------

<input type="radio"/>	1 year	\$100
-----------------------	--------	-------

<input type="radio"/>	2 year	\$200
-----------------------	--------	-------

<input type="radio"/>	3 year	\$300
-----------------------	--------	-------

Subscribe

EAGLE Standard

EAGLE Premium

EAGLE Premium

Includes 999 schematic sheets, 16 signal layers, and unlimited board areas.

- Access to the latest software releases
- Get 1-on-1 online support ([see all subscriber benefits](#))
- Available for Windows, Mac, and Linux ([see system requirements](#))

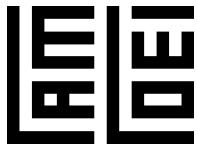
[SEE SUBSCRIBER BENEFITS](#)

<input checked="" type="radio"/>	Monthly	\$65
----------------------------------	---------	------

<input type="radio"/>	1 year	\$500
-----------------------	--------	-------

<input type="radio"/>	2 year	\$1,000
-----------------------	--------	---------

<input type="radio"/>	3 year	\$1,500
-----------------------	--------	---------



ftp://ftp.cadsoft.de/eagle/program/7.7/

Index of ftp://ftp.cadsoft.de/eagle/program/7.7/

[Up to higher level directory](#)

<u>Name</u>	<u>Size</u>	<u>Last Modified</u>
eagle-lin32-7.7.0.run	50282 KB	10/4/16 12:00:00 AM
eagle-lin32-7.7.0.run.INF	1 KB	10/4/16 12:00:00 AM
eagle-lin32.run		10/5/16 12:00:00 AM
eagle-lin64-7.7.0.run	49911 KB	10/4/16 12:00:00 AM
eagle-lin64-7.7.0.run.INF	1 KB	10/4/16 12:00:00 AM
eagle-lin64.run		10/5/16 12:00:00 AM
eagle-mac64-7.7.0.zip	54173 KB	10/4/16 12:00:00 AM
eagle-mac64-7.7.0.zip.INF	1 KB	10/4/16 12:00:00 AM
eagle-mac64.zip		10/5/16 12:00:00 AM
eagle-win32-7.7.0.exe	55208 KB	10/4/16 12:00:00 AM
eagle-win32-7.7.0.exe.INF	1 KB	10/4/16 12:00:00 AM
eagle-win32.exe		10/5/16 12:00:00 AM
eagle-win64-7.7.0.exe	57760 KB	10/4/16 12:00:00 AM
eagle-win64-7.7.0.exe.INF	1 KB	10/4/16 12:00:00 AM
eagle-win64.exe		10/5/16 12:00:00 AM
elektro-tutorial.pdf	4817 KB	5/8/09 12:00:00 AM



Eagle Document

EAGLE 7.7.0 Setup

EAGLE License

If you have received a license file, please select "Use license file" below.
Otherwise you can select "EAGLE Express" to run EAGLE as a free version.

To do the licensing when you start EAGLE for the first time, select "Don't license now".

Use license file
 EAGLE Express
 Don't license now

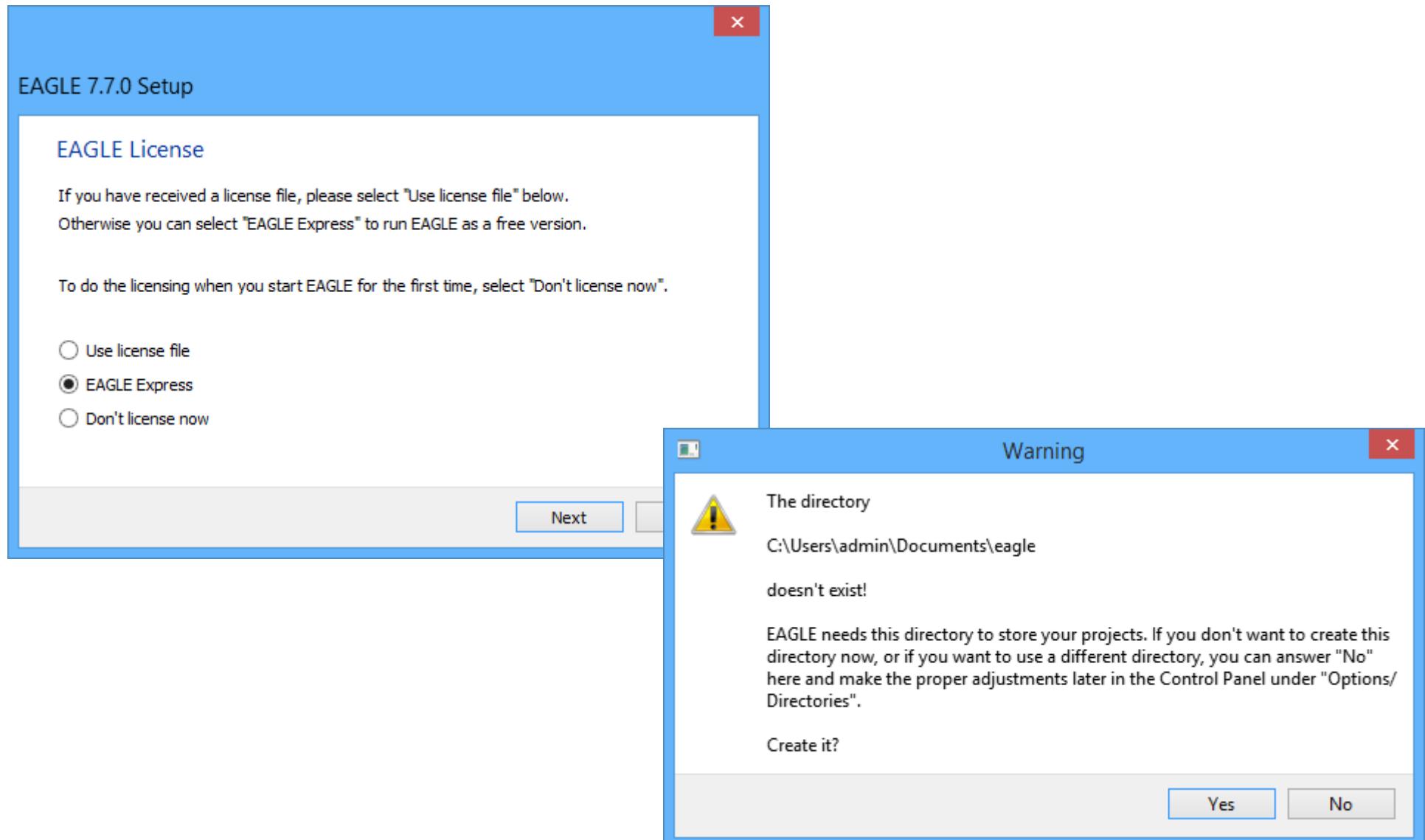
Next

The directory
C:\Users\admin\Documents\eagle
doesn't exist!

EAGLE needs this directory to store your projects. If you don't want to create this directory now, or if you want to use a different directory, you can answer "No" here and make the proper adjustments later in the Control Panel under "Options/Directories".

Create it?

Yes No



<https://sourceforge.net/projects/gerbv/files/>

[Home](#) / [Browse](#) / [Science & Engineering](#) / [Electronic Design Automation \(EDA\)](#) / [gerbv — a Gerber \(RS-274X\) viewer](#) / [Files](#)

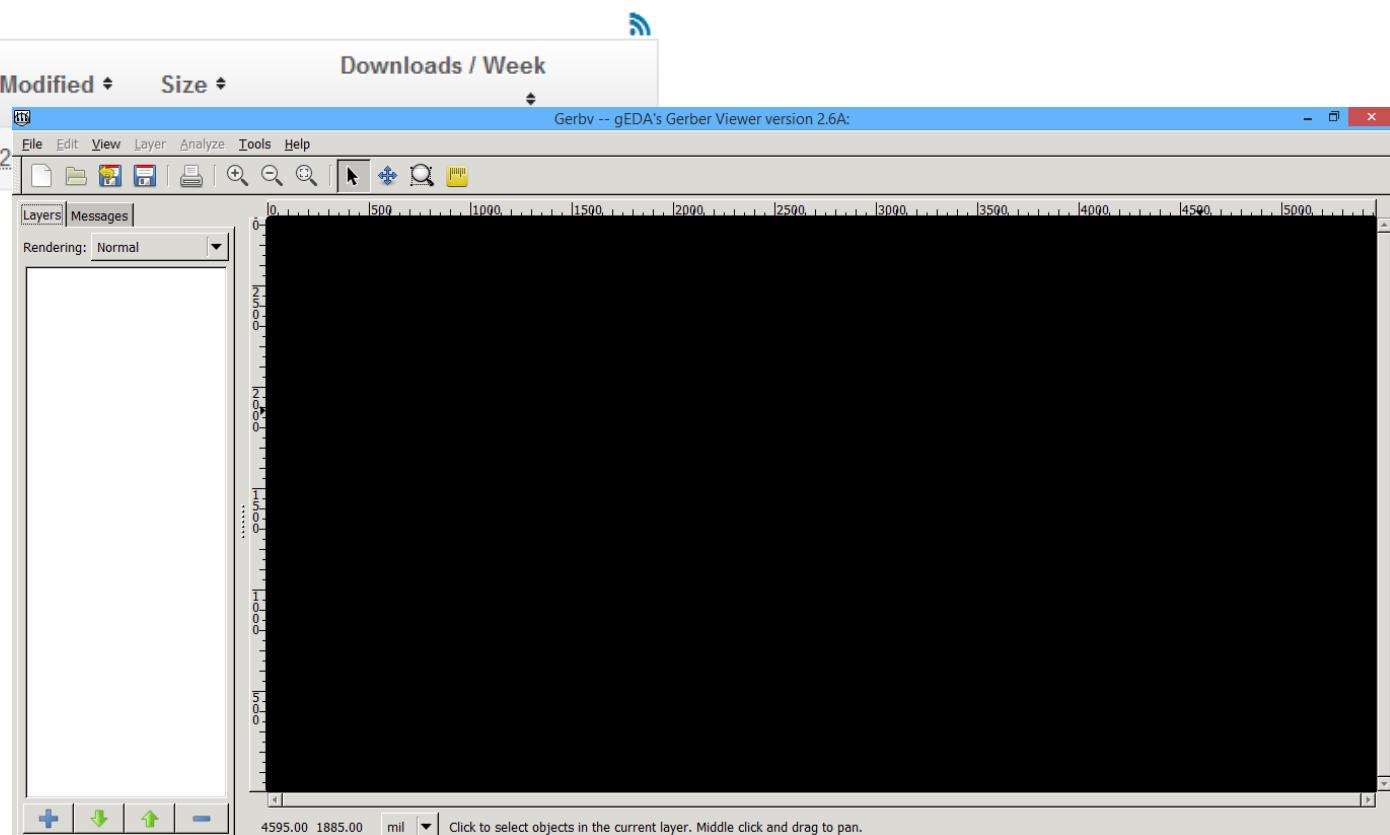
gerbv — a Gerber (RS-274X) viewer

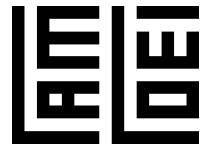
Brought to you by: [ahvezda](#), [asvl](#), [danmc](#), [spetm](#), [thepurlieu](#)

[Summary](#) | **Files** | [Reviews](#) | [Support](#) | [Wiki](#) | [Mailing Lists](#) | [Tickets](#) | [News](#) | [Git](#)

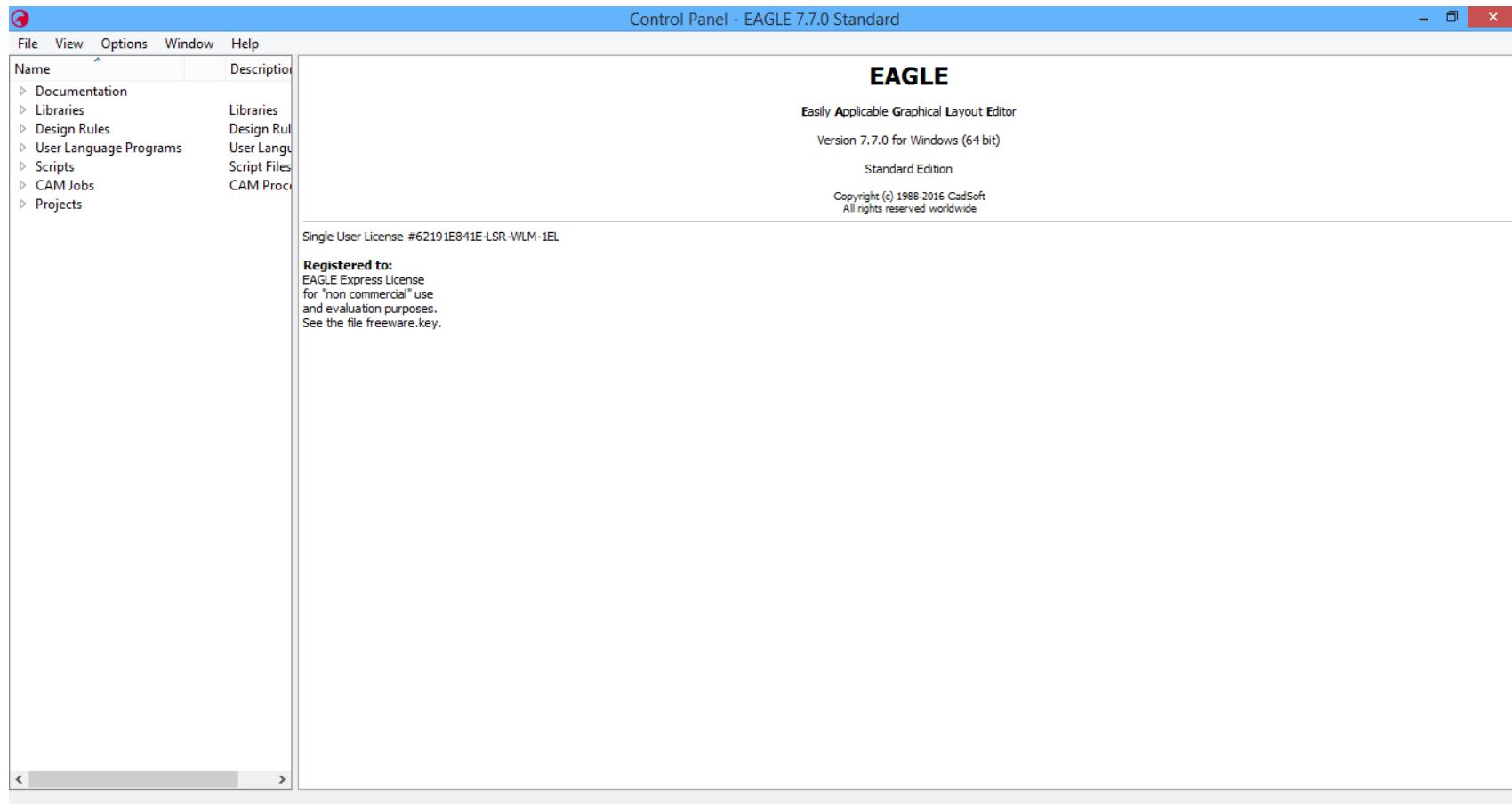
Looking for the latest version? [Download gerbv-win-static_20160713.zip \(4.8 MB\)](#)

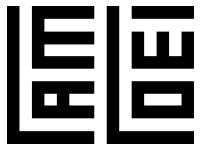
Home
Name
gerbv
Modified
Size
Downloads / Week
Totals: 1 Item



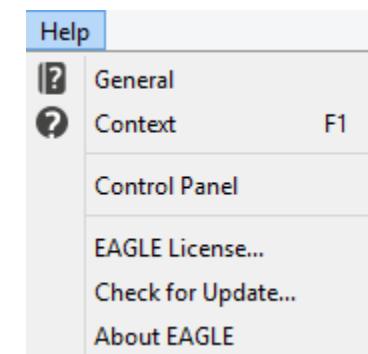
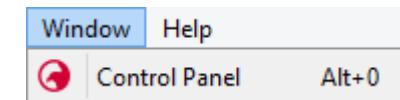
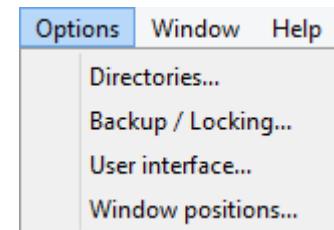
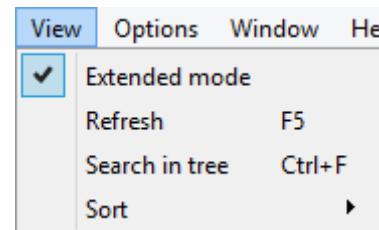
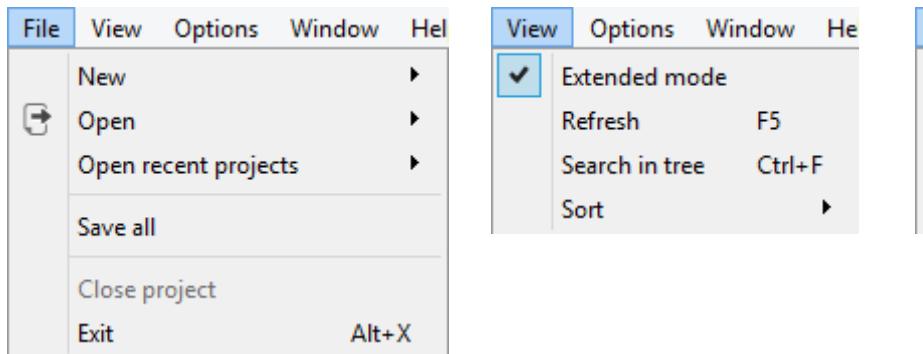


Eagle Control Panel

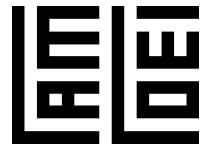




Control Panel Menu



Name	Description
Documentation	
Libraries	Libraries
Design Rules	Design Rules
User Language Programs	User Language Programs
Scripts	Script Files
CAM Jobs	CAM Processor Jobs
Projects	
eagle	
examples	Examples Folder
arduino	Arduino Mega 2560 Reference Board
elektro	Examples Folder for Electrical Schematics
hexapod	Hexapod Example Project
ltspice	Examples of imported schematics from LTspice.
seeed	Various projects from Seeed
singlesided	Example Project for Singlesided Autorouting
ti-launchpad	MSP430F5529 LaunchPad Development Kit
tutorial	Example Files for the Tutorial Projects



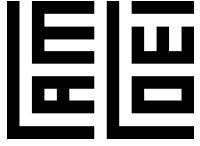
Eagle Folder

A screenshot of a Windows File Explorer window. The address bar shows the path: This PC > Local Disk (C:) > EAGLE-7.7.0. The main area displays a list of folders with the following details:

Name	Date modified	Type
bin	29/6/2560 10:06	File folder
cam	29/6/2560 10:05	File folder
doc	29/6/2560 10:05	File folder
dru	29/6/2560 10:05	File folder
lbr	29/6/2560 10:05	File folder
misc	29/6/2560 10:05	File folder
projects	29/6/2560 10:05	File folder
scr	29/6/2560 10:05	File folder
ulp	29/6/2560 10:05	File folder

C:\EAGLE-7.7.0

C:\Users\admin\Documents\eagle

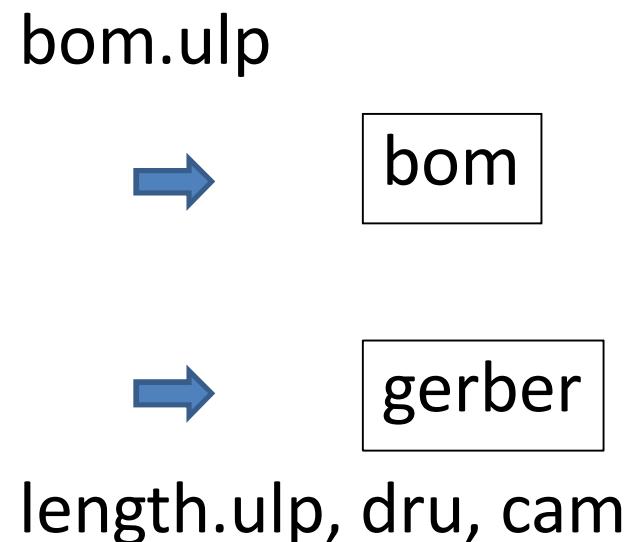
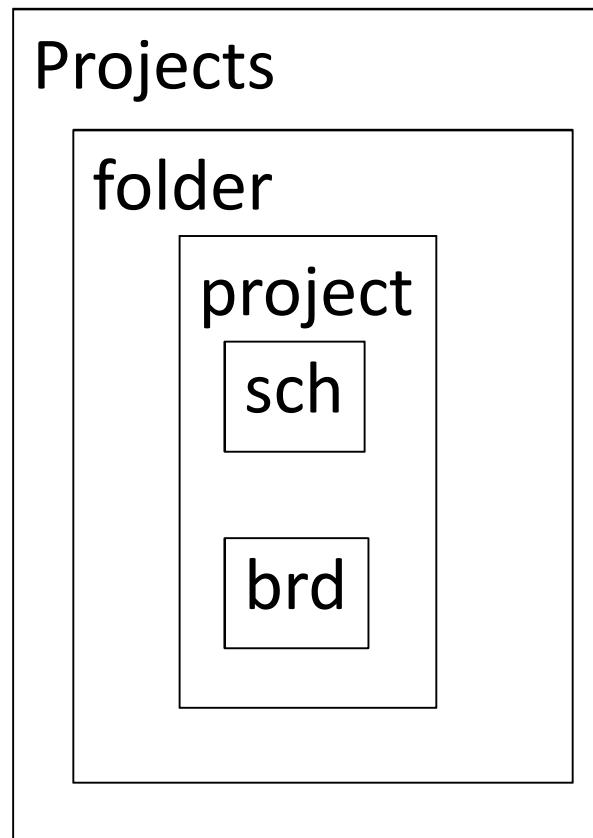


คำศัพท์

- Folder โฟลเดอร์ ซึ่งอนกันได้
- Project ไฟล์เดอร์สุดท้าย ที่ใช้เก็บ sch, brd
- sch - Schematic ผังวงจรรวม แสดงการต่อขาอุปกรณ์
- brd - Board แผ่นวงจรรวม
- bom – Bill of material รายการอุปกรณ์
- gerber – แบบพิมพ์เขียว pcb สำหรับส่งโรงงานผลิต
- dru – กฎโรงงานผลิต
- cam – แปลง brd เป็น gerber
- inch - นิว
- mil – 1/1000 นิว
- mm – มิลลิเมตร
- mic – 1/1000 มิลลิเมตร
- จุดไข่ปลา – 1 จุด = 2.54mm = 0.1 นิว



Projects

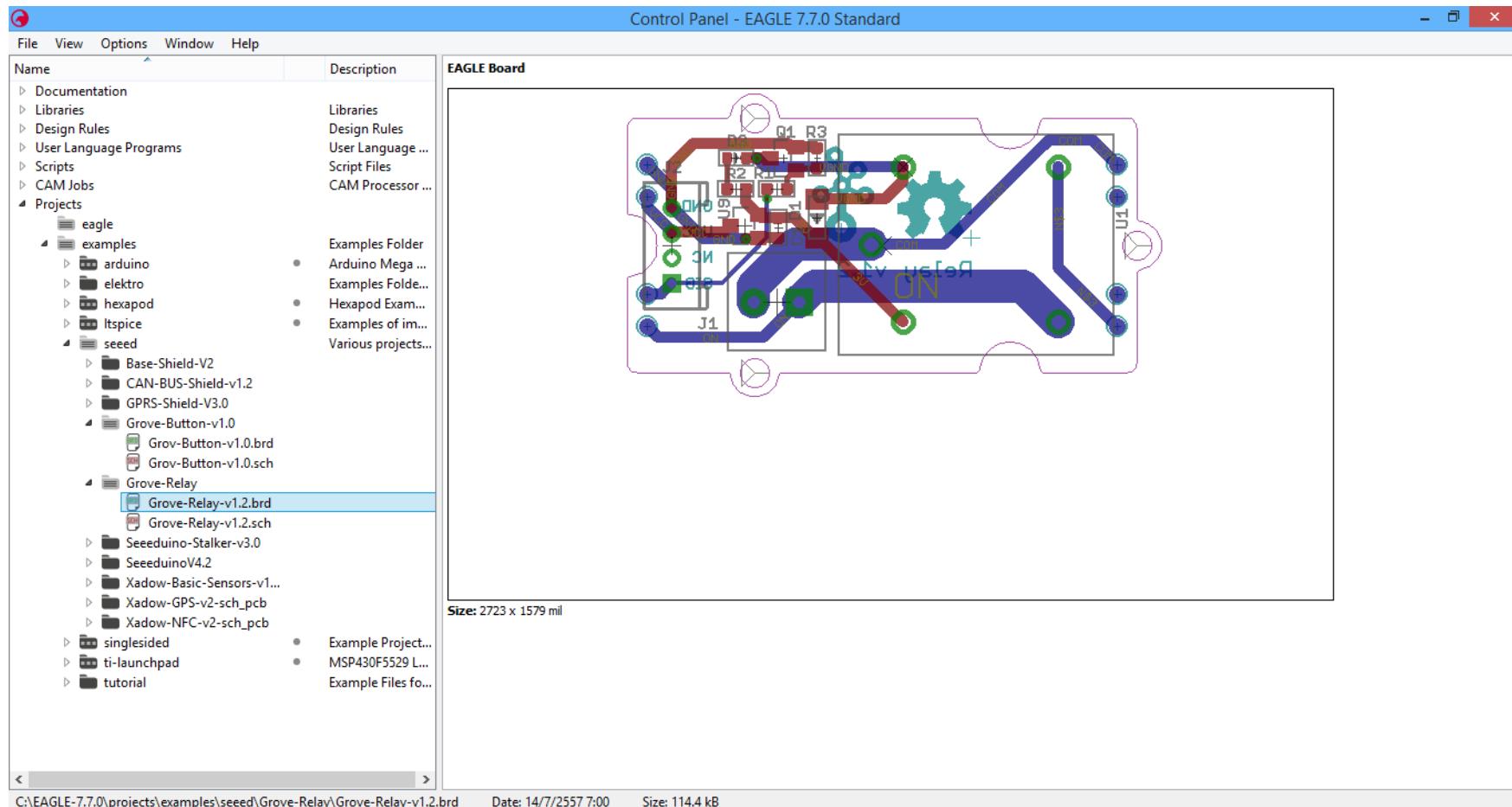


length.ulp, dru, cam

ชื่อไฟล์ sch กับชื่อไฟล์ brd ต้องเป็นชื่อเดียวกัน



Examples



Home > Get Started Now > Select a file on your computer



Iamloei

Home Shared Projects Pricing and Design Rules Support Blog Projects Order History Profile Cart Log out

OSH Park

Upload your design



You can upload your design as

- an Eagle **.brd** board file
- a KiCAD **.kicad_pcb** board file
- a **.zip** file containing Gerber CAM files



We support the default CAM filenames for most CAD packages. See our [design submission guidelines](#) or [design tool help](#) for more information.

Select a file on your computer

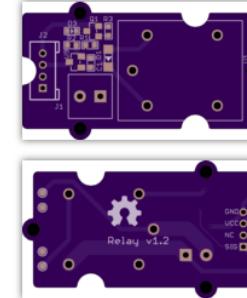
Designed and developed by [Rabid](#).

Detected 2 layer board of 1.65x0.94 inches (41.94x23.88mm). \$7.75 for three.

Your upload has finished processing. Enter the project details below and we'll move on to checking all the individual layers to make sure that they're correct.

Design notes:

- Processing Grove-Relay-v1.2.brd as Eagle CAM job.
- Detected 2 layer stackup: (1*16)
- 2 layer board of 1.65x0.94 inches.



Name

Description

Start Over

Continue

OSH Park
Verify your design

Board Top
This shows the final manufactured board as if you have it in your hand. You can see the raw gold copper, solder mask white, via holes, and the board outline. Internal cutouts are indicated by black outlines but if the image here is entirely white, you'll want to make sure there are no gaps in the board outline. Please make no dimension or measurement cuts.

Board Bottom
This shows the final manufactured board as if you have it in your hand. You can see the raw gold copper, solder mask white, via holes, and the board outline. Internal cutouts are indicated by black outlines but if the image here is entirely white, you'll want to make sure there are no gaps in the board outline. Please make no dimension or measurement cuts.

Drills
Drills should show up as white circles on a purple background. Drills are represented by their diameter in mils or Drill with (x,y) coordinates, and are not supported. Internal cutouts are indicated by black outlines but if the image here is entirely white, you'll want to make sure there are no gaps in the board outline. Please make no dimension or measurement cuts.

Top Solder Mask
This shows the portion of the silkscreen that should be a purple rectangle where to remove the purple solder mask. The gold-colored areas will be exposed on the final board, and purple areas will be covered in purple soldermask. If you submit an empty file we will remove the entire board, so make sure to fill the entire board with purple soldermask. To expose the entire board, submit this file a few times, once with a 1px border around the board, then a few more times with no border and expand at the expense of all everywhere and expand all the time. Please make sure to fill the entire board with purple soldermask.

Board Outline
The board outline should be a purple rectangle showing the board's edge of the board with no gaps. You can also show large irregular shapes and cutouts. See the [outline instructions](#) for more.

Bottom Silk Screen
This shows the portion of the silkscreen that should appear transparent as if it were being printed. You will ignore the portion of the silkscreen that extends beyond the board outline. We will automatically remove any silkscreen that is a logo isn't showing up on this layer, try changing the image to a higher resolution or a different image with 400 DPI or less, or check out our [Eagle Design Tools](#) page for more.

Top Layer
We will place copper everywhere we see gold color. If you are using Altium Designer or Altium Circutmaker, carefully examine the board to make sure there are no vias or other features being included on this layer. See [top](#) for more.

If you are using Eagle, be aware that areas are placed on the wrong side of the board. Make sure paper links between pads showing on this layer. Please make sure to fill the entire board with purple soldermask.

Bottom Solder Mask
This shows the portion of the silkscreen that should appear transparent as if you were looking down on it through the board from the top. We will place copper everywhere we see gold color. If you are using Altium Designer or Altium Circutmaker, carefully examine the board to make sure there are no vias or other features being included on this layer. See [bottom](#) for more.

If you are using Eagle, be aware that areas are placed on the same in mixed traces. If there are no areas, then the board will be fine. Please make sure to fill the entire board with purple soldermask.

Bottom Layer
We will ignore the portion of the silkscreen that extends beyond the board outline. We will ignore the portion of the silkscreen that increases drilled holes or exposed copper. A logo isn't showing up on this layer, try changing the image to a higher resolution or a different image with 400 DPI or less, or check out our [Eagle Design Tools](#) page for more.

Top Silk Screen
We will ignore the portion of the silkscreen that extends beyond the board outline. We will ignore the portion of the silkscreen that increases drilled holes or exposed copper. A logo isn't showing up on this layer, try changing the image to a higher resolution or a different image with 400 DPI or less, or check out our [Eagle Design Tools](#) page for more.

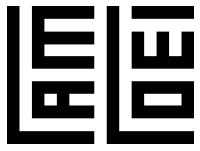
Bottom
Rendered from "Grove-Relay-v1.2.brd"

Approved

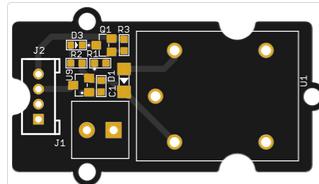
Approve and Order



oshpark

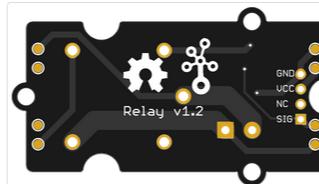


Board Images:



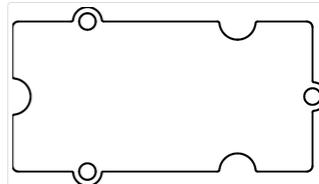
Top View - Complete Board

This is what your finished board will look like from the top.



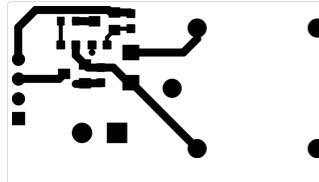
Bottom View - Complete Board

This is what your finished board will look like from the bottom.



Board Outline

This shows the outline of your board, it will be cut to this shape.



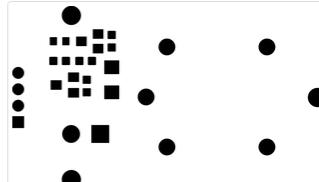
Top Copper Layer

This shows where the copper will remain after etching the top of the board.



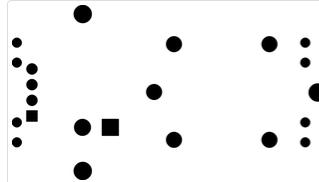
Bottom Copper Layer

This shows where the copper will remain after etching the bottom of the board. (mirrored)



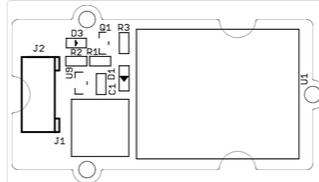
Top Soldermask

This shows where soldermask will be applied on the top over the bare copper board.



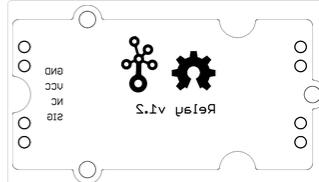
Bottom Soldermask

This shows where soldermask will be applied on the bottom over the bare copper board. (mirrored)



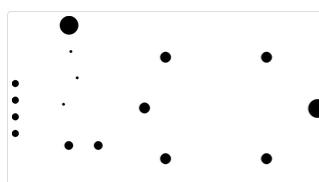
Top Silkscreen

This is the white print that will be printed on the top of the board.



Bottom Silkscreen

This is the white print that will be printed on the bottom of the board. (mirrored)



Drill Holes

This shows all the holes that will be drilled in your board.

pcbs

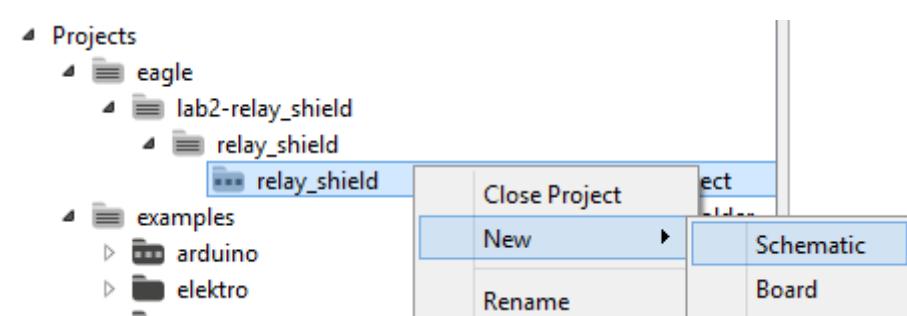
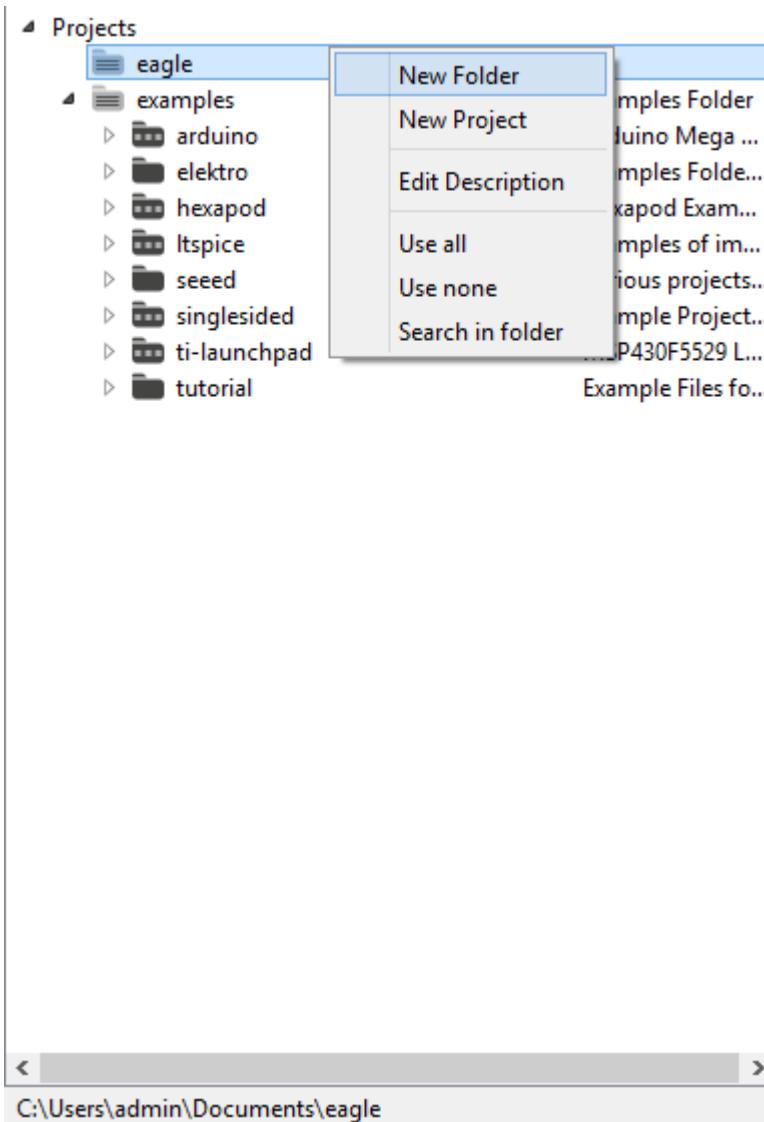


Break;

- 5 นาที



New Project

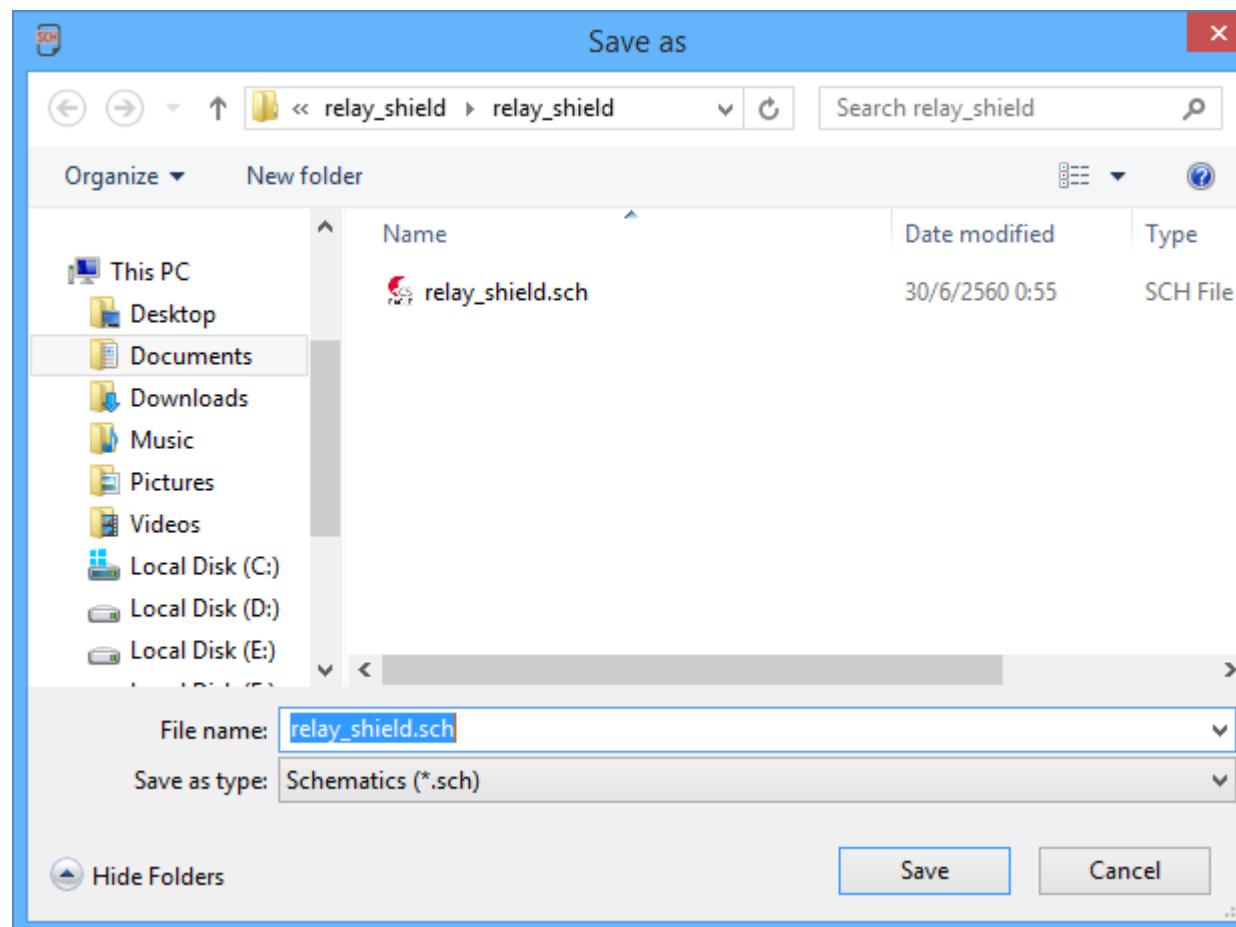


New > Schematic



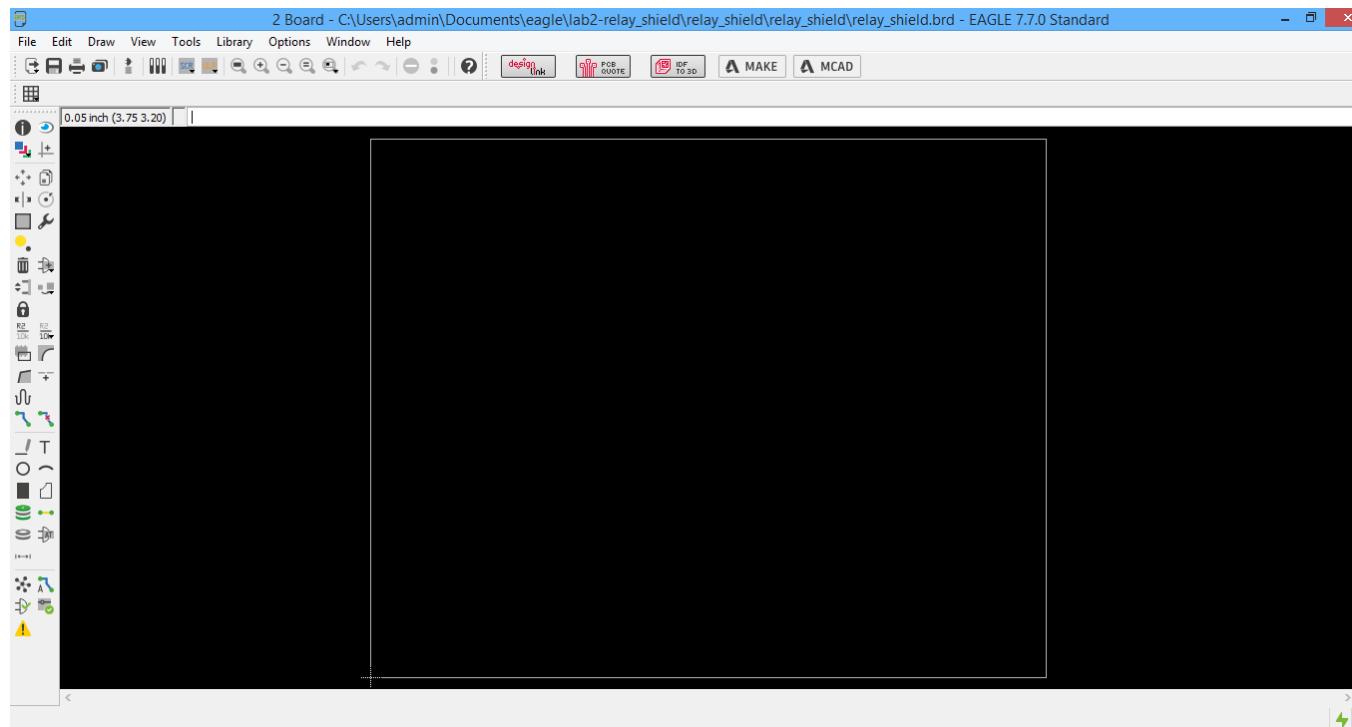
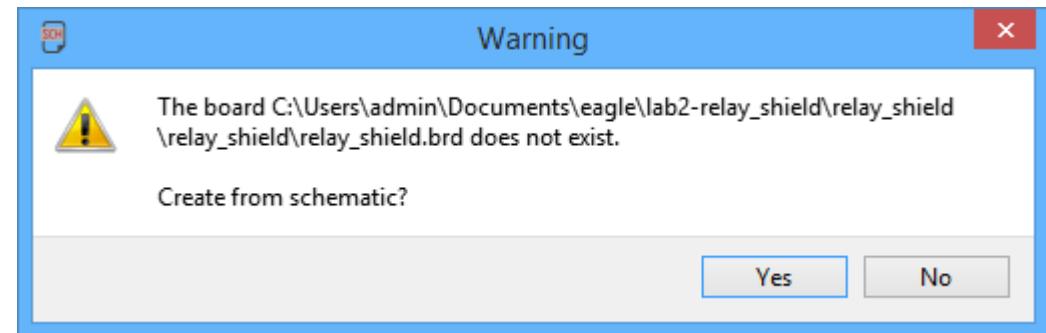
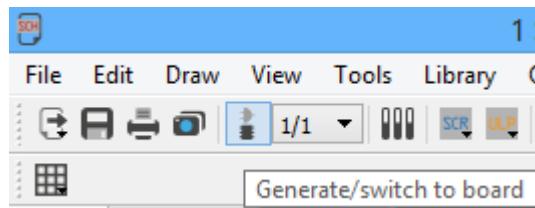
Save Schematic

Save ตั้งชื่อไฟล์ Schematic





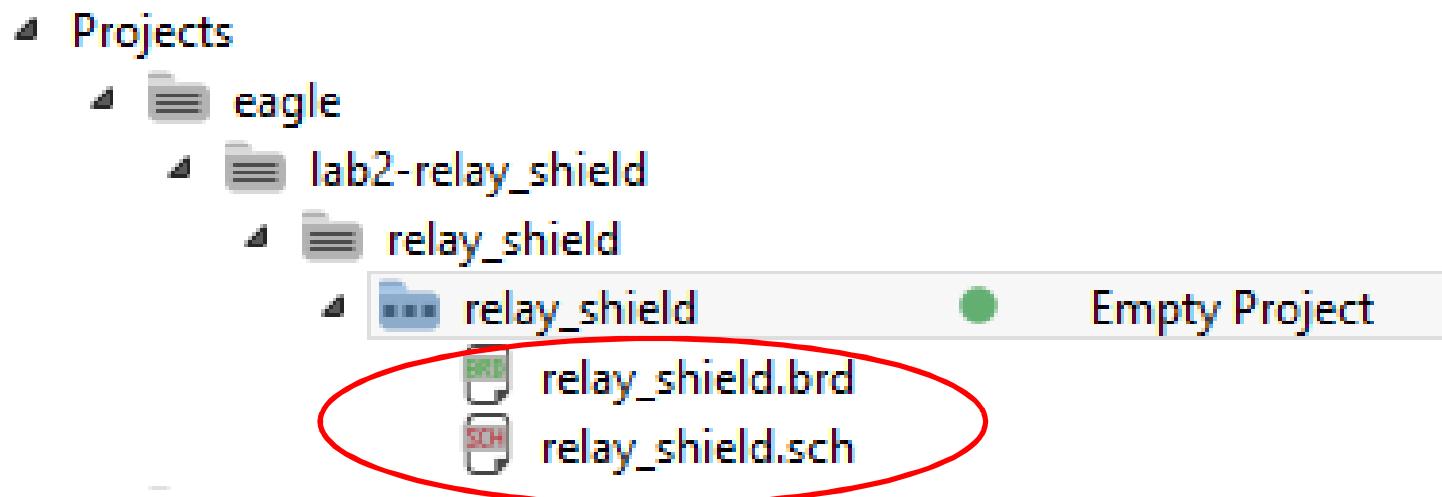
New Board



แล้วกดบันทึก



Lab 2 สร้าง sch กับ brd ที่ซื้อเหมือนกัน อยู่ภายใต้ projectเดียวกัน

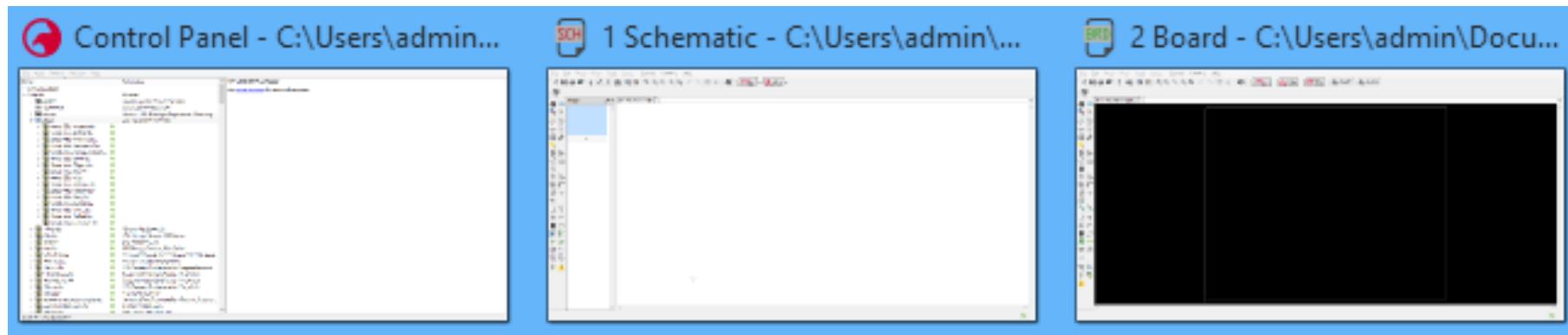




Control Panel

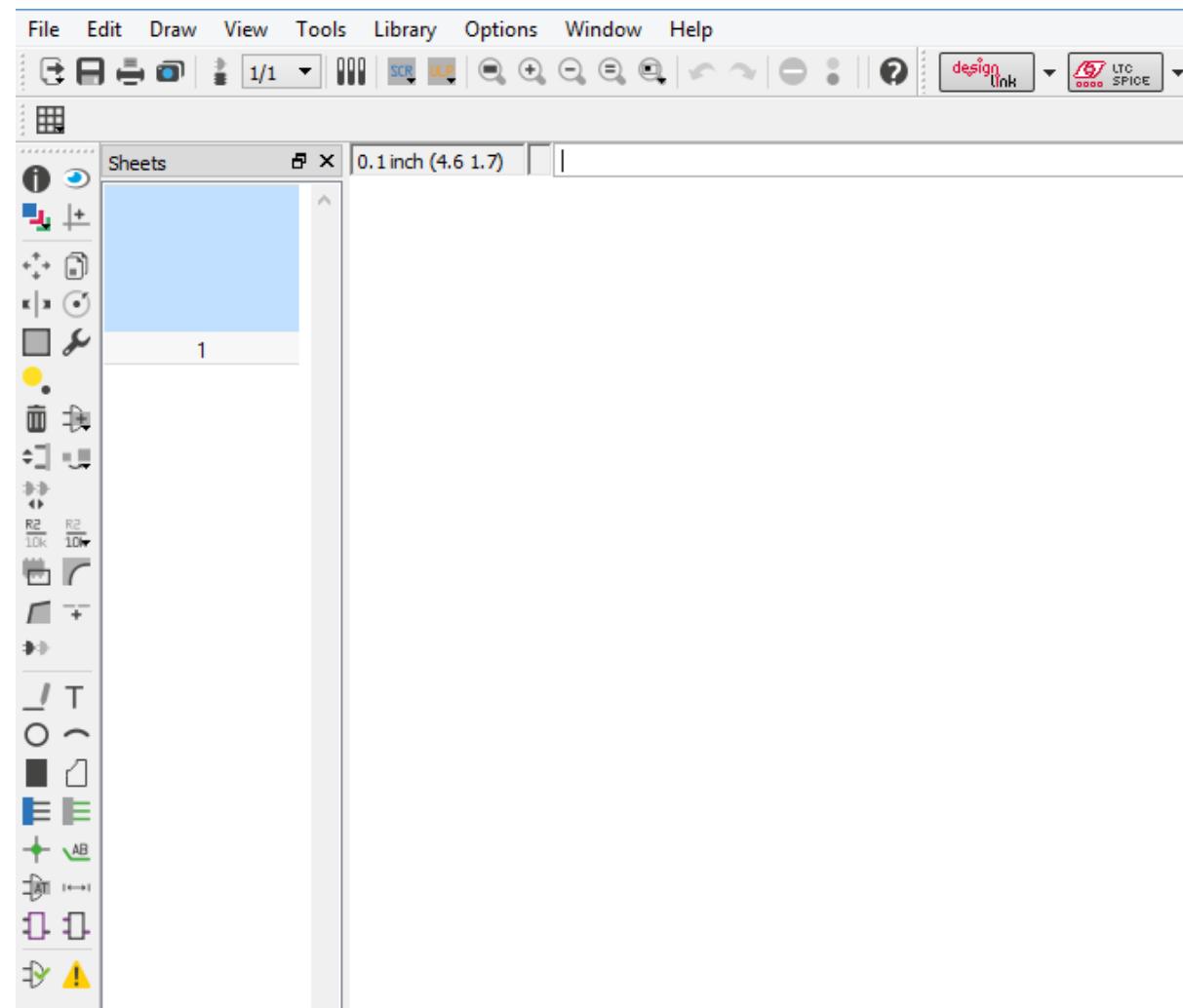
Schematic

Board



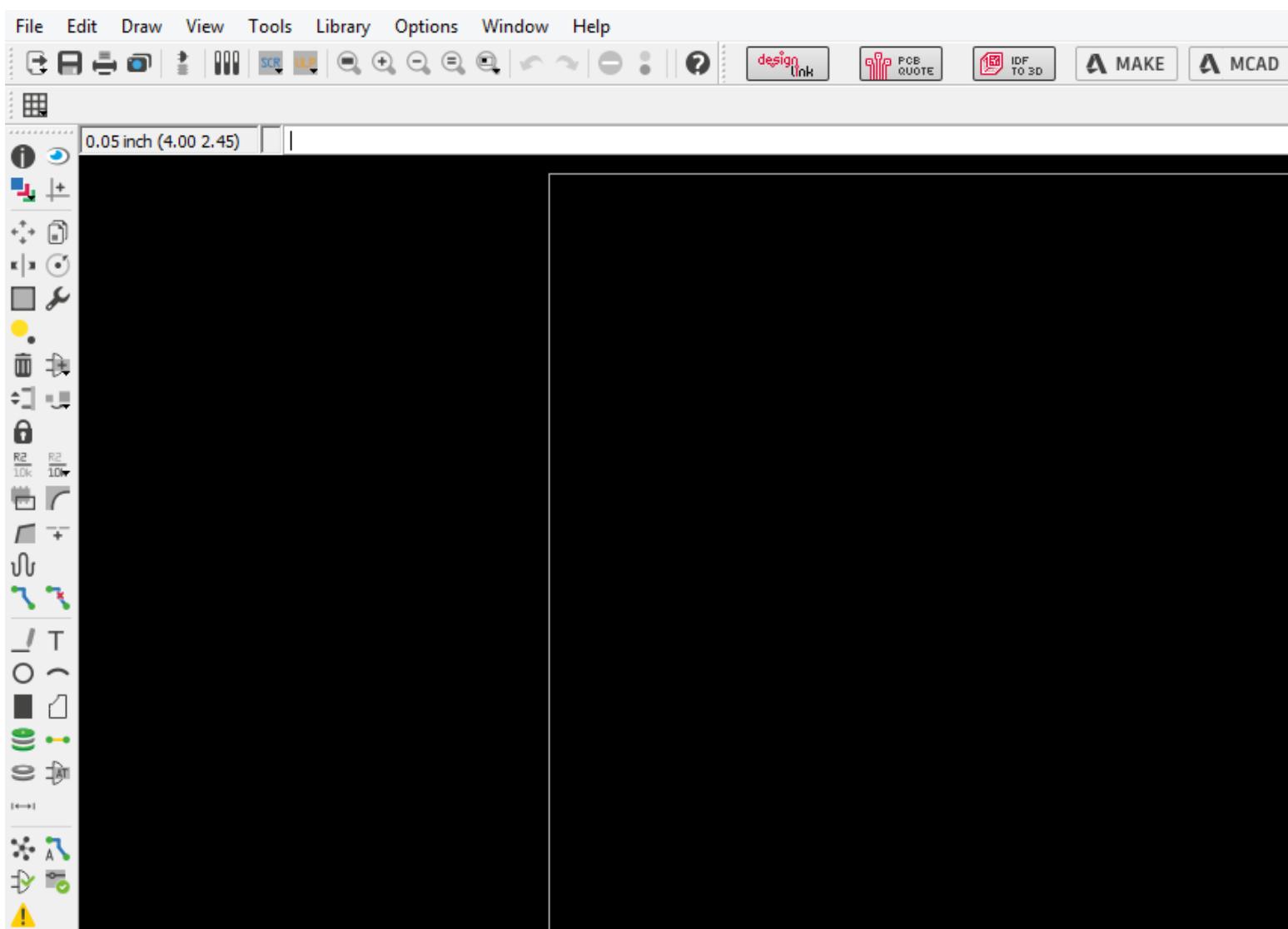


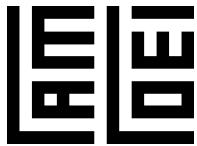
Schematic



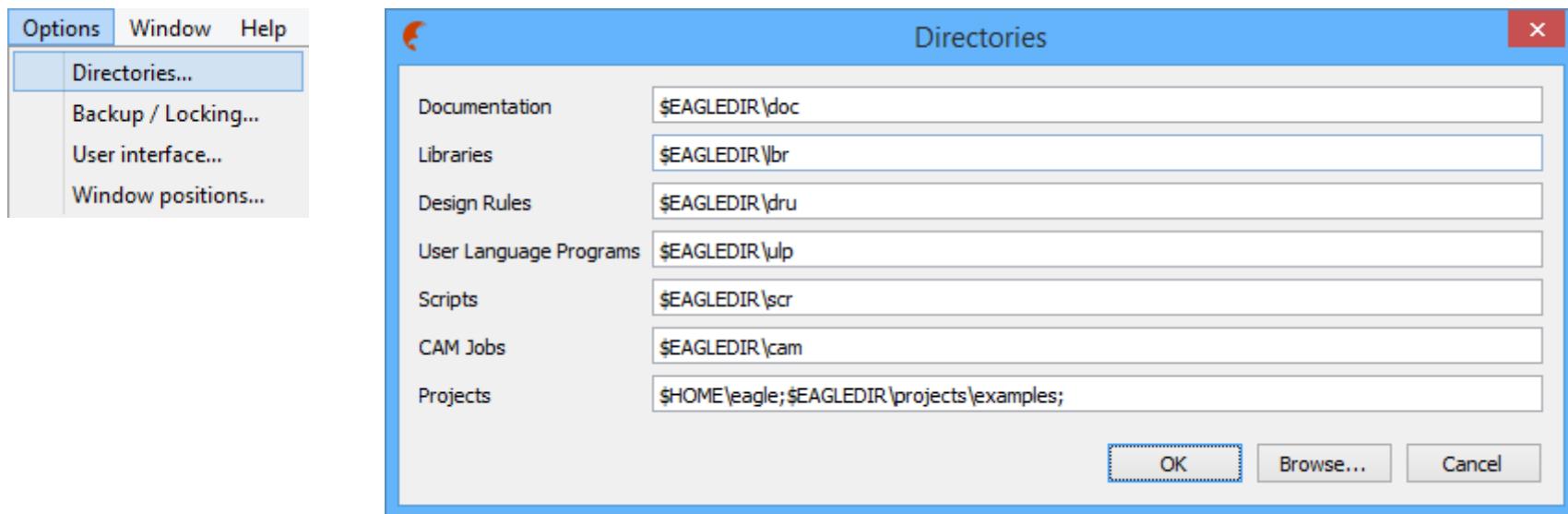


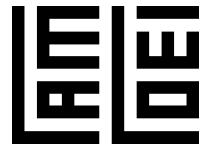
Board



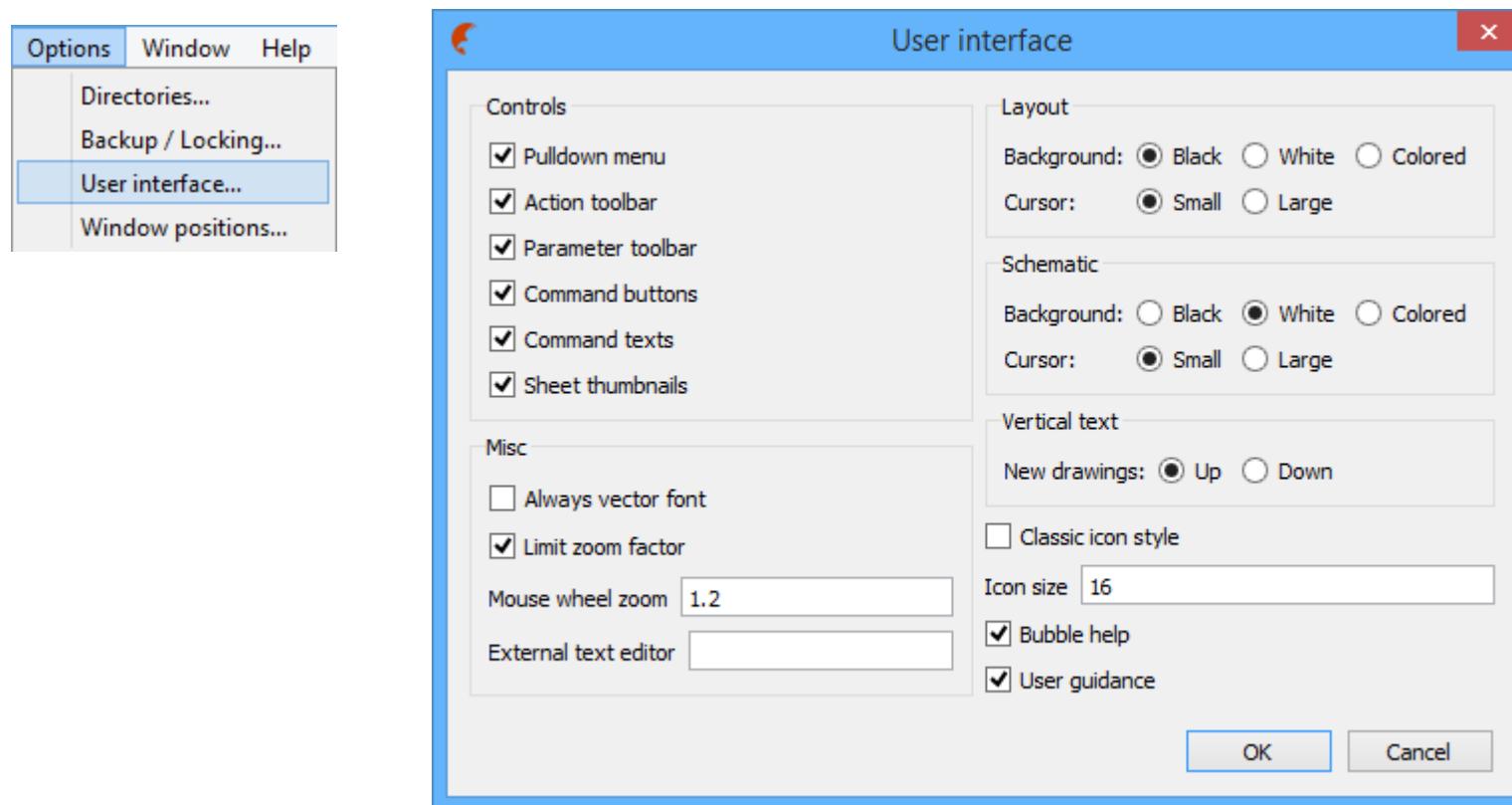


Control Panel > Options > Directories...



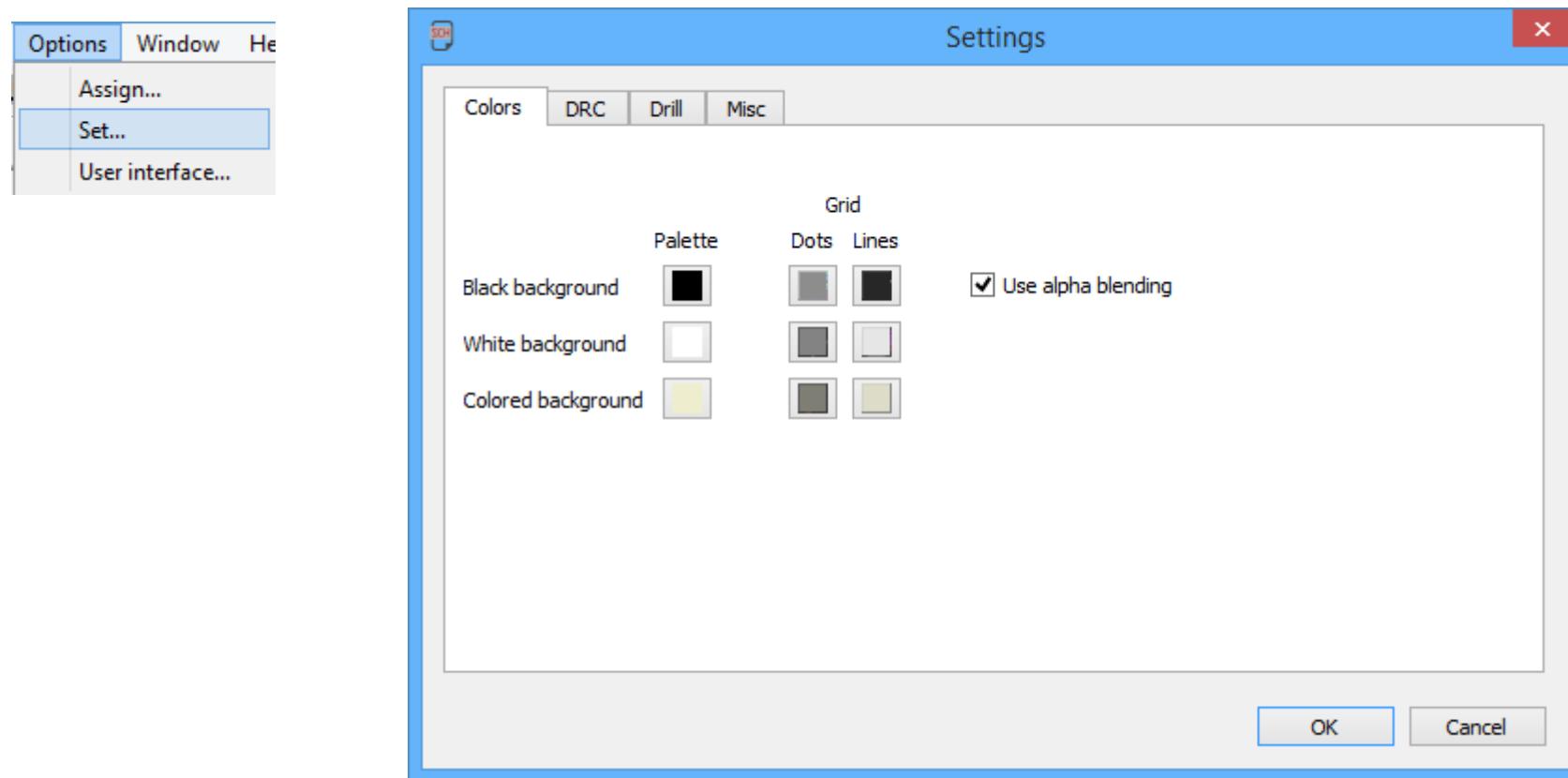


Control Panel > Options > User interface





Sch > Options > Set...
Brd > Options > Set...





Sch Button & Brd Button





Lab 3 កំណត់លក្ខណៈ lbr, dru, cam

Name	Date modified	Type	Size
elektro	29/6/2560 10:05	File folder	
element14	29/6/2560 10:05	File folder	
Itspice	29/6/2560 10:05	File folder	
seed	29/6/2560 10:05	File folder	
Adafruit-Eagle-Library-master	30/6/2560 14:53	File folder	
opl_package	30/6/2560 14:53	File folder	
SparkFun-Eagle-Libraries-master	30/6/2560 14:53	File folder	
DESCRIPTION	13/12/2549 4:16	File	2 KB
40xx.lbr	13/12/2554 5:12	LBR File	370 KB
41xx.lbr	13/12/2554 5:12	LBR File	29 KB
45xx.lbr	13/12/2554 5:12	LBR File	230 KB
74ac-logic.lbr	13/12/2554 5:12	LBR File	380 KB
751xx.lbr	13/12/2554 5:12	LBR File	79 KB

PC > Local Disk (C:) > EAGLE-7.7.0 > cam				
Name	Date modified	Type	Size	
Elecrow_Gerber_Generator_DrillAlign.cam	29/6/2560 20:08	Altium CAMtastic ...	11 KB	
Elecrow_Gerber_Generator_4-layer_1-2-15... Seeed_Gerber_Generator_2-layer.cam	29/6/2560 20:07	Altium CAMtastic ...	13 KB	
PCBs_ioEagle7.2.cam	29/6/2560 20:03	Altium CAMtastic ...	11 KB	
OSHPark-4layer-Eagle7.2.cam	29/6/2560 19:58	Altium CAMtastic ...	7 KB	
OSHPark-2layer-Eagle7.2.cam	29/6/2560 19:56	Altium CAMtastic ...	10 KB	
gerb274x-mill-drill.cam	31/8/2558 7:04	Altium CAMtastic ...	9 KB	
Seeed_Gerber_Generator_4-layer_1-2-15... Seeed_Gerber_Generator_2-layer.cam	3/12/2556 14:42	Altium CAMtastic ...	13 KB	

PC ▶ Local Disk (C:) ▶ EAGLE-7.7.0 ▶ dru

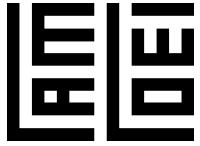
Name	Date modified	Type	Size
📁 Eurocircuits	29/6/2560 10:05	File folder	
📁 Multi-CB	29/6/2560 10:05	File folder	
📁 WEdirekt	29/6/2560 10:05	File folder	
📄 default.dru	13/12/2549 4:16	DRU File	3 KB
📄 DESCRIPTION	13/12/2549 4:16	File	1 KB
📄 Elecrow_2-layer_PCB_eagle_rule.dru	29/6/2560 20:07	DRU File	3 KB
📄 Elecrow_4-layer_PCB_eagle_rule.dru	29/6/2560 20:07	DRU File	3 KB
📄 oshpark-2layer.dru	29/6/2560 19:53	DRU File	3 KB
📄 OSHPark-4layer.dru	29/6/2560 19:56	DRU File	3 KB
📄 PCBs_io.dru	29/6/2560 19:58	DRU File	3 KB
📄 SeeedStudio_2layer_DRU_no_angle_20140...	17/6/2557 16:36	DRU File	3 KB
📄 SeeedStudio_4layer_DRU_no_angle_20140...	26/8/2558 9:41	DRU File	2 KB

//ให้ลบเปลี่ยน หรือเปลี่ยน lbr เดิมก่อน
ปิดเปิด Eagle ใหม่ หรือกด F5

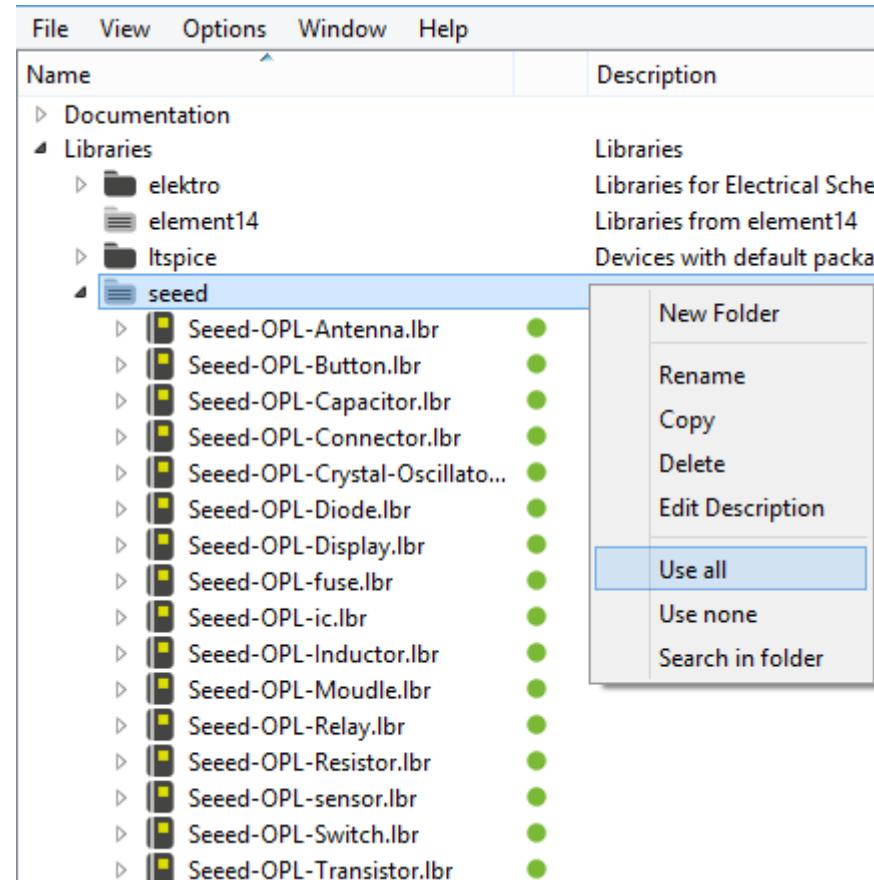


Control panel จะมีรายชื่อเพิ่มขึ้นมา

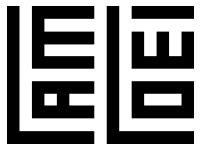
Libraries	Libraries
Adafruit-Eagle-Library-master	Libraries for Electrical Schema
elektro	Libraries from element14
element14	Devices with default packages
Itspice	
opl_package	
seed	OPL Libraries from Seeed
SparkFun-Eagle-Libraries-master	
Design Rules	Design Rules
Eurocircuits	EAGLE Design Rules
Multi-CB	EAGLE Design Rules
WE direkt	EAGLE Design Rules
default.dru	EAGLE Design Rules
Elecrow_2-layer_PCB_eagle_rule....	EAGLE Design Rules
Elecrow_4-layer_PCB_eagle_rule....	EAGLE Design Rules
oshpark-2layer.dru	EAGLE Design Rules
OSHPark-4layer.dru	EAGLE Design Rules
PCBs_io.dru	MakerWorks Design Rules
SeeedStudio_2layer_DRU_no_an...	EAGLE Design Rules
SeeedStudio_4layer_DRU_no_an...	EAGLE Design Rules
CAM Jobs	CAM Processor Jobs
Elecrow_Gerber_Generator_4-lay...	Seeed Studio 2-layer PCB Gerber Generator
Elecrow_Gerber_Generator_Drill...	Seeed Studio 2-layer PCB Gerber Generator
excellon.cam	Generates Excellon Drill Data
gerb274x-4layer.cam	Generates Extended Gerber Format for a 4 la
gerb274x-mill-drill.cam	Generates Extended Gerber Format
gerb274x.cam	Generates Extended Gerber Format
gerber.cam	Generates Gerber Format
layout2.cam	Generates EPS Format
OSHPark-2layer-Eagle7.2.cam	For 2 layer PCB designs using Eagle version
OSHPark-4layer-Eagle7.2.cam	For 4 layer PCB designs using Eagle version
PCBs_ioEagle7.2.cam	2 layer PCB designs for Marker.Works using
schematic.cam	Example for cam2printer.ulp
Seeed_Gerber_Generator_2-layer...	Seeed Studio 2-layer PCB Gerber Generator
Seeed_Gerber_Generator_4-layer...	Seeed Studio 2-layer PCB Gerber Generator



Use all Libraries

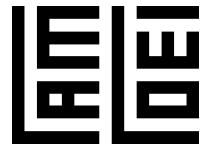


เปิดใช้เฉพาะเท่าที่จำเป็น จะทำงานเร็วกว่า

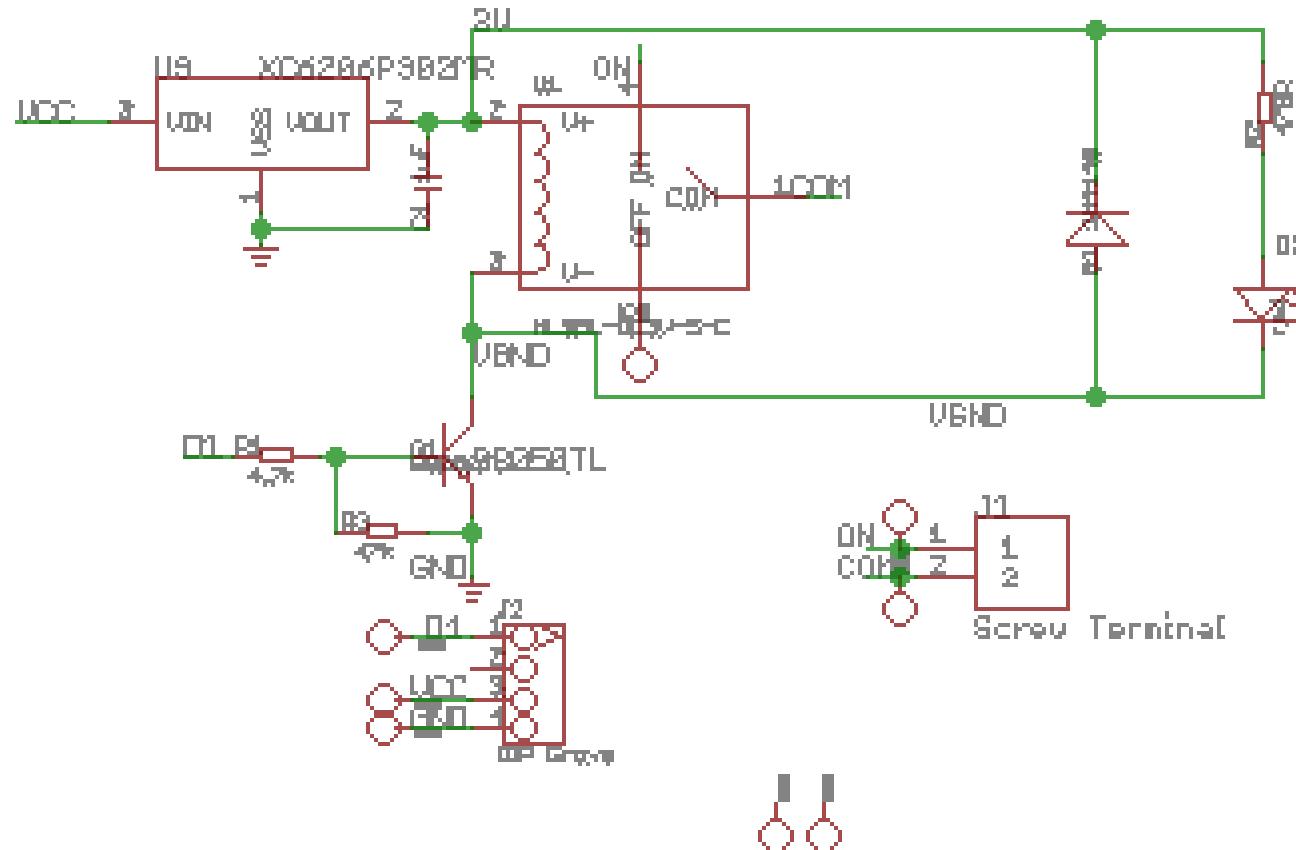


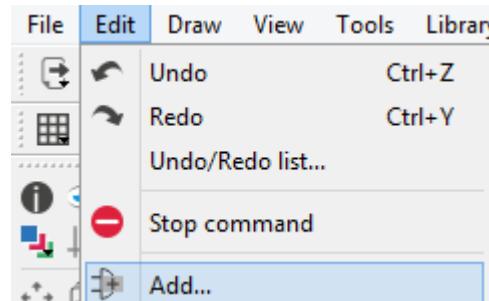
Control Panel - C:\User\	
Name	Description
Documentation	
Libraries	
Adafruit-Eagle-Library-master	
adafruit.lbr	
opl_package	
SparkFun-Eagle-Libraries-master	
Design Rules	Design Rules
Eurocircuits	
Multi-CB	
WEdirekt	
default.dru	EAGLE Design Rules
Elecrow_2-layer_PCB_eagle_rule....	EAGLE Design Rules
Elecrow_4-layer_PCB_eagle_rule....	EAGLE Design Rules
oshpark-2layer.dru	EAGLE Design Rules
OSHPark-4layer.dru	EAGLE Design Rules
PCBs_io.dru	MakerWorks Design Rules
SeedStudio_2layer_DRU_no_an...	EAGLE Design Rules
SeedStudio_4layer_DRU_no_an...	EAGLE Design Rules
User Language Programs	User Language Programs
Scripts	Script Files
CAM Jobs	CAM Processor Jobs
Elecrow_Gerber_Generator_4-lay...	Seeed Studio 2-layer PCB Ge...
Elecrow_Gerber_Generator_Drill...	Seeed Studio 2-layer PCB Ge...
excellon.cam	Generates Excellon Drill Data
gerb274x-4layer.cam	Generates Extended Gerber ...
gerb274x-mill-drill.cam	Generates Extended Gerber ...
gerb274x.cam	Generates Extended Gerber ...
gerber.cam	Generates Gerber Format
layout2.cam	Generates EPS Format
OSHPark-2layer-Eagle7.2.cam	For 2 layer PCB designs usin...
OSHPark-4layer-Eagle7.2.cam	For 4 layer PCB designs usin...
PCBs_ioEagle7.2.cam	2 layer PCB designs for Mar...
schematic.cam	Example for cam2printer.ulp
Seed_Gerber_Generator_2-layer...	Seeed Studio 2-layer PCB Ge...
Seed_Gerber_Generator_4-layer...	Seeed Studio 2-layer PCB Ge...
Projects	
eagle	
lab2-relay_shield	
relay_shield	
relay_shield	
relay_shield.brd	
relay_shield.sch	
examples	Examples Folder

เปิดใช้ lbr ที่จำเป็น



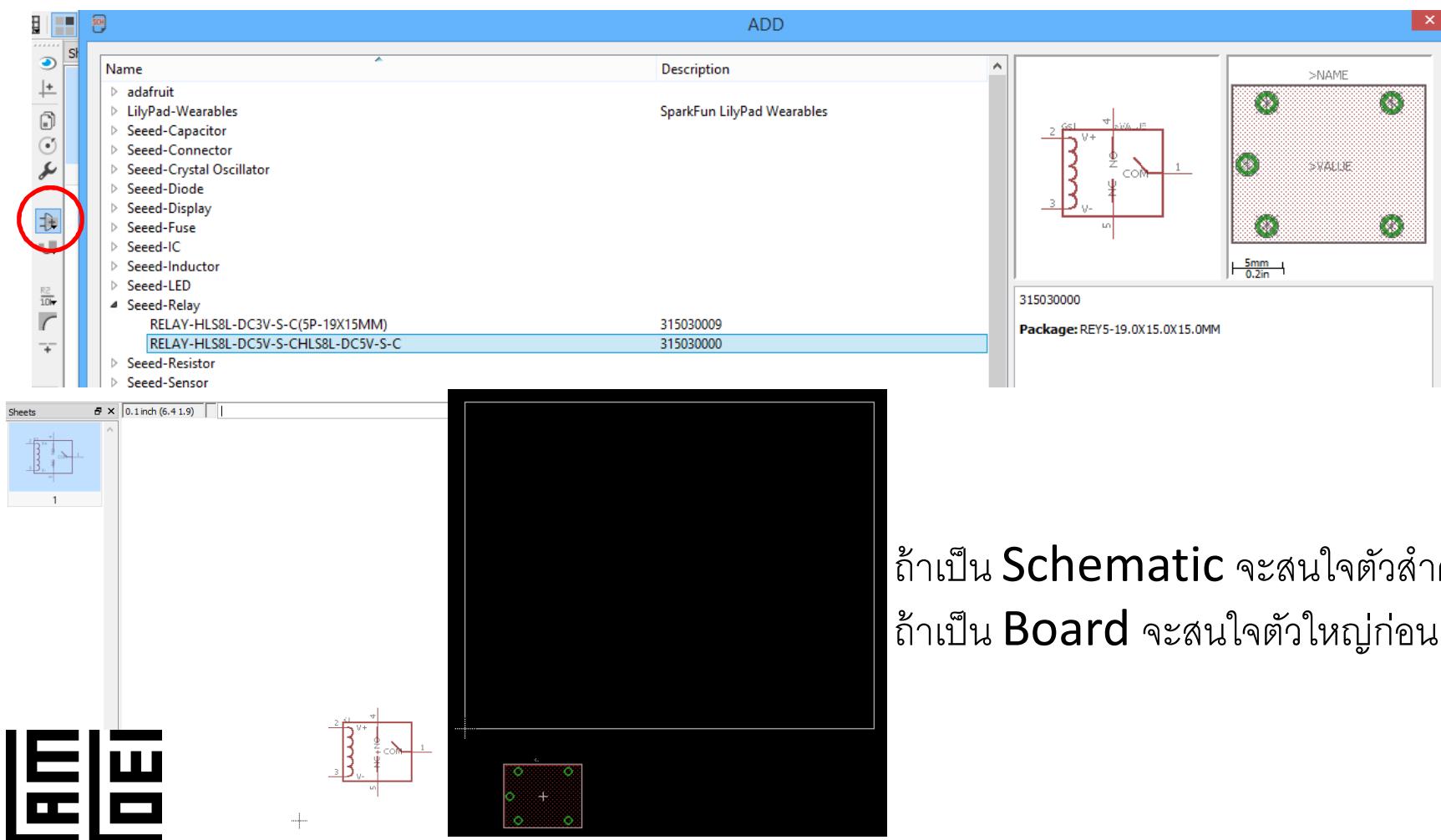
Lab 4 สร้าง relay module



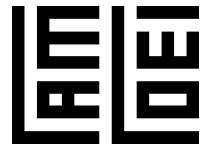


Lab4 Schematic > Edit > Add...

Sch > ADD > Seeed-Relay > RELAY-DC5V

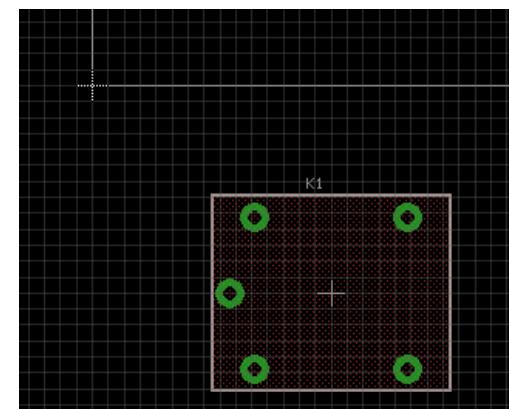
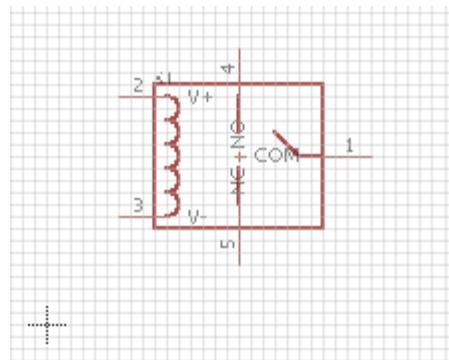
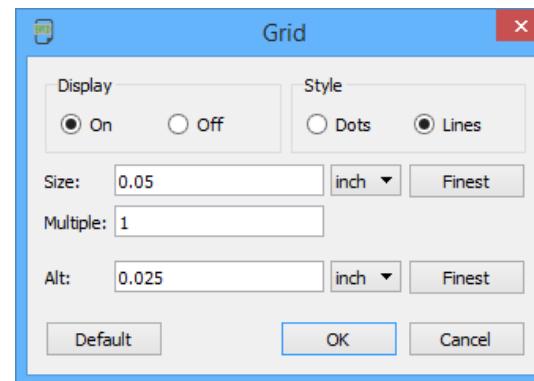
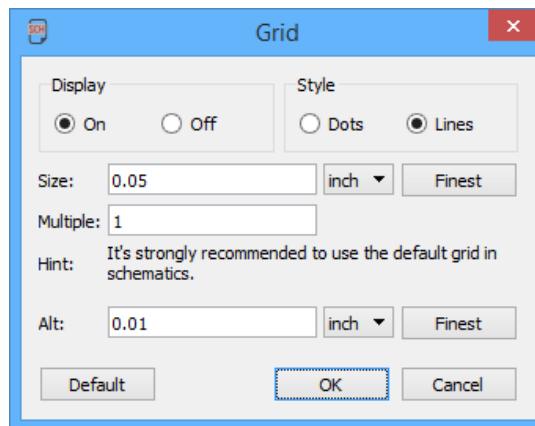


ถ้าเป็น Schematic จะสนใจตัวสำคัญก่อน
ถ้าเป็น Board จะสนใจตัวใหญ่ก่อน



Show Grid

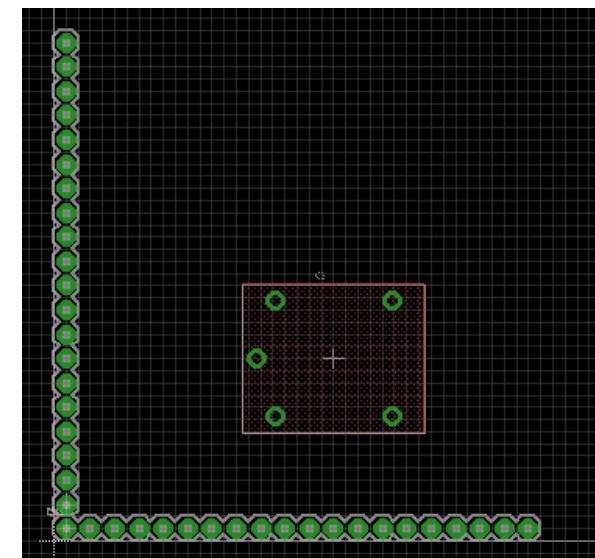
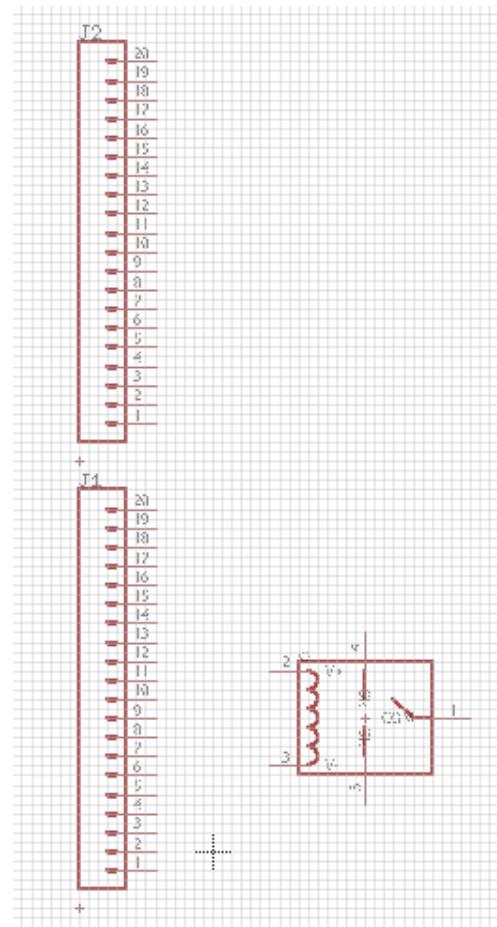
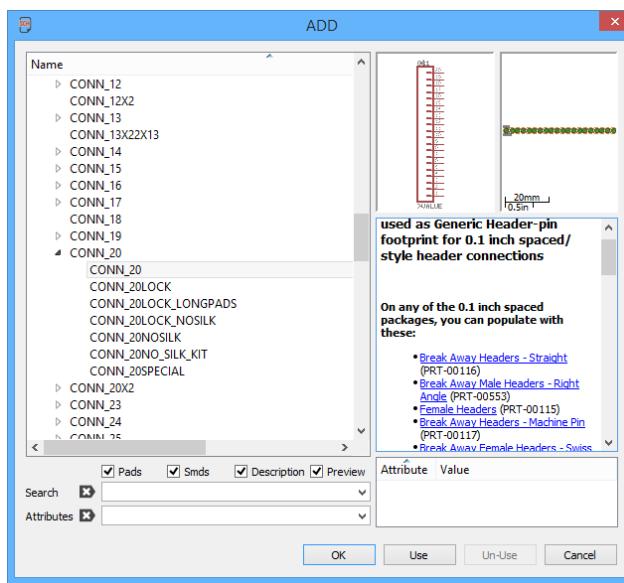
- Schematic > View > Grid... เลือก On
- Board > View > Grid... เลือก On

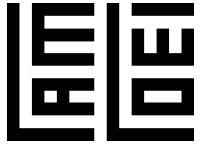




Pin Grid

Sch > Add > SparkFun_Connects > CONN_20 > CONN_20" >
ຈານ 2 ຂື່ນ



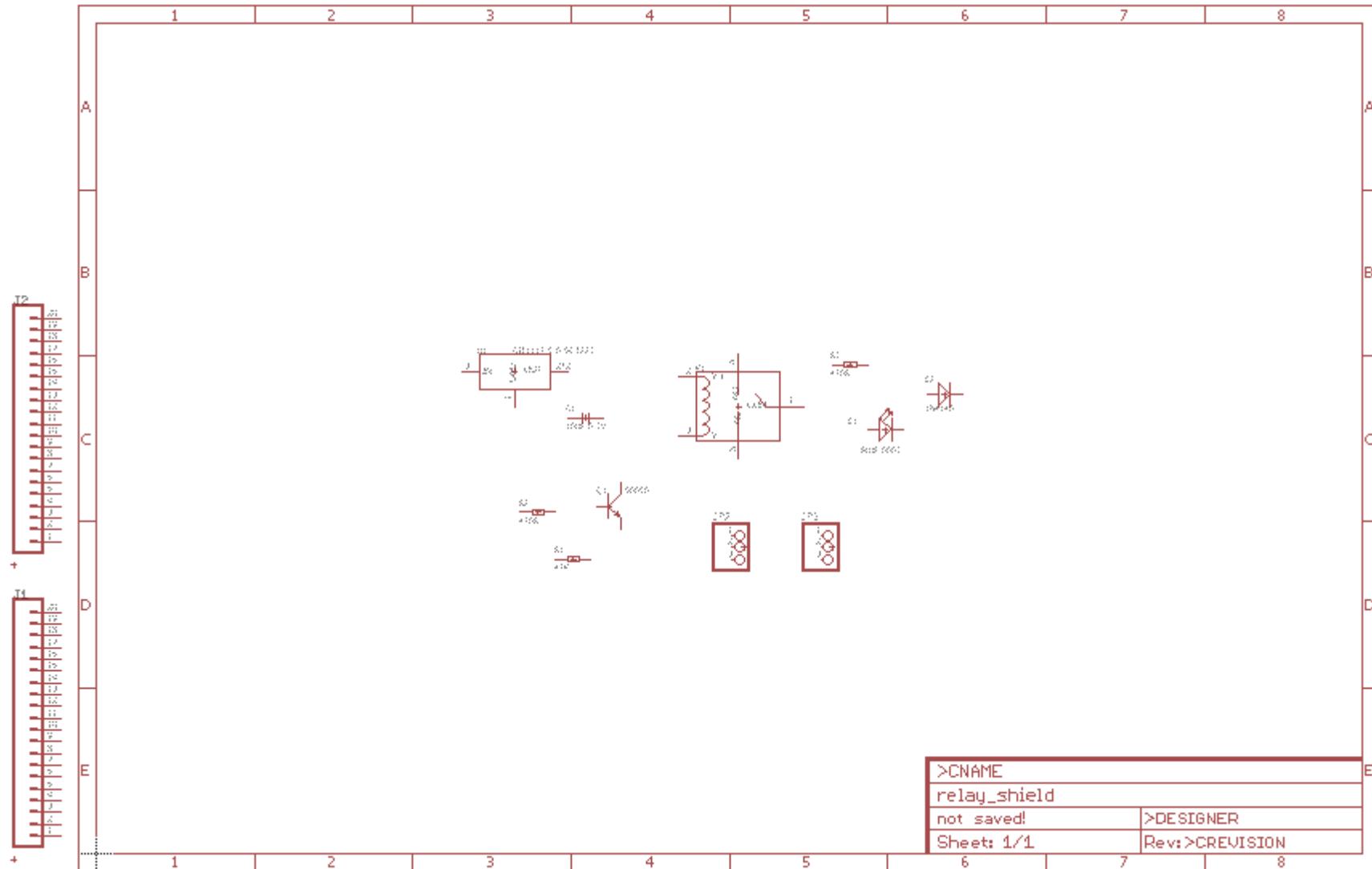


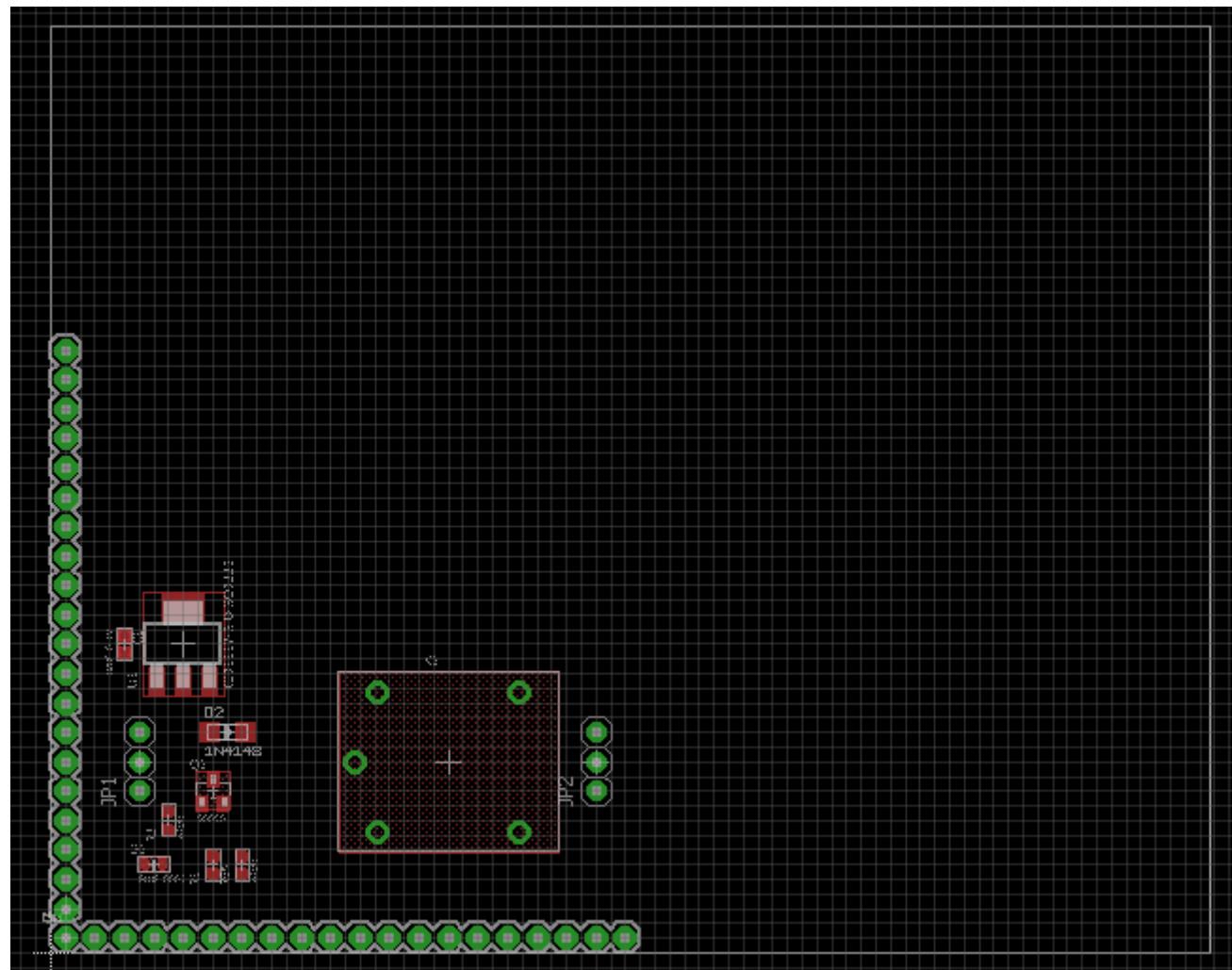
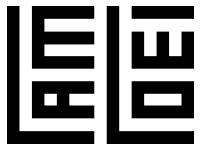
ເພີ່ມອຸປະກອນ

- Sch > ADD > Seeed-Transistor > s8050
- Sch > ADD > Seeed-Resistors > SMD-res-47k-0603
- Sch > ADD > Seeed-Resistors > SMD-res-470-0603 *2
- Sch > ADD > Seeed-LED > SMD-LED-0603
- Sch > ADD > Seeed-Diode > SMD 1N4148 (LL-34)
- Sch > ADD > Seeed-IC > PMIC-CJT1117-5.0(SOT223)
- Sch > ADD > Seeed-Capacitor > 10uf 6.3v 0603
- Sch > ADD > Adafruit > PINHD-1x3 > PINHD-1x3*2
- Sch > ADD > SparkFun-Aesthetics > FRAME-A4L



Lab 5 เรียนอุปกรณ์

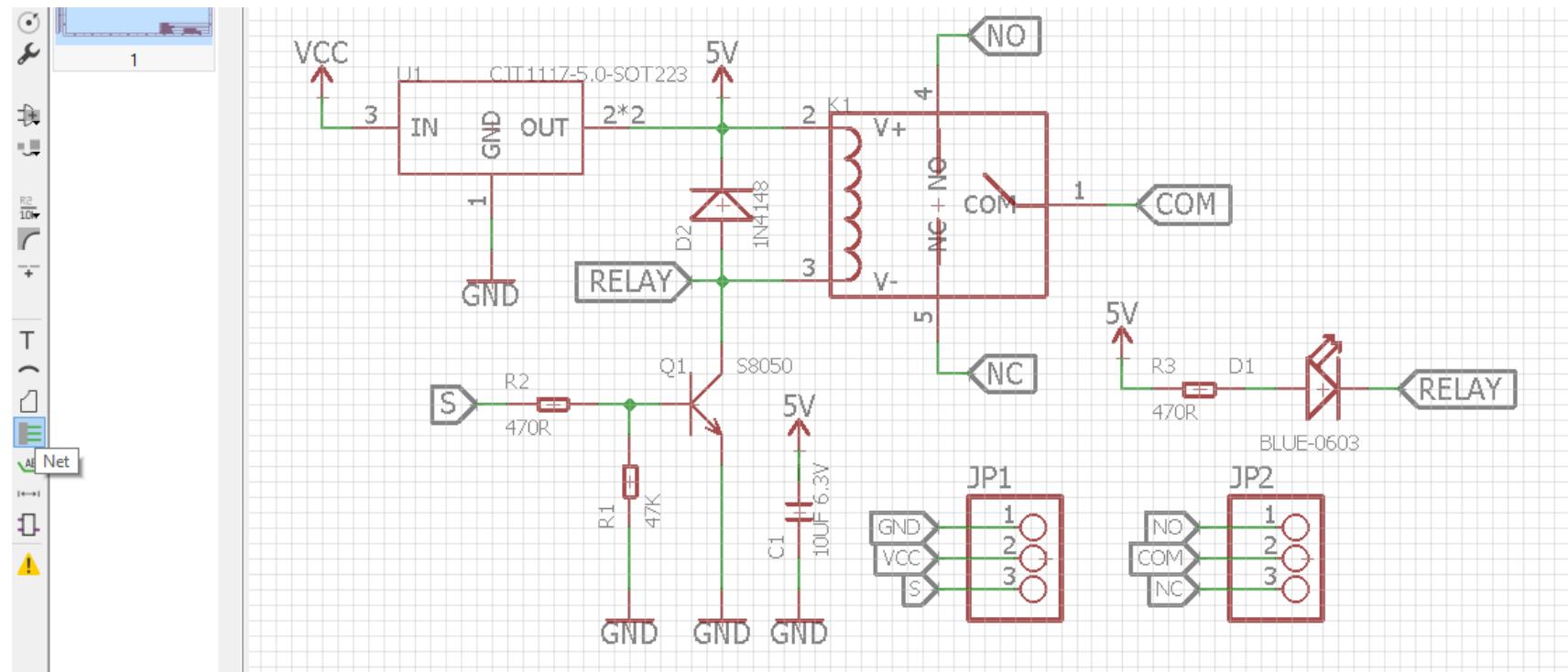




สามารถลงได้ทีละอุปกรณ์ได้



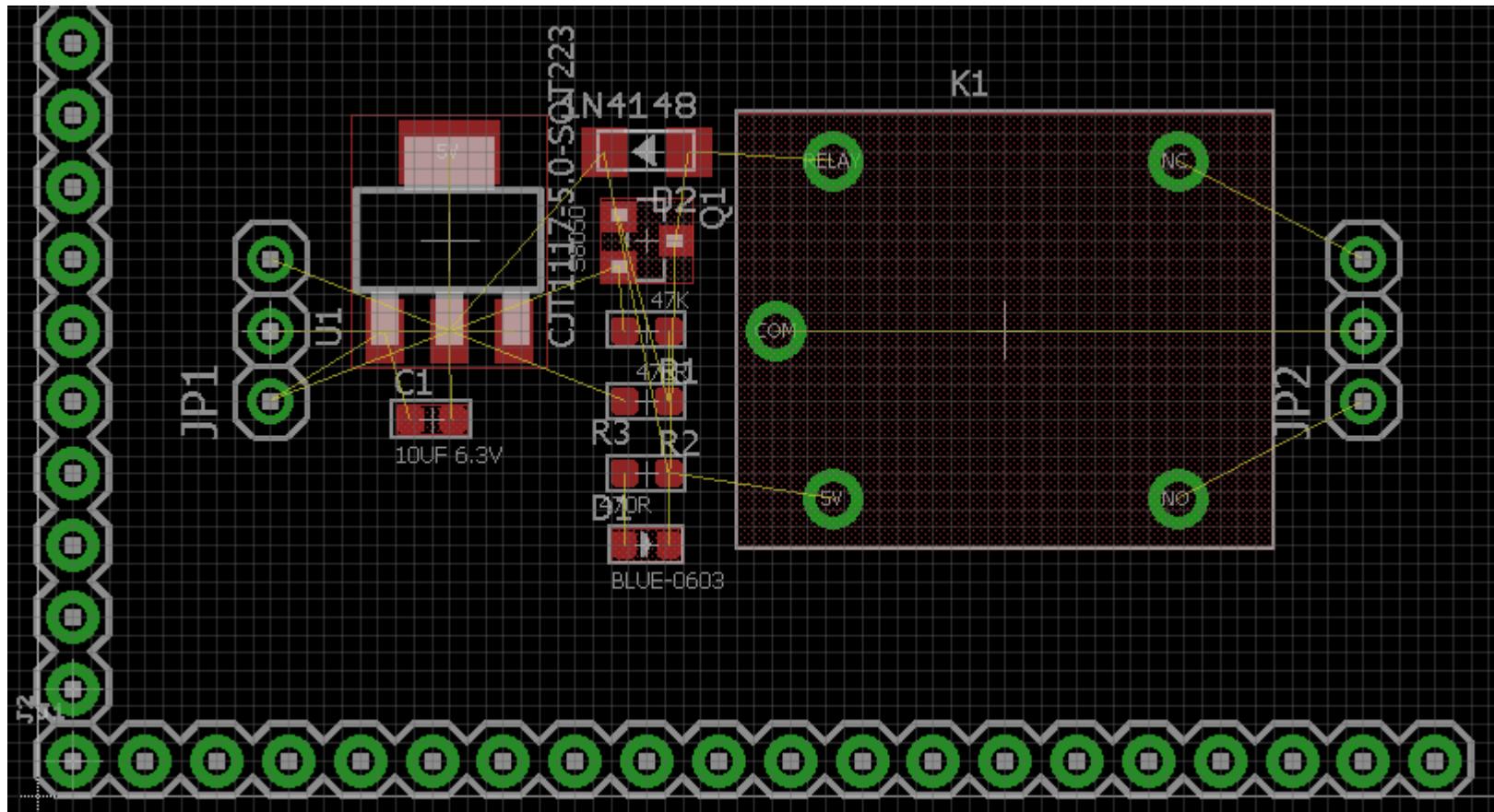
Lab 6 ลากเส้นโดยใช้ Net



Sch > ADD > SparkFun-PowerSymbols > VCC, 5V, GND



เรียนรู้การณ์บันบอร์ด



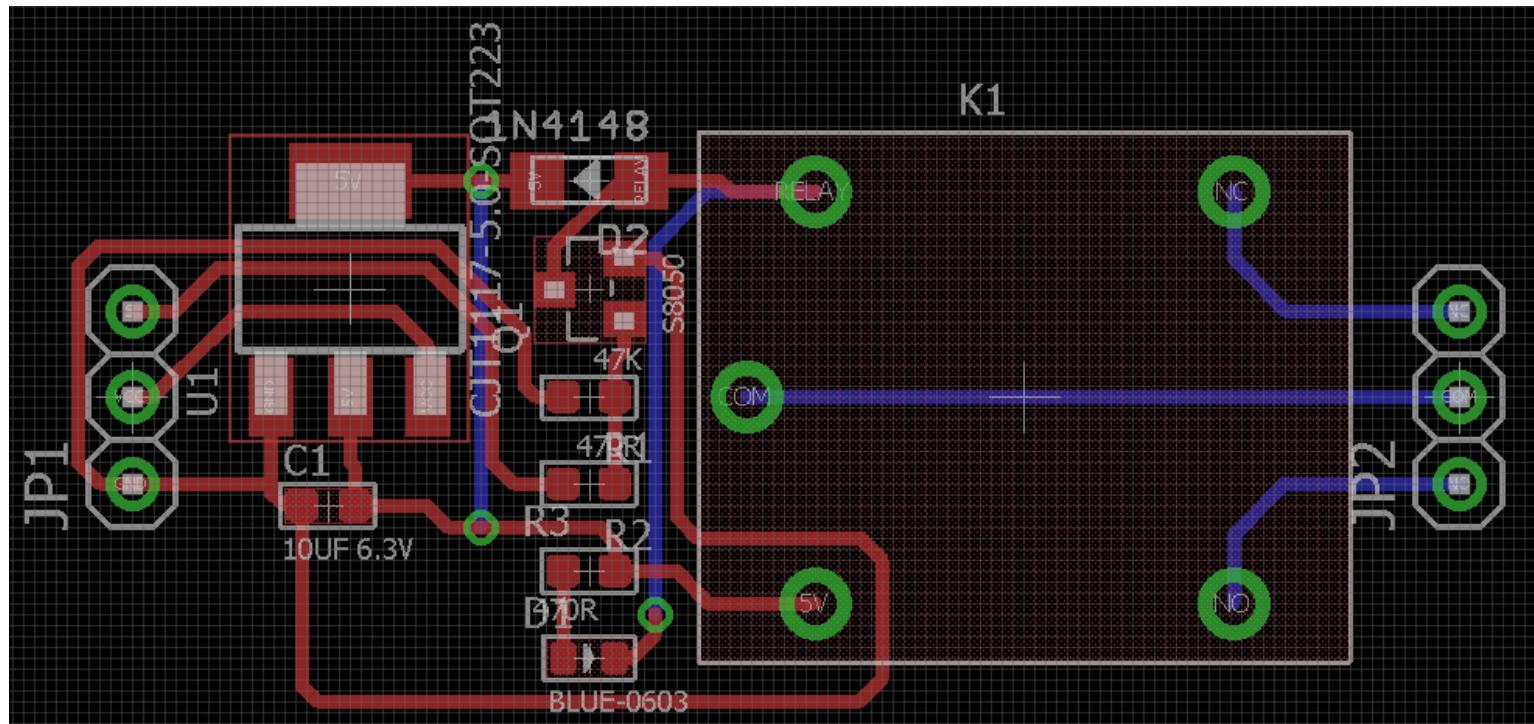


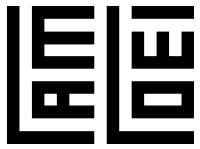
layer

Layers:	
Nr	Name
1	Top
16	Bottom
17	Pads
18	Vias
19	Unrouted
20	Dimension
21	tPlace
22	bPlace
23	tOrigins
24	bOrigins
25	tNames
26	bNames
27	tValues
28	bValues
29	tStop
30	bStop
31	tCream
32	bCream
33	tFinish
34	bFinish
35	tGlue
36	bGlue
37	tTest
38	bTest
39	tKeepout
40	bKeepout
41	tRestrict
42	bRestrict
43	vRestrict
44	Drills
45	Holes
46	Milling
47	Measures
48	Document
49	Reference
51	tDocu
52	bDocu

RAM
DEI

ลากเส้น Route





ตรวจสอบการครบถ้วน length.ulp

File Edit Draw View Tools LIP

New Ctrl+N
Open... Ctrl+O
Open recent ▾
Save Ctrl+S
Save as...
Save all
Print setup...
Print... Ctrl+P
CAM Processor...
Switch to schematic
Import ▾
Export ▾
SCR Execute Script...
Run ULP... **Run ULP...**
Close Ctrl+F4
Exit Alt+X

Run

bom.ulp cam2dxf.ulp cam2image.ulp
cam2print.ulp centroid-screamingcircuits-smclul
change-pad-in-lbr.ulp

Eagle: length.ulp

This ULP calculates the signal length of routed tracks in the layout
run length [name | name* | *name | *name*]
run length name [name name ...]
EXAMPLE:
run length +D -D
run length d0 d2 d7 A*
run length d*
run length *
Wires in layers 1 to 16 will be added, airwires will be shown separately.
If net names are specified, the differences in length will be calculated in procentual values.
The procentual difference is based on the shortest of the specified tracks which is taken as 100%.
Parallel tracks and polygons are not taken into consideration.
Only for all signals it is possible to change the sorting of the displayed signals in the list.
Author: support@cadsoft.de

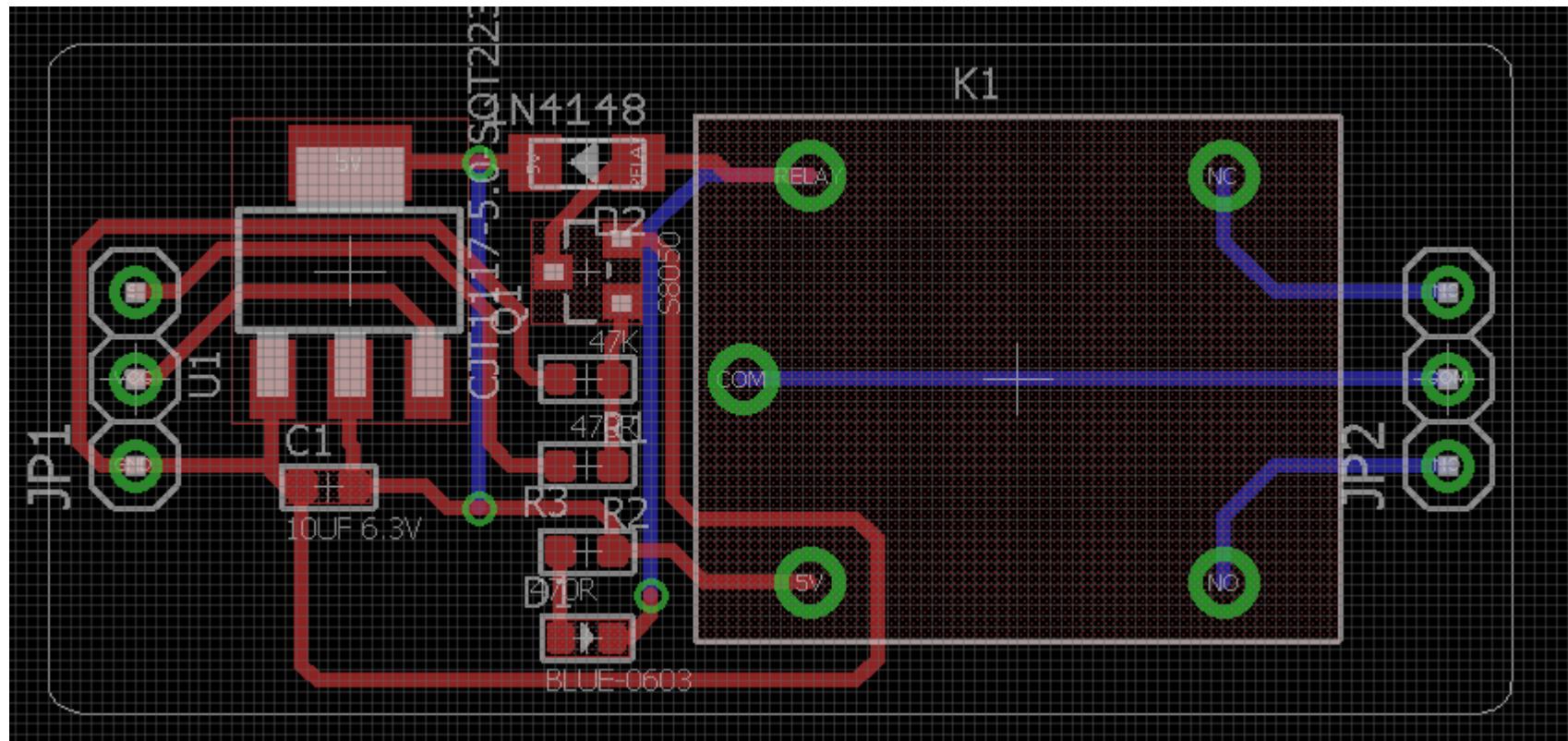
Eagle: Wire length of Layout

Signal	l [mm]	diff. [mm]	diff. [%]	unrouted [mm]
5V	33.960325140	31.420325140	1237.021	--
COM	20.800000000	18.260000000	718.898	--
GND	74.133899662	71.593899662	2818.657	--
NS2	2.540000000	0.000000000	0.000	--
NS3	4.931210245	2.391210245	94.142	--
NC	9.169604615	6.629604615	261.008	--
NO	9.169604615	6.629604615	261.008	--
RELAY	27.957981846	25.417981846	1000.708	--
S	17.961153673	15.421153673	607.132	--
VCC	11.632943428	9.092943428	357.990	--

Ok Save Help

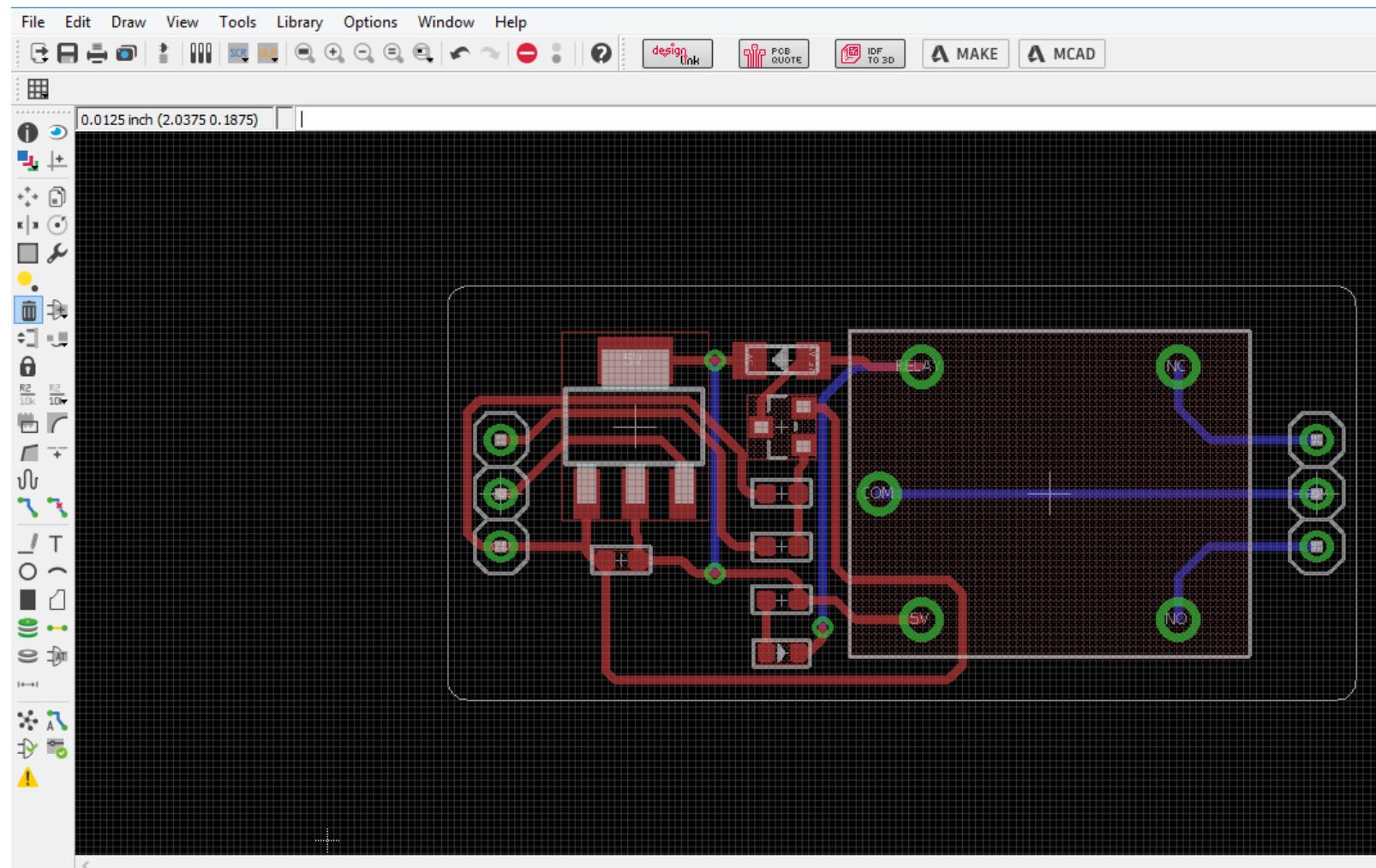


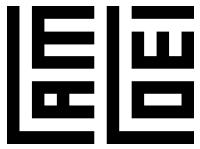
Lab 7 ทำข้อมูลร์ด



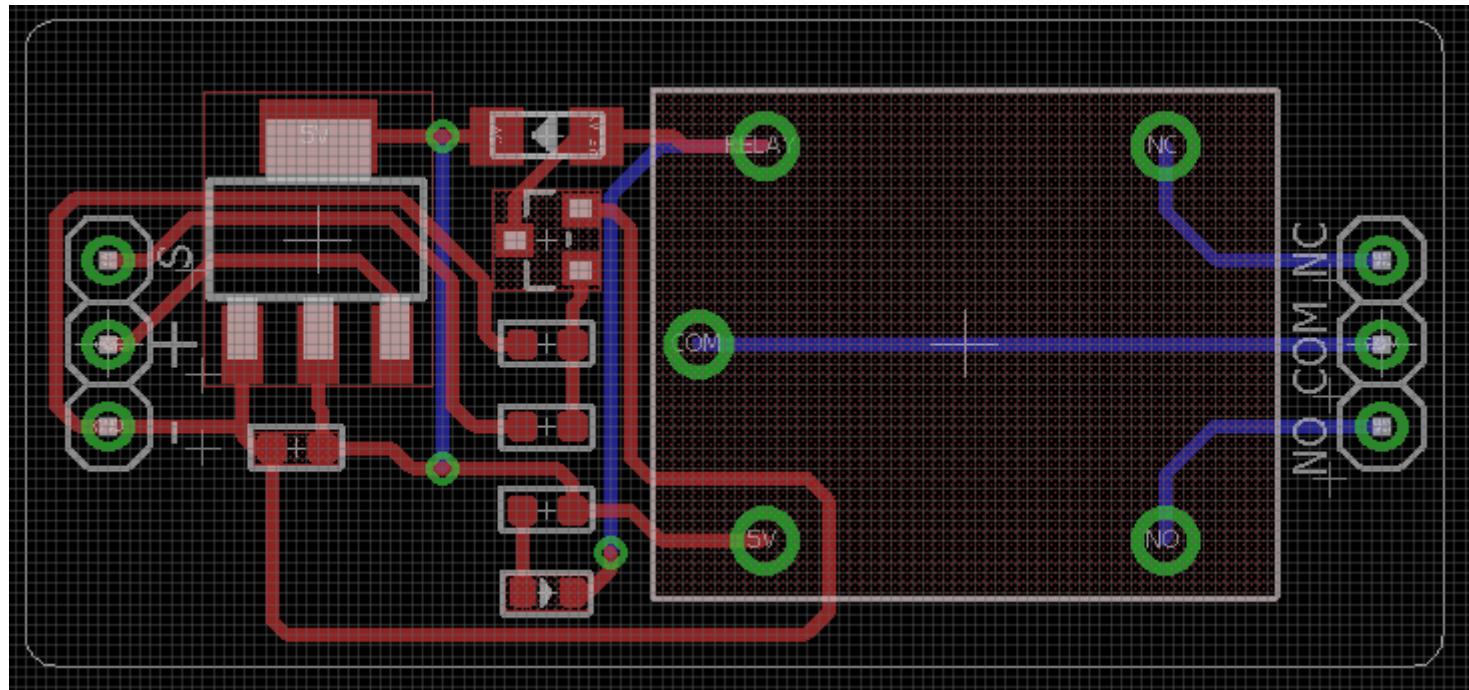


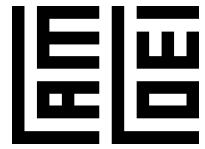
Lab 8 ลบส่วนไม่ใช้งาน



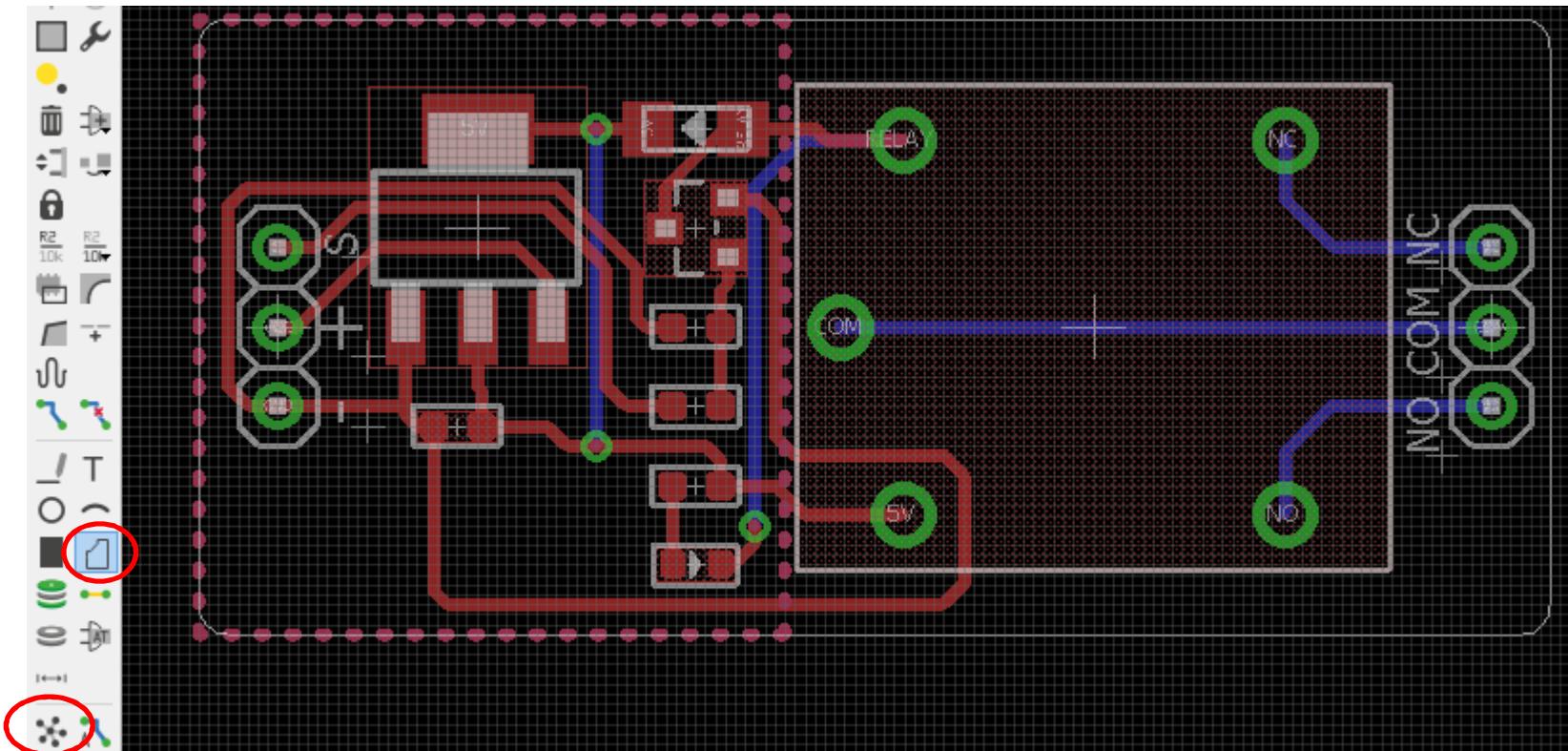


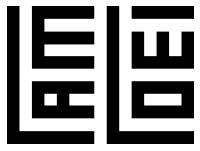
ลงรายละเอียด





ໃສ່ Ground Plane





ตรวจ DRU oshpark-2layer.dru

The screenshot shows the Eagle DRC (Design Rule Check) interface. A red circle highlights the 'Limit' input field in the 'Masks' tab of the DRC dialog, which contains the value '50mil'. Below the dialog, a message in Thai reads: 'ต้องไม่มี Error จึงถูก' (There must be no errors to be correct).

DRC (oshpark-2layer)

File Layers Clearance Distance Sizes Restring Shapes Supply Masks Misc

Mask values are defined in percent of the smaller dimension of smds, pads and vias (limited by Min and Max).
Stop masks are generated for smds, pads and those vias that have a drill diameter that exceeds Limit.
Cream masks are generated for smds only.

Check Select Cancel Apply



Lab 9 Gen CAM file

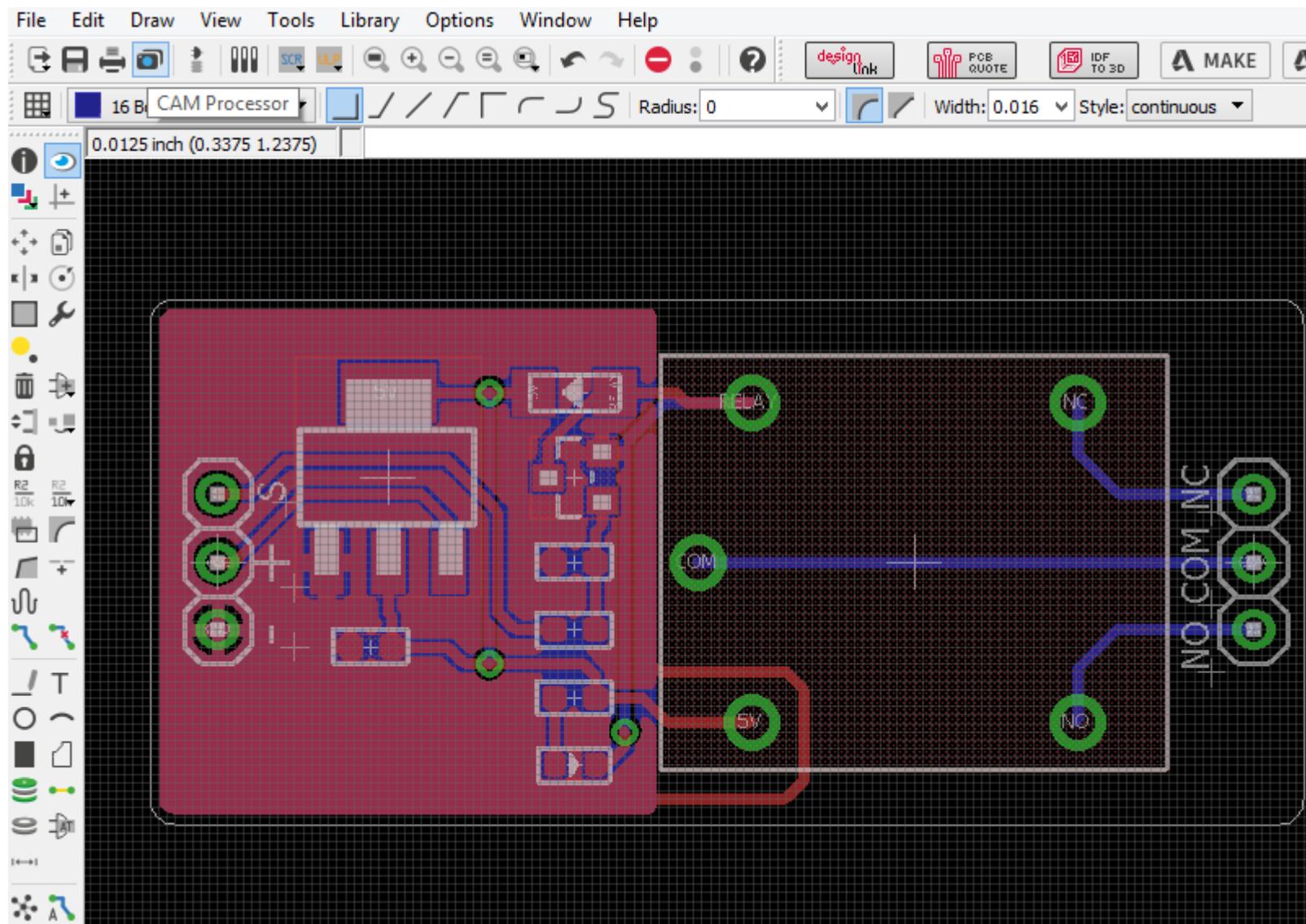
PC > Documents > eagle > lab2-relay_shield > relay_shield > relay_shield

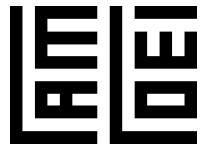
Name	Date modified	Type	Size
relay_shield.brd.lck	30/6/2560 15:47	LCK File	1 KB
relay_shield.sch.lck	30/6/2560 15:47	LCK File	1 KB
eagle.epf	30/6/2560 15:16	Exchange Certifica...	18 KB
relay_shield.b#1	30/6/2560 19:43	B#1 File	40 KB
relay_shield.b#2	30/6/2560 19:28	B#2 File	54 KB
relay_shield.b#3	30/6/2560 19:23	B#3 File	53 KB
relay_shield.b#4	30/6/2560 19:00	B#4 File	49 KB
relay_shield.b#5	30/6/2560 16:31	B#5 File	46 KB
relay_shield.b#6	30/6/2560 16:23	B#6 File	46 KB
relay_shield.b#7	30/6/2560 15:43	B#7 File	21 KB
relay_shield.b#8	30/6/2560 1:01	B#8 File	12 KB
relay_shield.brd	30/6/2560 19:49	BRD File	41 KB
relay_shield.s#1	30/6/2560 19:29	S#1 File	123 KB
relay_shield.s#2	30/6/2560 17:23	S#2 File	112 KB
relay_shield.s#3	30/6/2560 15:43	S#3 File	16 KB
relay_shield.sch	30/6/2560 19:50	SCH File	52 KB

ไฟล์เดอร์ก่อนสร้าง CAM File

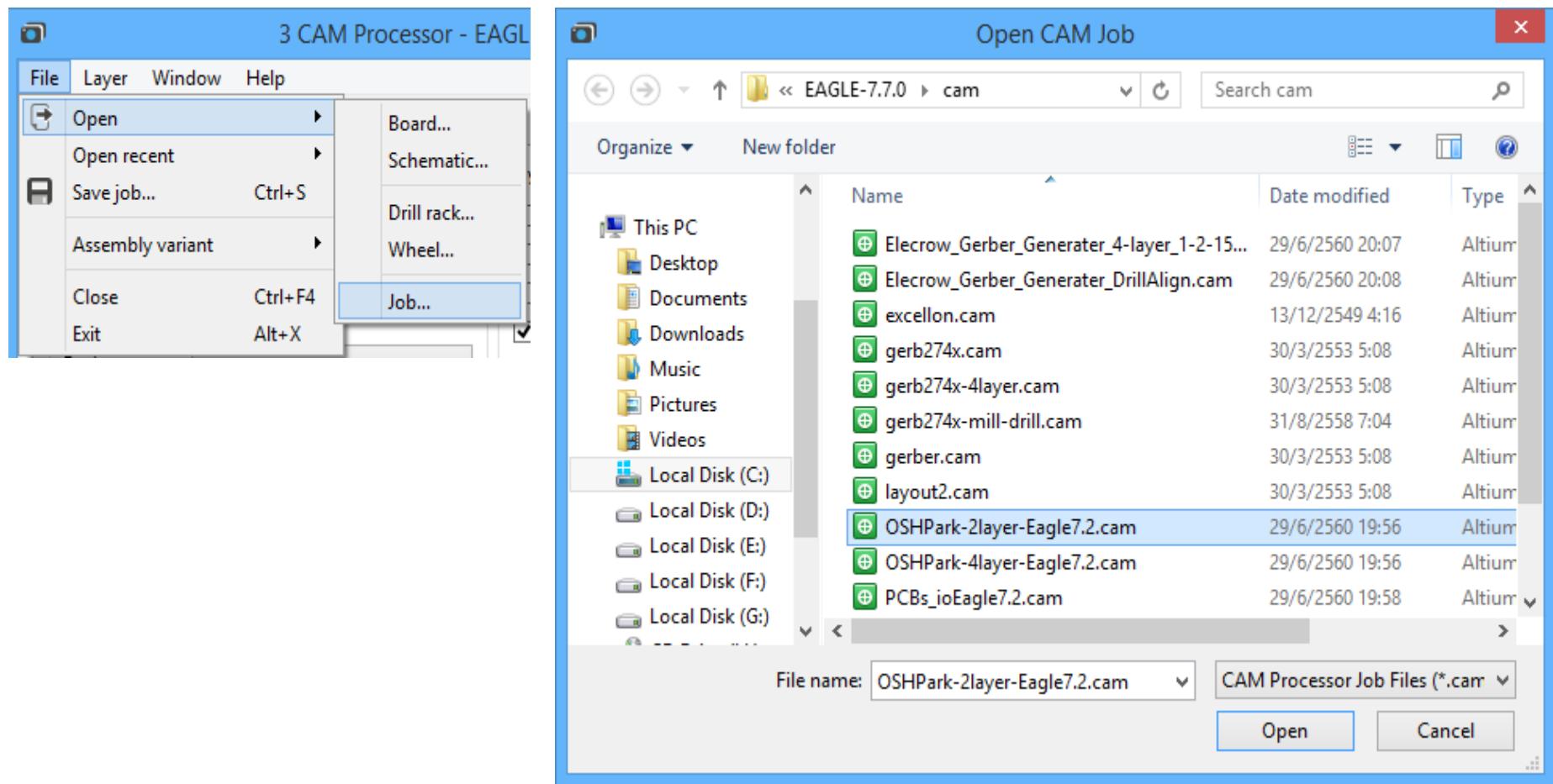


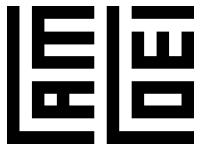
คลิกปุ่ม CAM Processor



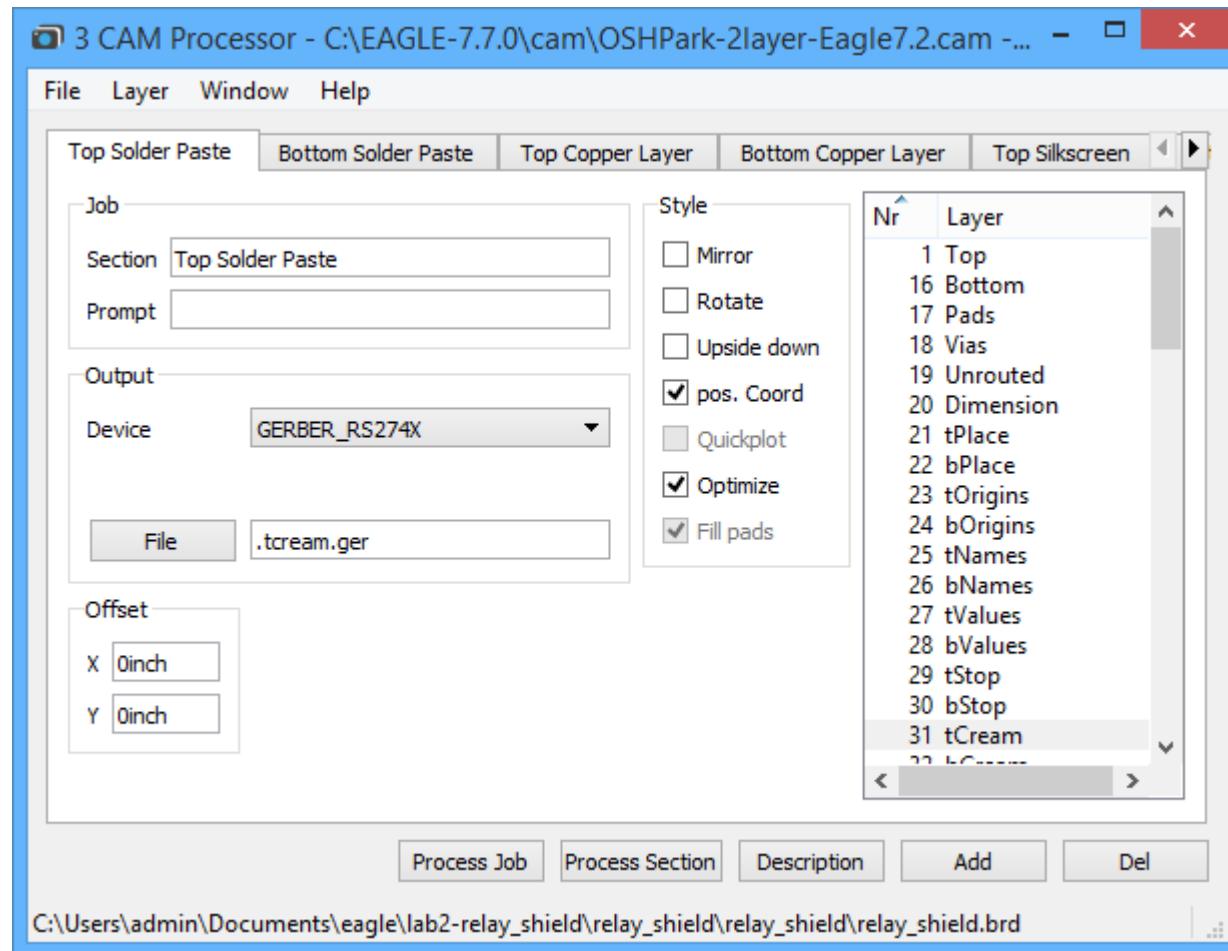


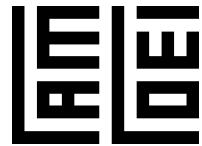
File > Open > Job...





Process Job





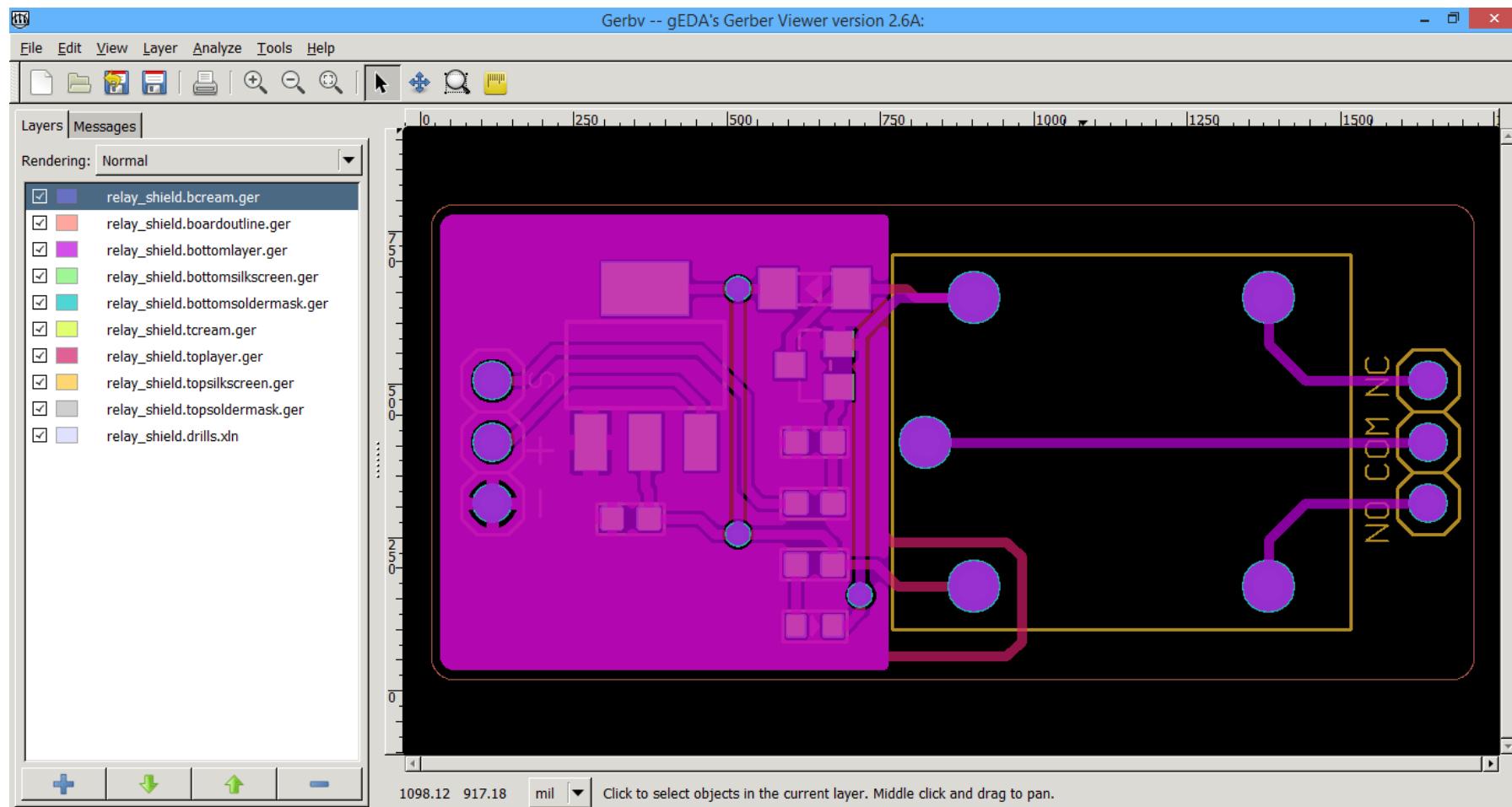
CAM File

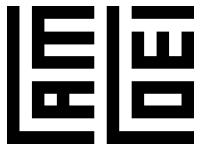
PC ▶ Documents ▶ eagle ▶ lab2-relay_shield ▶ relay_shield ▶ relay_shield

Name	Date modified	Type	Size
relay_shield.bcream.ger	30/6/2560 19:58	GER File	1 KB
relay_shield.bcream.gpi	30/6/2560 19:58	GPI File	1 KB
relay_shield.boardoutline.ger	30/6/2560 19:58	GER File	5 KB
relay_shield.boardoutline.gpi	30/6/2560 19:58	GPI File	2 KB
relay_shield.bottomlayer.ger	30/6/2560 19:58	GER File	1 KB
relay_shield.bottomlayer.gpi	30/6/2560 19:58	GPI File	2 KB
relay_shield.bottomsilkscreen.ger	30/6/2560 19:58	GER File	1 KB
relay_shield.bottomsilkscreen.gpi	30/6/2560 19:58	GPI File	1 KB
relay_shield.bottomsoldermask.ger	30/6/2560 19:58	GER File	1 KB
relay_shield.bottomsoldermask.gpi	30/6/2560 19:58	GPI File	2 KB
relay_shield.drills.dri	30/6/2560 19:58	DRI File	1 KB
relay_shield.drills.xln	30/6/2560 19:58	XLN File	1 KB
relay_shield.tcream.ger	30/6/2560 19:58	GER File	3 KB
relay_shield.tcream.gpi	30/6/2560 19:58	GPI File	2 KB
relay_shield.toplayer.ger	30/6/2560 19:58	GER File	6 KB
relay_shield.toplayer.gpi	30/6/2560 19:58	GPI File	2 KB
relay_shield.topsilkscreen.ger	30/6/2560 19:58	GER File	5 KB
relay_shield.topsilkscreen.gpi	30/6/2560 19:58	GPI File	2 KB
relay_shield.topsoldermask.ger	30/6/2560 19:58	GER File	4 KB
relay_shield.topsoldermask.gpi	30/6/2560 19:58	GPI File	2 KB
relay_shield.sch	30/6/2560 19:50	SCH File	52 KB
relay_shield.brd	30/6/2560 19:49	BRD File	41 KB
relay_shield.b#1	30/6/2560 19:43	B#1 File	40 KB
relay_shield.s#1	30/6/2560 19:29	S#1 File	123 KB
33.5 KB			

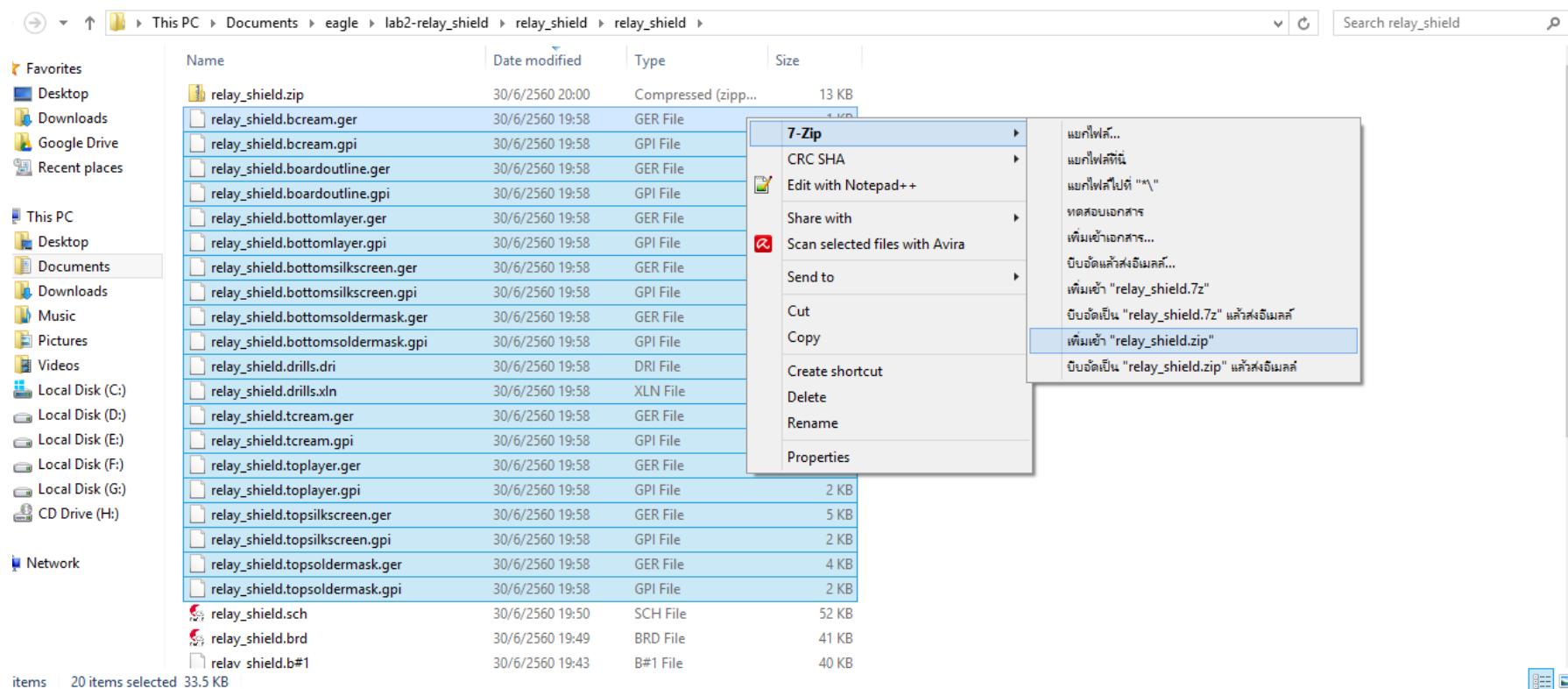


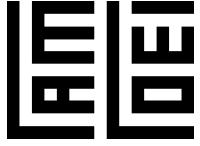
ตรวจด้วย Gerbv





Zip cam file





OSHPark
Verify your design

Board Top

This shows the final manufactured board as if you held it in your hand.

Your design should show gold copper, purple mask, white silk, black drills, and the board outline. Internal cutouts are indicated by a black outline but are not filled in.

If the image here is entirely white, you want to find and fix any gaps in the board outline.

There should be no dimensions or measurement rules.

Board Bottom

This shows the final manufactured board as if you held it in your hand.

Your design should show gold copper, purple mask, white silk, black drills, and the board outline.

Internal cutouts are indicated by a black outline but are not filled in.

If the image here is entirely white, you want to find and fix any gaps in the board outline.

There should be no dimensions or measurement rules.

Drills

Drills should show up as white circles on a purple background.

We will merge all the expected in Excellon or NC Drill format with 2.4 precision, absolute coordinates, and no zero suppression.

We will only place the holes that pass through copper.

Drills smaller than our minimums will be increased back to the minimum size in the design rules:

- No overlapping drill holes.
- No blind or buried holes.
- Small arcs and plated holes are not supported.
- Drills must be circular.
- Holes can't be oval or other non-circular drills.

We have more info on our [drills help page](#).

Top Silk Screen

We will ignore the portion of the silkscreen that crosses over the board outline.

We will automatically remove any silkscreen that crosses outside holes or exposed copper.

If a layer isn't showing up on this layer, try changing your gerber file extension to .gbr. If you still don't see the image with 400 DPI or less, or check out our [Silkscreen render tips page](#).

Bottom Layer

This layer should appear "inverted" as if you were looking down at it through the board from the top.

If you are using Altium Designer or Altium Circuits, carefully examine the board outline layer being included in this layer. See [litzz for more](#).

If you are using Eagle, be aware that areas may not appear the same as in most traces. If there are no differences, then the layer is correct. If there are, please review your .litzz file for errors.

See our [design tools page](#) for more.

Top Solder Mask

Soldermask layers show us where to remove the purple solder resist. The gold-colored areas will be covered with purple solder mask.

If you submit an empty area, the entire board will be covered in purple soldermask.

To expose the entire board, submit this file as a single polygon that covers the entire board. We will remove all mask everywhere and expose all the copper and board substrate.

Bottom Solder Mask

This layer should appear inverted as if you were looking down at it through the board from the top.

Soldermask layers show us where to remove the green solder resist. The gold-colored areas will be exposed on the final board, and purple areas will be covered with green soldermask.

To expose the entire board, submit this file as a single polygon that covers the entire board. We will remove all mask everywhere and expose all the copper and board substrate.

Board Outline

The board outline should be a wavyline (no gaps) purple outline showing at least the edge of the board's footprint.

The outline layer can also show large off-holes and holes with irregular board shapes, but they will be listed in the smaller rectangle to avoid confusion. For example, a hole with a two-inch diameter would cost the same as a square of four square inches.

We will only accept octagons or rounded rectangles, but they usually come out okay if you follow the [values and sizes instructions](#).

Top Layer

We will place copper everywhere we see gold color in the gerber files.

If you are using Altium Designer or Altium Circuits, carefully examine the board outline layer being included in this layer. See [litzz for more](#).

If you are using Eagle, be aware that the areas are not necessarily the same as in most traces. Copper links between pads showing on this layer, copper links between pads showing on the layer above, and so on.

See our [design tools page](#) for more.

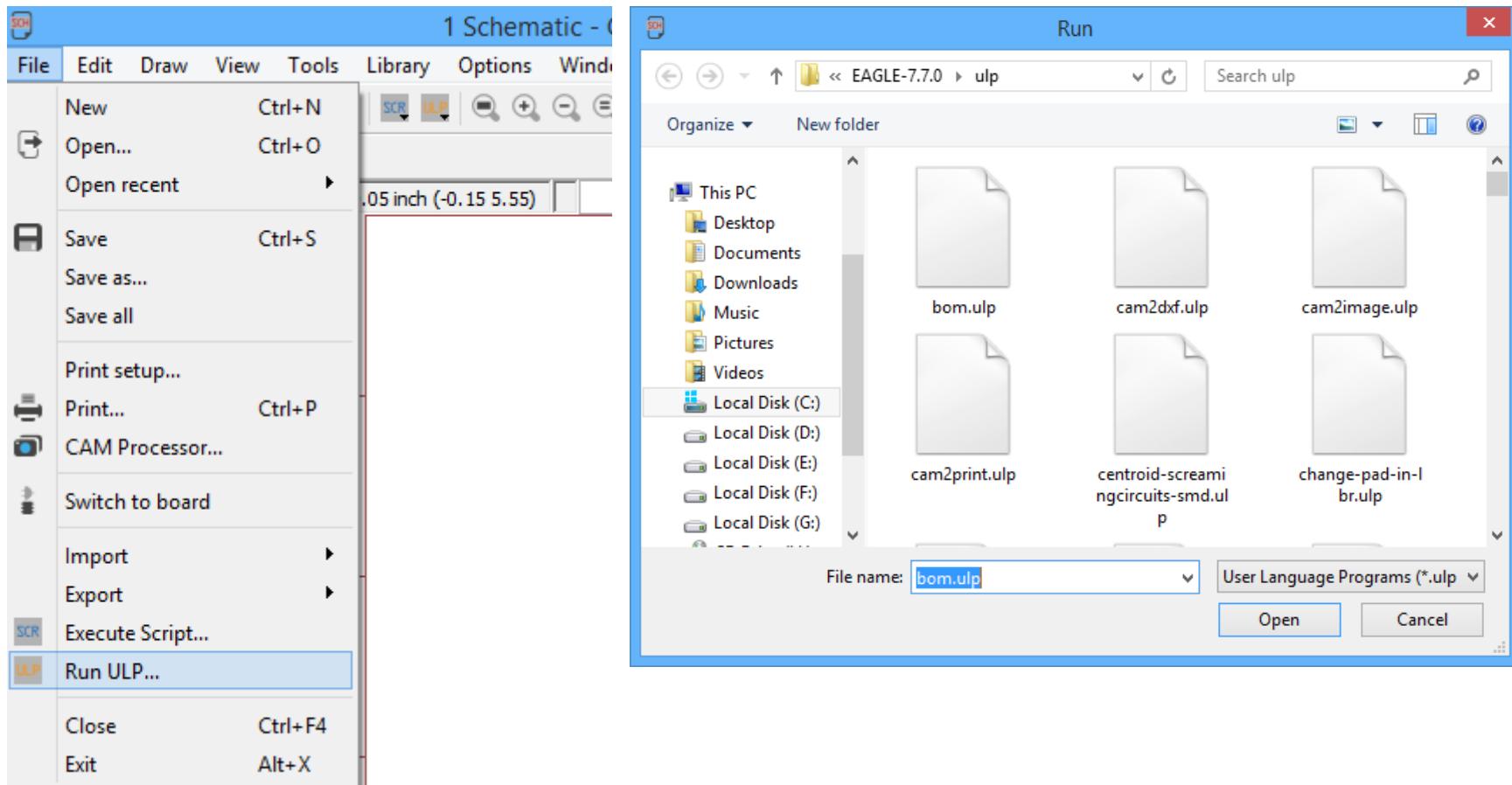
Start Over Approve Approve and Order

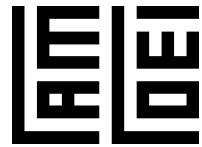
Designed and generated by Fabu

Lab 10 Check oshpark



Lab 11 BOM File





เลือก CSV แล้ว Save

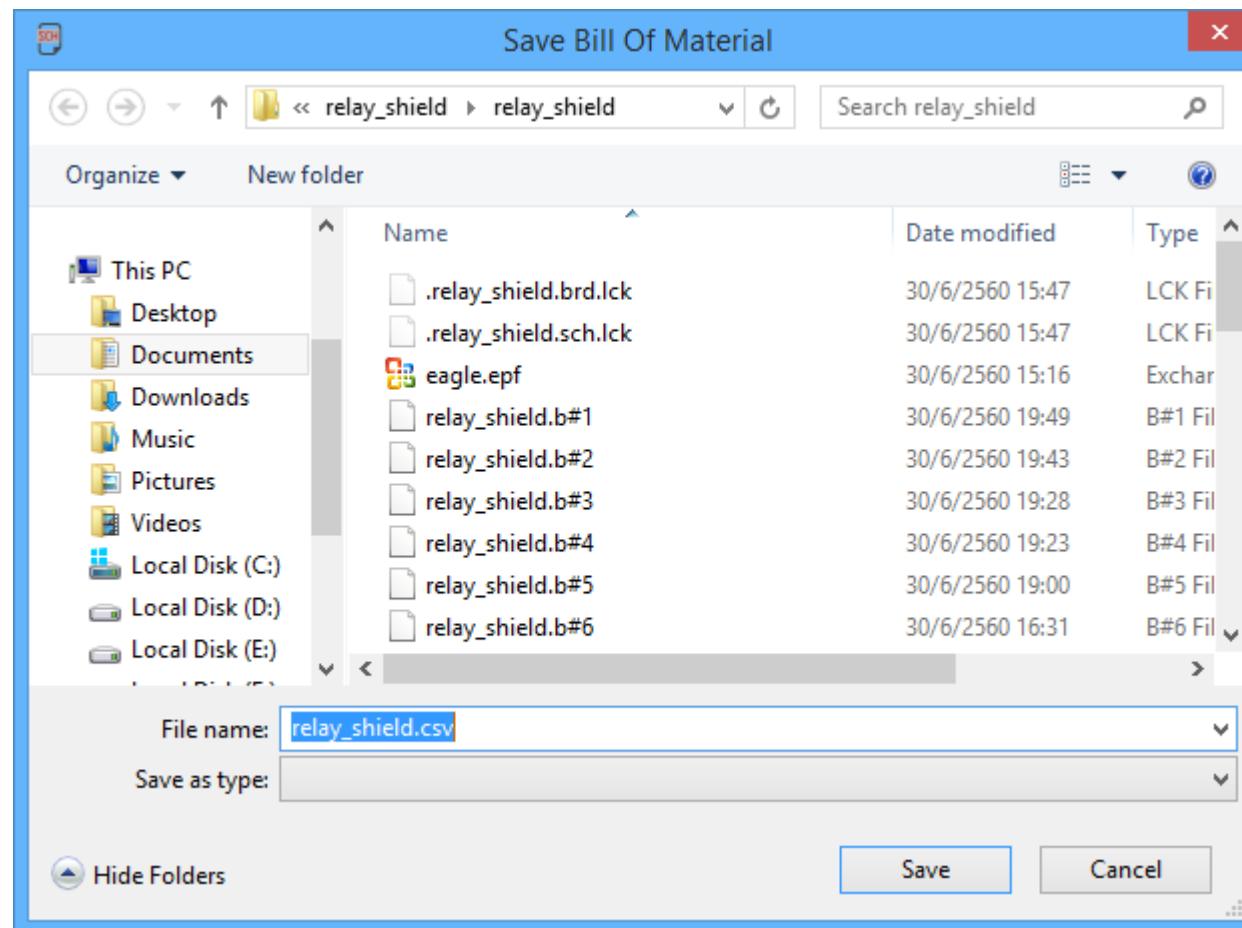
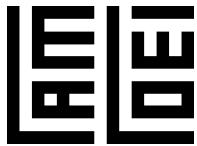
Eagle: Bill Of Material

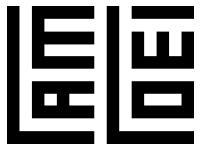
Part	Value	Device	Package	Description	MPN	VALUE
C1	10UF 6.3V	CERAMIC-10UF-6.3V-20%-X5R(0603)	C0603	302010286	CC0603MRX5R5BB106	10UF 6.3V
D1	BLUE-0603	SMD-LED-CLEAR-BLUE(0603)	LED-0603	304090045	19-217-BHC-ZL1M2RY-3T	BLUE-0603
D2	1N4148	SMD-DIODE-SWITCH-1N4148(LL-34)	LL-34	304040016	1N4148	1N4148
JP1		PINHD-1X3	1X03		PIN HEADER	
JP2		PINHD-1X3	1X03		PIN HEADER	
K1		RELAY-HLS8L-DC5V-S-CHLS8L-DC5V-S-C	REY5-19.0X15.0X15.0MM	315030000		
Q1	S8050	SMD-TRANSISTORS-NPN-25V-500MW-S8050(SOT-23)	SOT-23	305010017	S8050	S8050
R1	470R	SMD-RES-470R-5%-1/10W(0603)	R0603	301010088	RC0603JR-07470RL	470R
R2	470R	SMD-RES-470R-5%-1/10W(0603)	R0603	301010088	RC0603JR-07470RL	470R

List type Output format

Parts Text
 Values CSV
 List attributes HTML

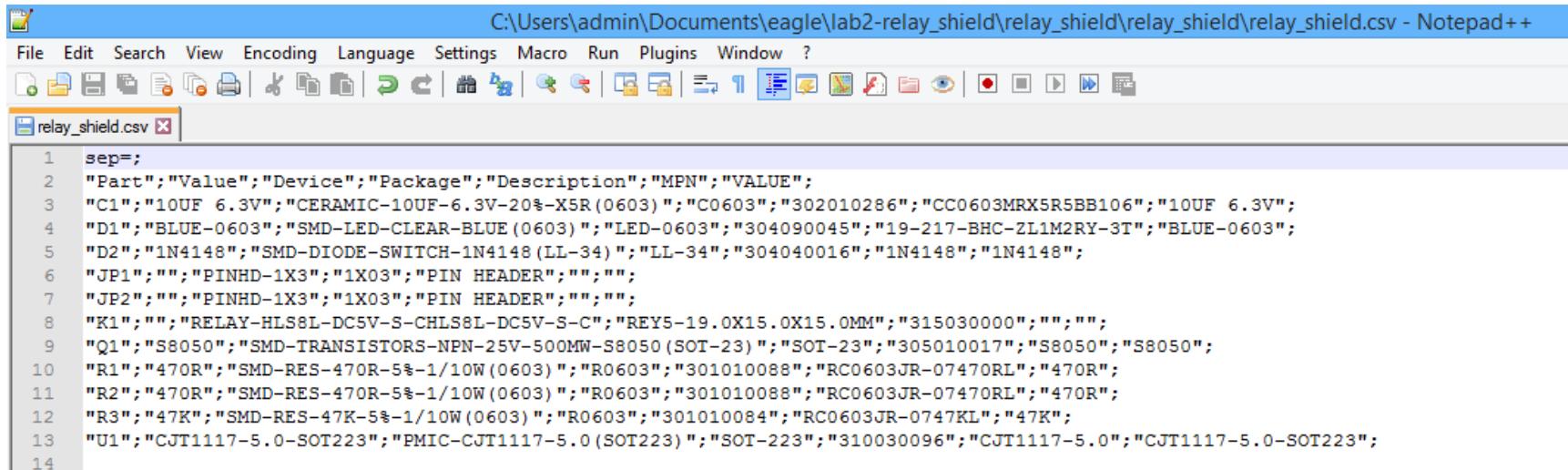
View Save... Help Close Version 1.11



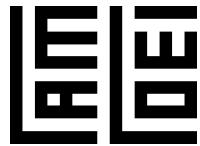


sep=;

- เปิดไฟล์ csv ด้วย Notepad
- พิมพ์ sep=; ในบรรทัดแรก



```
C:\Users\admin\Documents\eagle\lab2-relay_shield\relay_shield\relay_shield.csv - Notepad++
File Edit Search View Encoding Language Settings Macro Run Plugins Window ?
relay_shield.csv x
1 sep=;
2 "Part";"Value";"Device";"Package";"Description";"MPN";"VALUE";
3 "C1";"10UF 6.3V";"CERAMIC-10UF-6.3V-20%-X5R (0603)";"C0603";"302010286";"CC0603MRX5R5BB106";"10UF 6.3V";
4 "D1";"BLUE-0603";"SMD-LED-CLEAR-BLUE (0603)";"LED-0603";"304090045";"19-217-BHC-ZL1M2RY-3T";"BLUE-0603";
5 "D2";"1N4148";"SMD-DIODE-SWITCH-1N4148 (LL-34)";"LL-34";"304040016";"1N4148";"1N4148";
6 "JP1";"";"PINHD-1X3";"1X03";"PIN HEADER";"";;
7 "JP2";"";"PINHD-1X3";"1X03";"PIN HEADER";"";;
8 "K1";"";"RELAY-HLS8L-DC5V-S-CHLS8L-DC5V-S-C";"REY5-19.0X15.0X15.0MM";"315030000";"";;
9 "Q1";"S8050";"SMD-TRANSISTORS-NPN-25V-500MW-S8050 (SOT-23)";"SOT-23";"305010017";"S8050";"S8050";
10 "R1";"470R";"SMD-RES-470R-5%-1/10W (0603)";"R0603";"301010088";"RC0603JR-07470RL";"470R";
11 "R2";"470R";"SMD-RES-470R-5%-1/10W (0603)";"R0603";"301010088";"RC0603JR-07470RL";"470R";
12 "R3";"47K";"SMD-RES-47K-5%-1/10W (0603)";"R0603";"301010084";"RC0603JR-0747KL";"47K";
13 "U1";"CJT1117-5.0-SOT223";"PMIC-CJT1117-5.0 (SOT223)";"SOT-223";"310030096";"CJT1117-5.0";"CJT1117-5.0-SOT223";
14
```



เปิด CSV ด้วย Excel



Lab12 เปิด bom template

The screenshot shows two Excel spreadsheets side-by-side. The left spreadsheet, titled 'bom.xlsx', has a header row with 'Name' in bold. Below it, there are 12 rows of data with columns for 'Item', 'Quantity', 'Reference', and 'Part'. The right spreadsheet, titled 'relay_shield.csv', has a header row with 'Part', 'Value', 'Device', 'Package', 'Description', 'MPN', and 'VALUE'. It contains 12 rows of component details. A red circle highlights the status bar at the bottom of the screen, which displays 'ค่าเฉลี่ย: 1.222222222 จำนวน: 9 ผลรวม: 11'.

Name			
Item	Quantity	Reference	Part
1	1	C1	
2	1	D1	
3	1	D2	
4	2	JP1, JP2	
5	1	K1	
6	1	Q1	
7	2	R1, R2	
8	1	R3	
9	1	U1	
13			+/-
14			
15			
16			
17			
18			
19			
20			
21			
22			
23			
24			
25			

A	B	C	D	E	F	G
1	Part	Value	Device	Package	Description	MPN
2	C1	10UF	6.3V CERAMIC	-C0603	3.02E+08	CC0603MR10UF 6.3V
3	D1	BLUE-0603	SMD-LED-C	LED-0603	3.04E+08	19-217-BH BLUE-0603
4	D2	1N4148	SMD-DIODE	LL-34	3.04E+08	1N4148 1N4148
5	JP1		PINHD-1X	1X03	PIN HEADER	
6	JP2		PINHD-1X	1X03	PIN HEADER	
7	K1		RELAY-HL	REY5-19.0	3.15E+08	
8	Q1	S8050	SMD-TRANSISTOR	-23	3.05E+08	S8050 S8050
9	R1	470R	SMD-RESISTOR	-R0603	3.01E+08	RC0603JR-470R
10	R2	470R	SMD-RESISTOR	-R0603	3.01E+08	RC0603JR-470R
11	R3	47K	SMD-RESISTOR	-R0603	3.01E+08	RC0603JR-47K
12	U1	CJT1117-5PMIC-CJT1	SOT-223		3.1E+08	CJT1117-5 CJT1117-5-S
13						
14						
15						
16						
17						
18						
19						
20						
21						
22						
23						
24						

ค่าเฉลี่ย: 1.222222222 จำนวน: 9 ผลรวม: 11



Lab13 ค้นหา bom

Contact Mouser (Bangkok) +66 2694-2310 | [Feedback](#)

 MOUSER ELECTRONICS

All ▾ Part No. / Keyword

Stocked RoHS Compliant

Products Manufacturers Applications Services & Tools Catalogue Help

All Products > Passive Components > Capacitors > MLCCs > Multilayer Ceramic Capacitors MLCC - SMD/SMT > Murata Electronics GRM188R60J106ME47J

[See an Error?](#)

 **Mouser Part No:** 81-GRM188R60J106ME7J
Manufacturer Part No: GRM188R60J106ME47J
Manufacturer: Murata Electronics
Description: Multilayer Ceramic Capacitors MLCC - SMD/SMT 10uF 6.3Volts 20%
[Learn more about Murata Electronics GRM188R60J106ME47J](#)


[Enlarge](#)

Images are for reference only
See Product Specifications

[Add to Compare List](#) [Share](#) [Email](#) [Facebook](#) [G+](#) [0](#)

Real Time Availability

Stock:	65,396 Can Dispatch Immediately
On Order:	20000 View Delivery Dates
Factory Lead Time: 37 Weeks	

Enter Quantity: [Buy](#)
 Minimum: 1 Multiples: 1
[This Product Ships FREE](#)

Select Packaging Option Below

Pricing (USD)

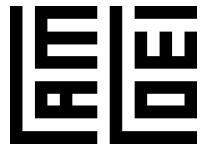
<input type="radio"/> Cut Tape	1: \$0.15
	10: \$0.105
	100: \$0.049
	500: \$0.035
	1,000: \$0.028
	2,500: \$0.024

MouseReel™ Reel Service Fee
\$7.00 Price will be calculated in basket.

Full Reels of 10000
To purchase full reel, order in multiples of 10000:

Specifications **Features** **Documents (1)** **My Notes**

Product Category:	Multilayer Ceramic Capacitors MLCC - SMD/SMT	<input checked="" type="checkbox"/>
Manufacturer:	Murata	<input type="checkbox"/>
RoHS:	 Details	<input type="checkbox"/>
Series:	GRM	<input type="checkbox"/>
Capacitance:	10 uF	<input type="checkbox"/>
Voltage Rating DC:	6.3 VDC	<input type="checkbox"/>
Dielectric:	X5R	<input type="checkbox"/>
Tolerance:	20 %	<input type="checkbox"/>
Case Code - in:	0603	<input type="checkbox"/>
Case Code - mm:	1608	<input type="checkbox"/>
Height:	0.8 mm	<input type="checkbox"/>

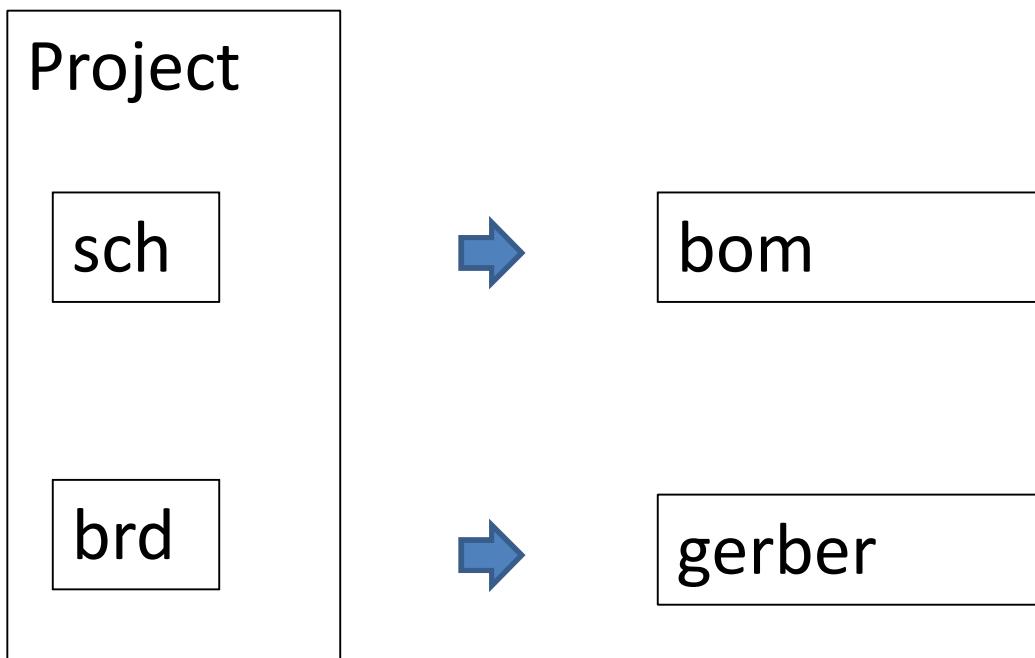


Bom.xlsx

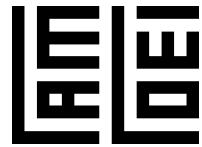
A	B	C	D	E	F	G	H	I
1		Relay Module			v1.0	2/9/2560		
Item	Quantity	Reference	Part	Footprint	Mfg	Mfg P/N	Vendor	Vendor P/N
1	1	C1	Ceramic Capacitors MLCC - SMD/SMT 10uF 6.3Volts 20%	0603	Murata Electronics	GRM188R60J106ME47J	Mouser	81-GRM188R60J106
2	1	D1	SMD Blue Clear 470nm	0603	Lite-On	LTST-C190TBKT	Mouser	859-LTST-C190
3	1	D2	General Purpose, Power, Switching 100V Io/200mA T/R	LL-34	Fairchild Semiconductor	FDLL4148	Mouser	512-FDLI
4	2	JP1, JP2						
5	1	K1	Relays Power PCB Relay SPDT Sealed 5VDC	SPDT (1 Form C)	Omron	G5LE-14-DC5	Mouser	653-G5LE-14
6	1	Q1	Bipolar Transistors - BJT 625mW, 25V, 1500mA	SOT-23-3	Micro Commercial Components (MCC)	MMSS8050-L-TP	Mouser	833-MMSS8050
7	2	R1, R2	SMD 470 OHM 1%	0603	Yageo	RC0603FR-07470RL	Mouser	603-RC0603FR-074
8	1	R3	SMD 47K OHM 1%	0603	Yageo	RC0201FR-0747KL	Mouser	603-RC0201FR-074
9	1	U1	LDO Voltage Regulators 5.0V 1A Positive	SOT-223-3	ON Semiconductor	NCP1117ST50T3G	Mouser	863-NCP1117ST5



สรุป



sch กับ brd ต้องเป็นชื่อเดียวกัน

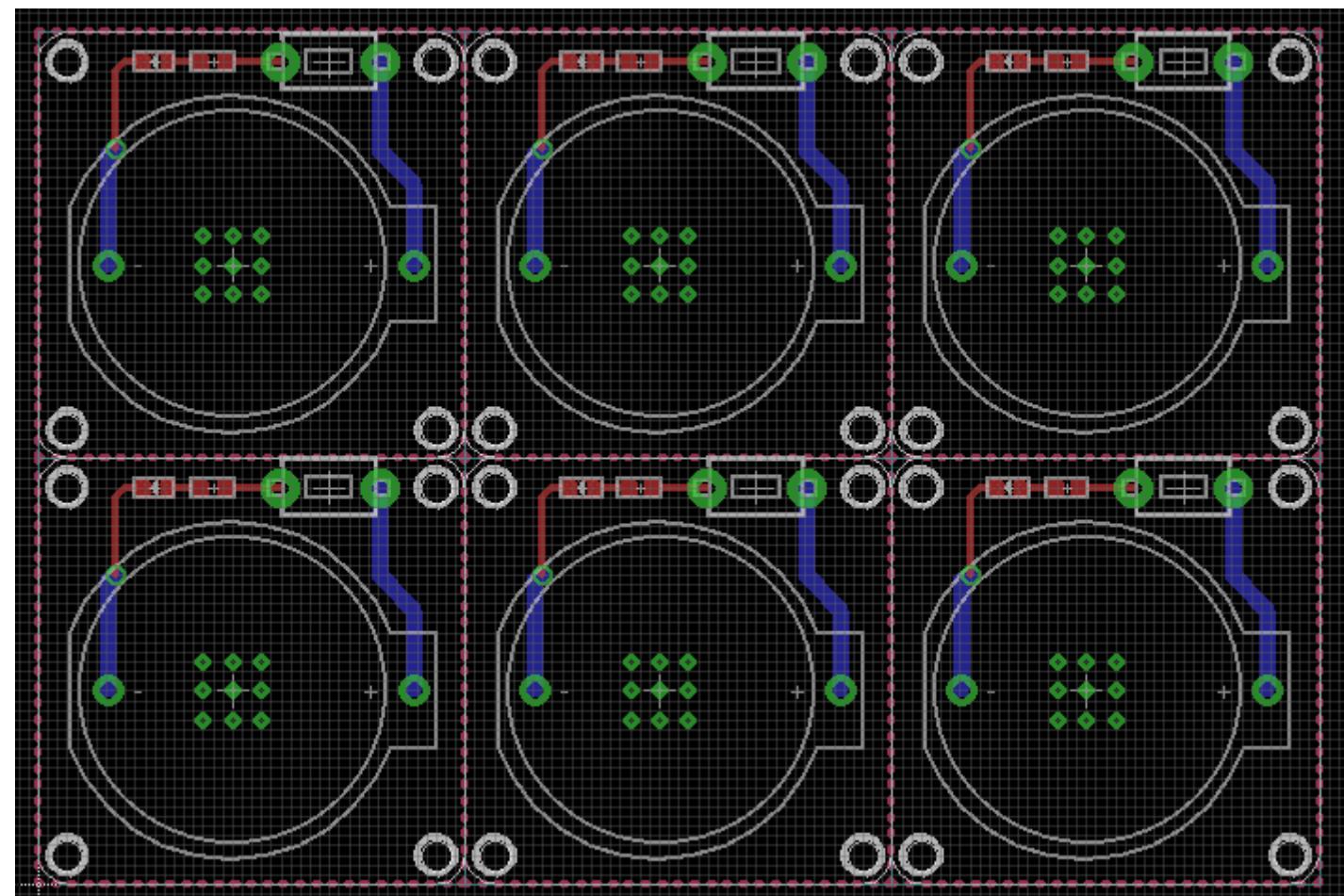


File > New

Edit > Paste

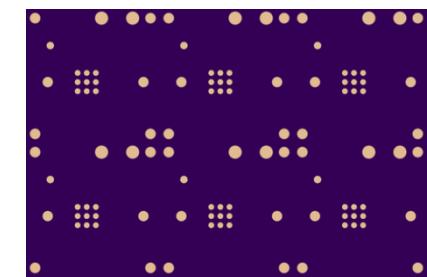
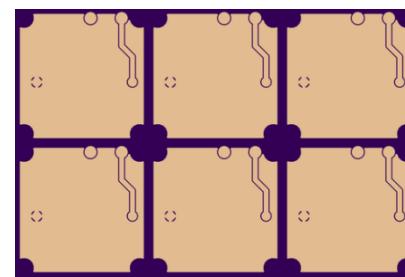
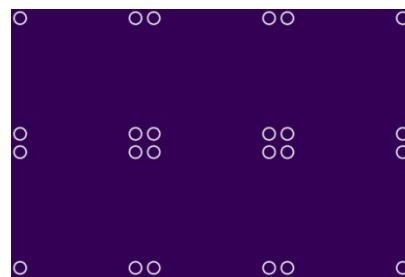
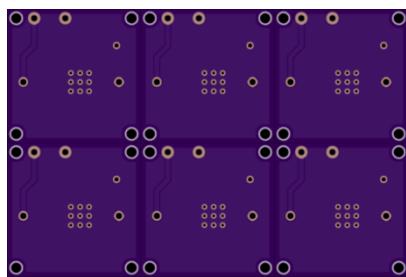
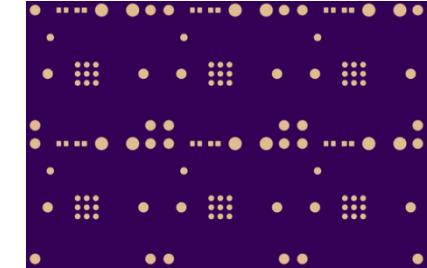
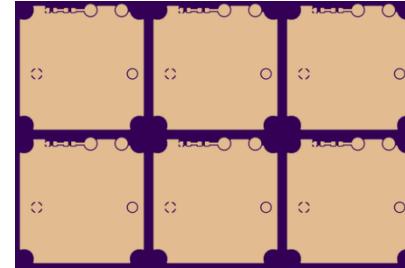
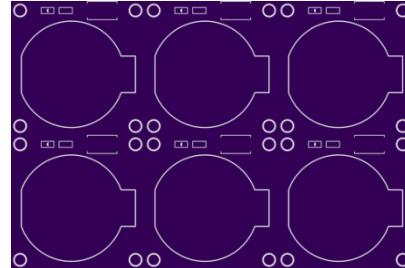
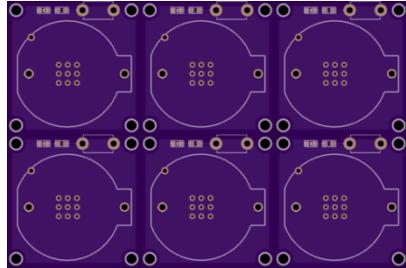
Wire > layer 46 width: 0 เพื่อแบ่งช่อง

แล้วบันทึก switch2032-panel.brd

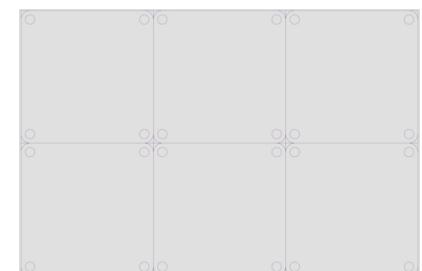
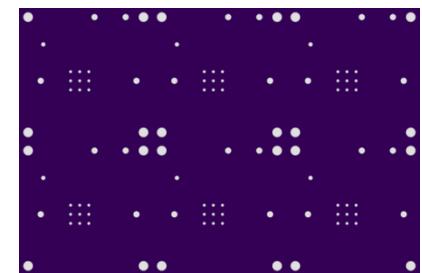


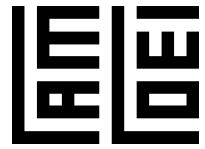


สร้าง gerber ใหม่



ให้คำนวณ cost ของ panel





Lab 5 เพิ่ม Logo

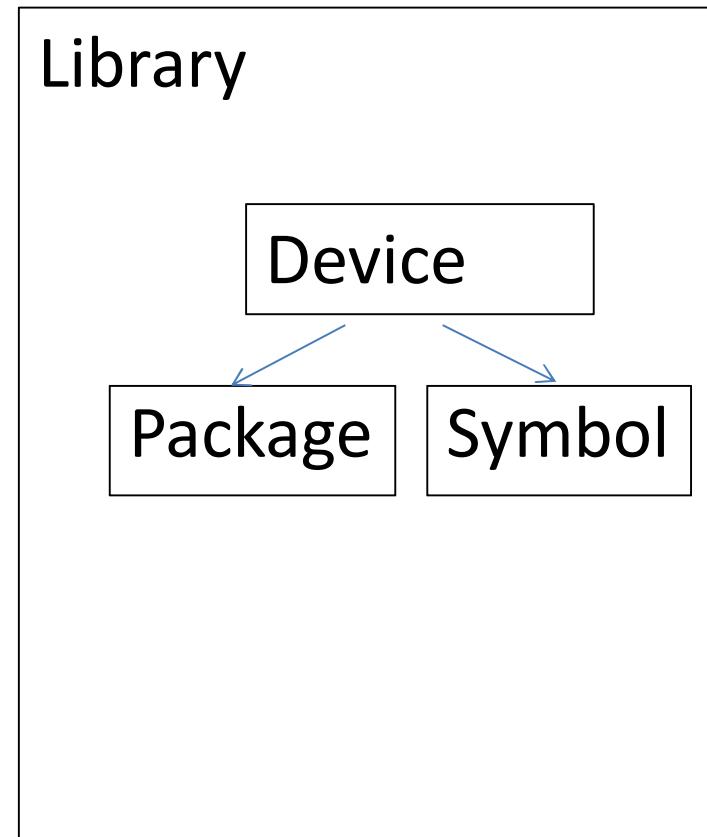
เตรียมภาพนามสกุล **bmp** แบบ 2 สี ขาวดำ

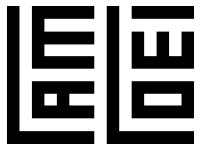




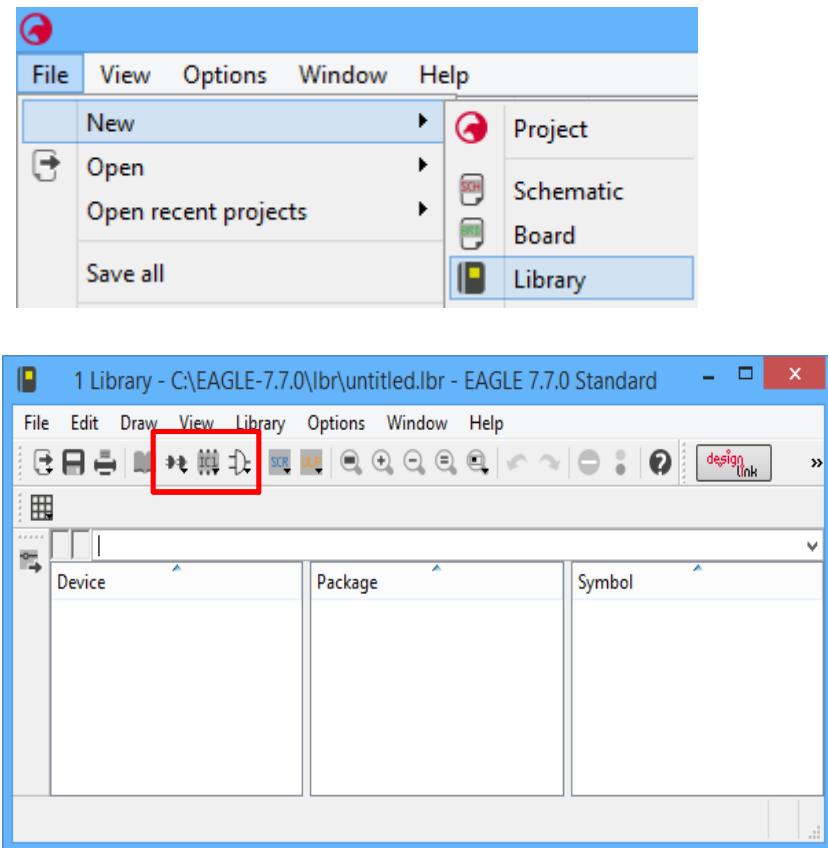
Library

- Device > Lib
- Package > Brd
- Symbol > Sch





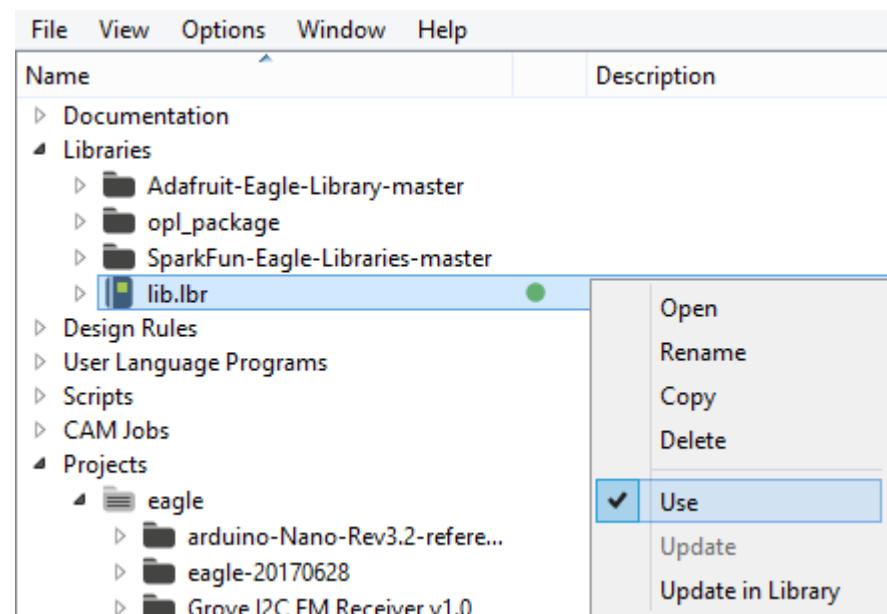
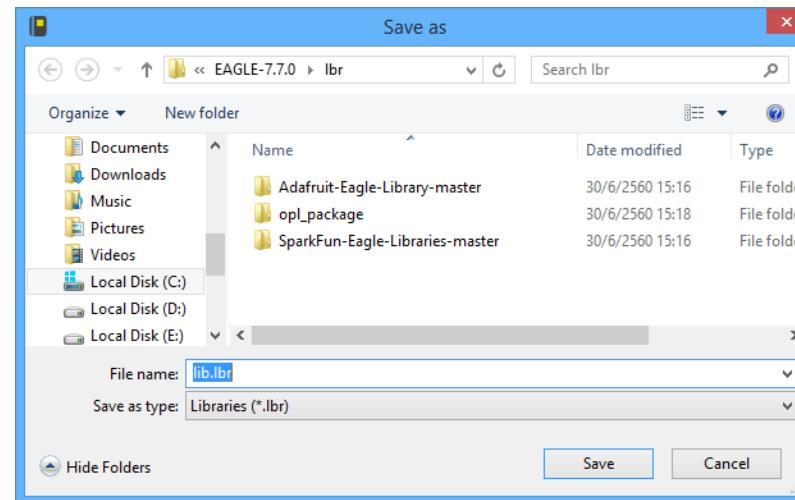
File > New > Library

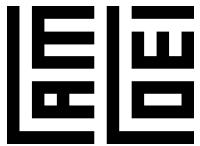


สร้าง library ใหม่

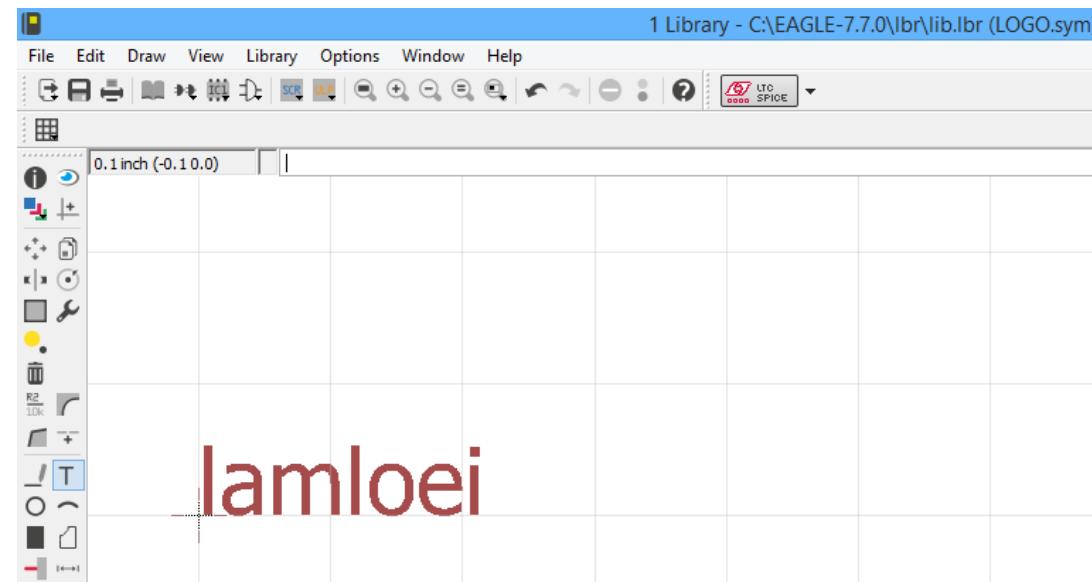
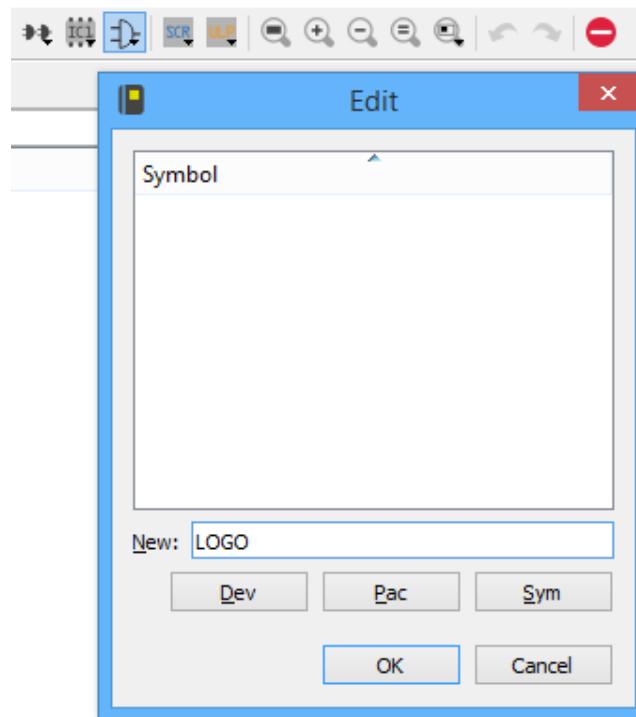
แล้วบันทึก

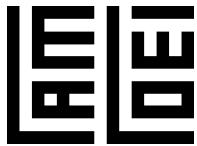
คลิกขวา Use



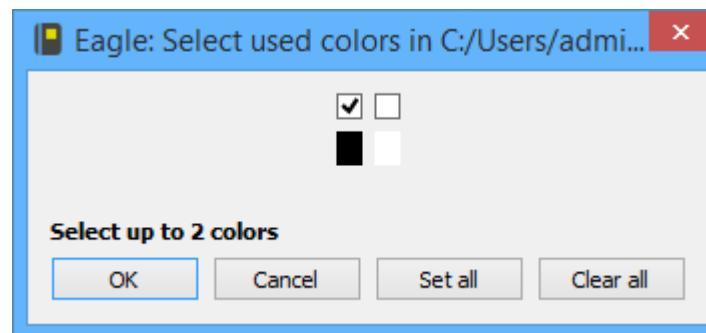
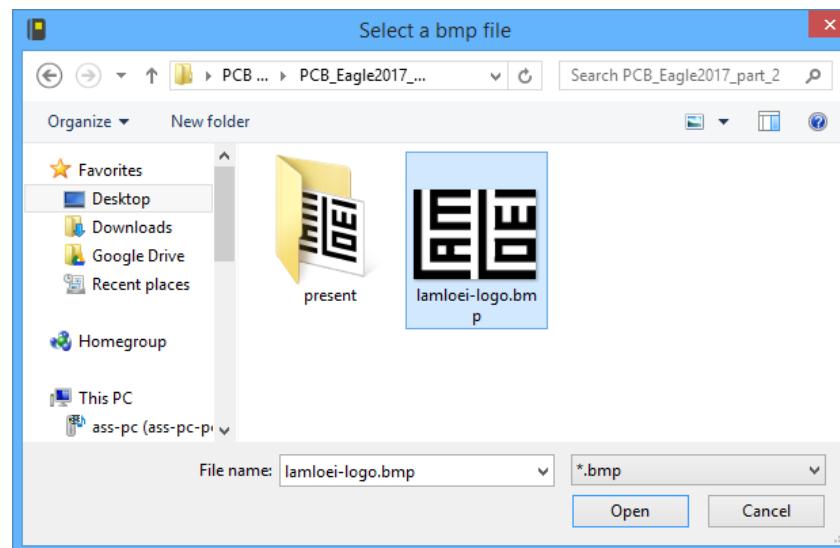
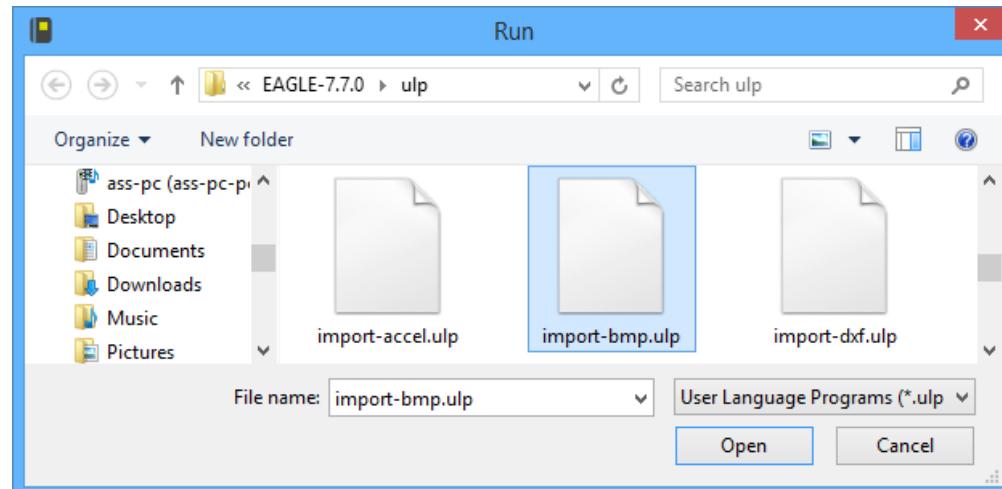
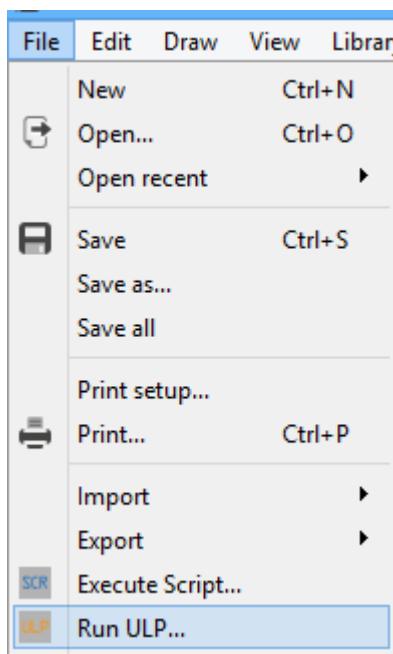
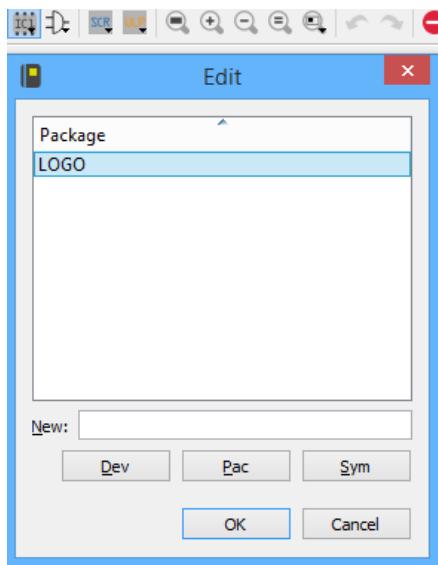


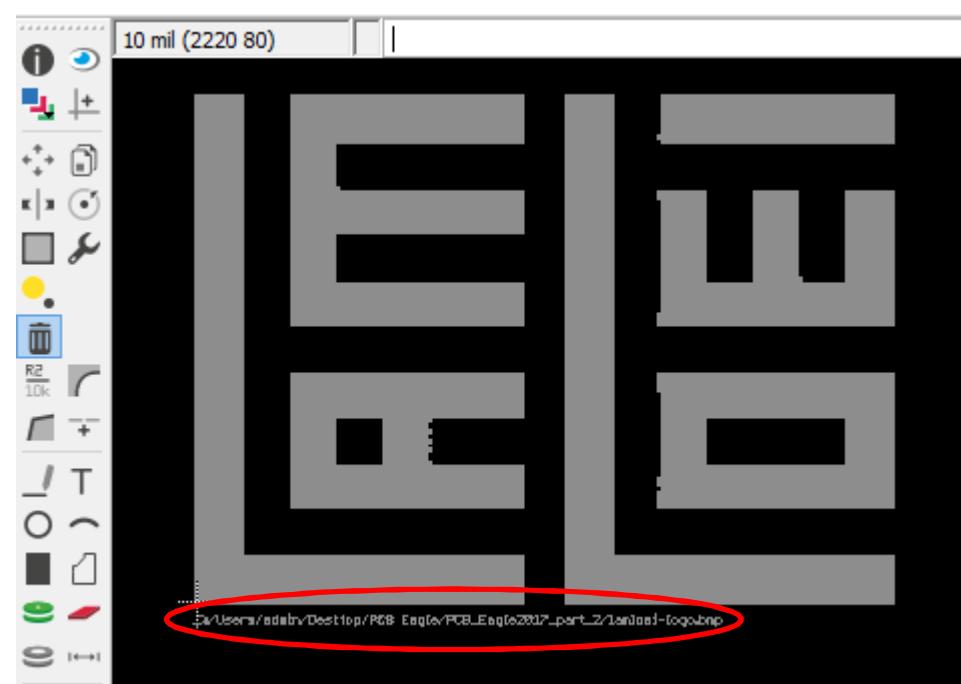
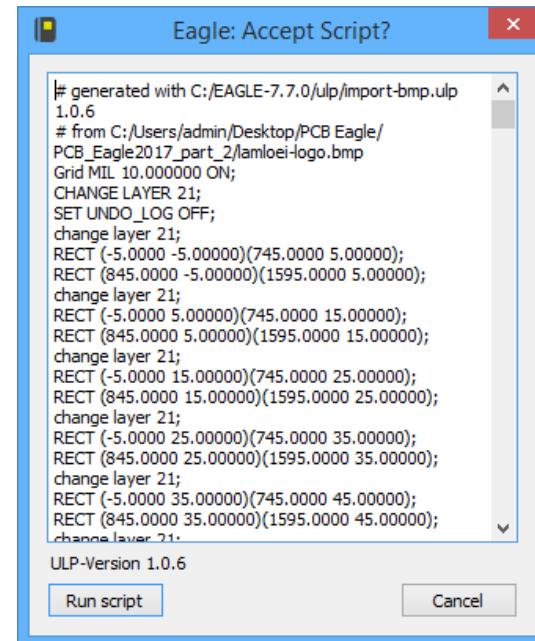
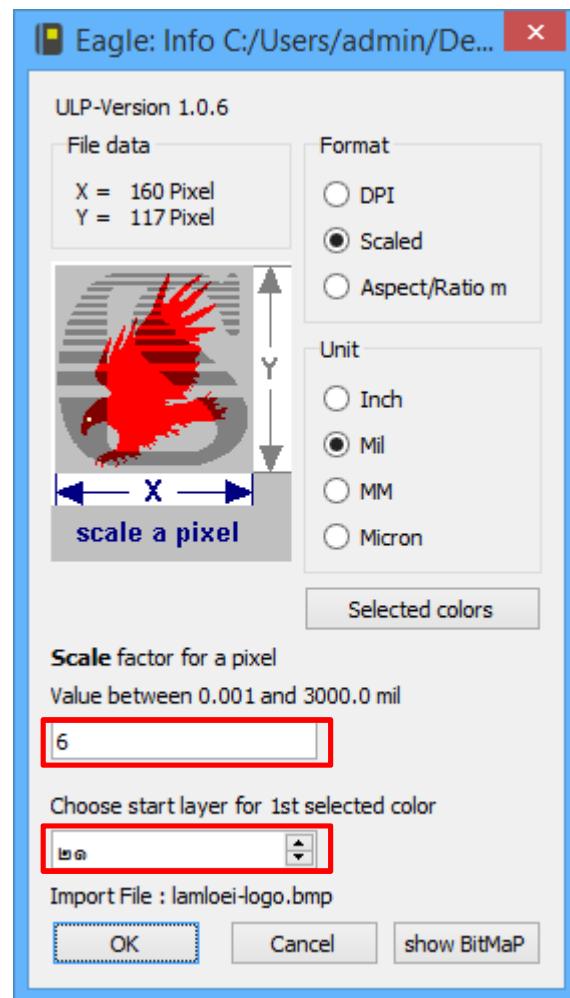
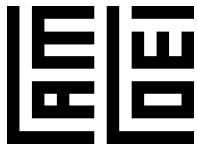
New Symbol





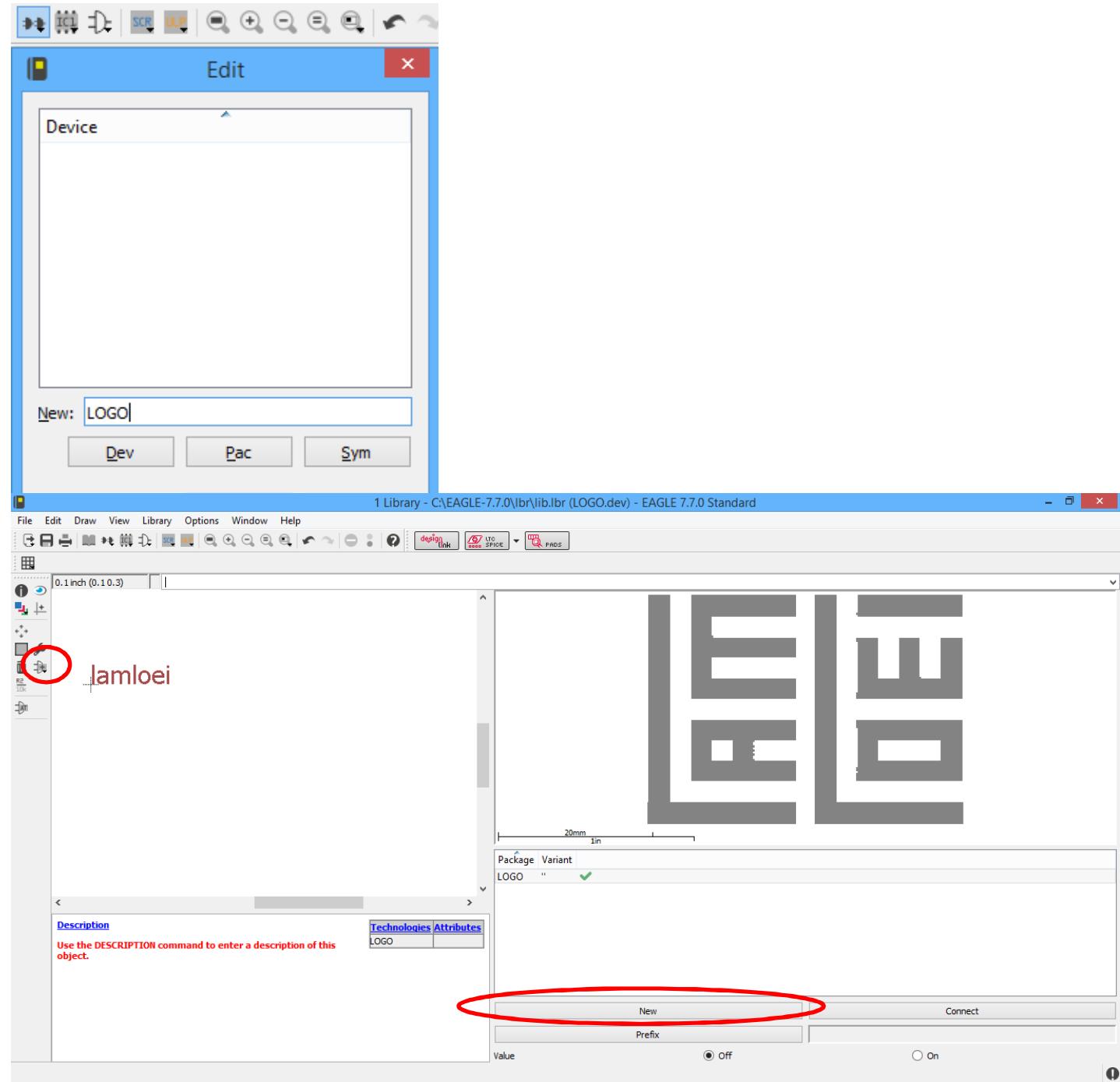
New Package



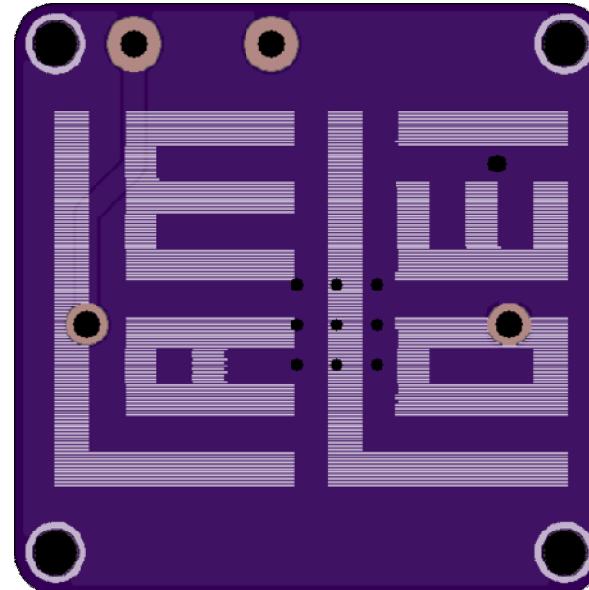
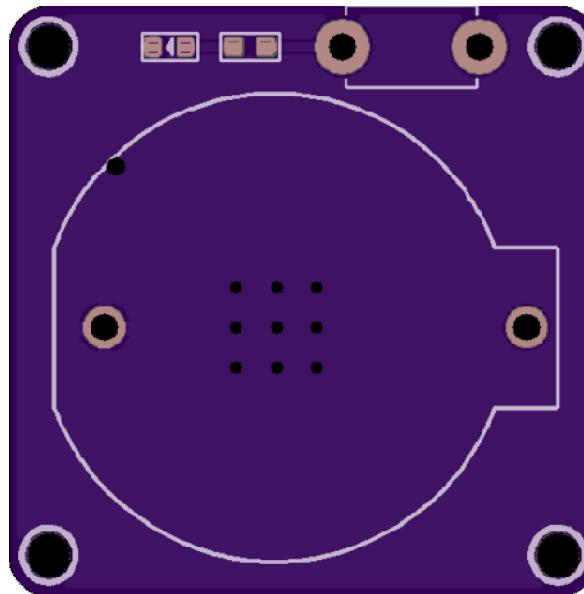
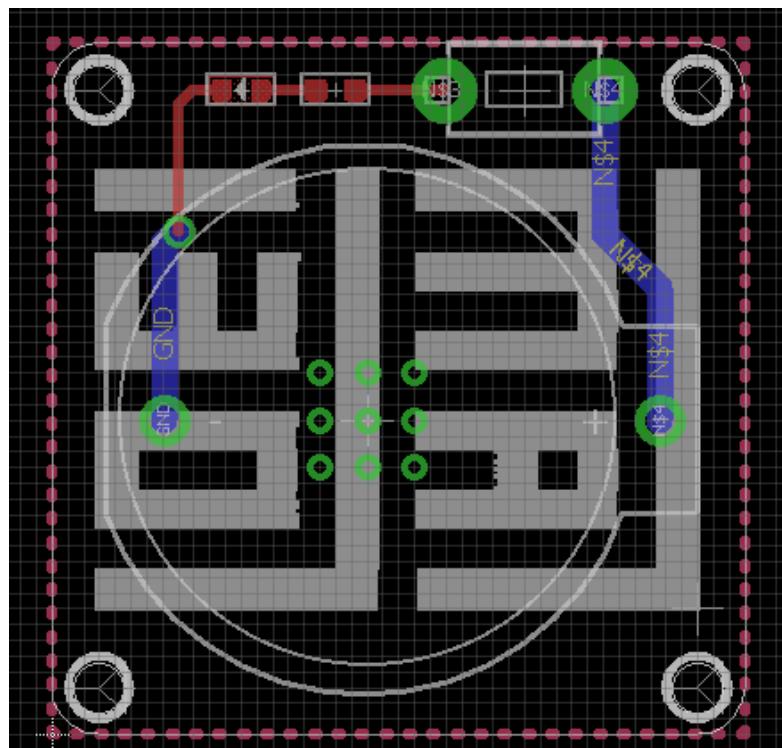
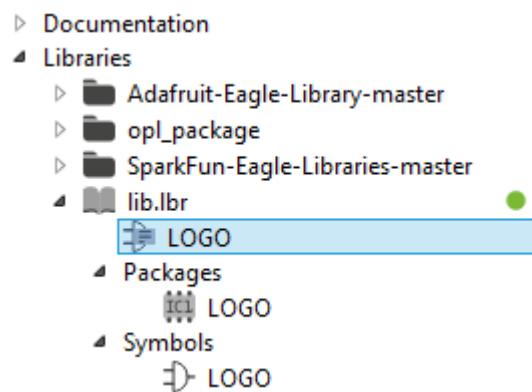
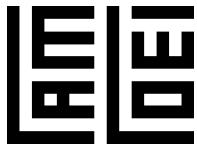


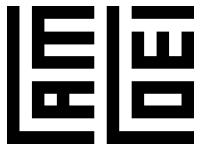


New Device



แล้วบันทึก



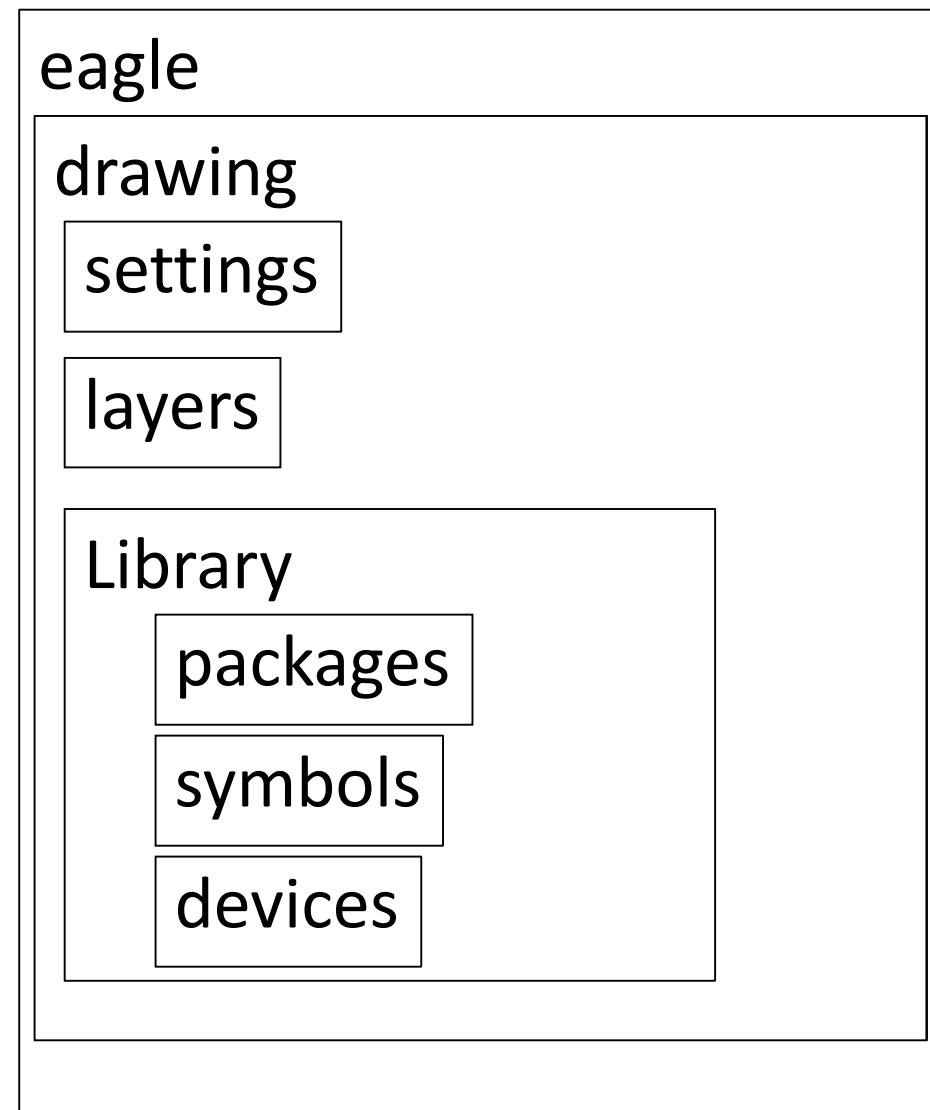


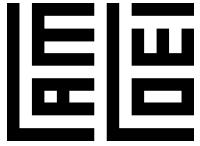
Lab6 xml

```
Seeed-Relay.lbr x
1   <?xml version="1.0" encoding="utf-8"?>
2   <!DOCTYPE eagle SYSTEM "eagle.dtd">
3   <eagle version="6.4">
4     <drawing>
5       <settings>
6         <setting alwaysvectorfont="no"/>
7         <setting verticaltext="up"/>
8       </settings>
9       <grid distance="50" unitdist="mil" unit="mil" style="lines" multiple="1" display="yes"
10      <layers>
168     <library>
169       <packages>
170         <package name="REY5-19.0X15.0X15.0MM">
184       </packages>
185       <symbols>
186         <symbol name="RELAY-HLS8L-DC5V-S-C">
209       </symbols>
210       <devicesets>
211         <deviceset name="RELAY-HLS8L-DC3V-S-C(5P-19X15MM)" prefix="K" uservalue="yes">
234         <deviceset name="RELAY-HLS8L-DC5V-S-C" prefix="K" uservalue="yes">
254       </devicesets>
255     </library>
256   </drawing>
257 </eagle>
```



โครงสร้าง xml ของ library





Q & A

