

تطبيقات ميكاترونيك -1-

Lecture No. 2

SCH PCB By Eagle part 2

روبوت و أنظمة ذكية - سنة ثالثة

Dr. Eng. Essa Alghannam

Dr. Eng Fadi Mtaweg

2022-2023



الغاية من هذه المحاضرة:

تعليم استخدام برنامج EAGLE لتصميم دارة الكترونية مطبوعة.

- تعلم Schematic
 - تعلم PCB
- قواعد التصميم الإلكتروني
 - فحص الأخطاء

الغاية النهائية تصميم دارة المشروع لمادة تطبيقات 1 باستخدام هذا البرنامج.



2 Board - C:\Users\ESSA\Desktop\1.brd - EAGLE 7.6.0 Professiona

0.05 inch (6.45 0.15)

4 ⊭ ⊹ © +> ⊙ ∎ &

•. ≣ ‡

¢∏ -∭ 6

世 『 デ 小 く

* * *

File Edit Draw View Tools Library Options Window Help



2 Schematic - C:\Users\ESSA\Documents\EAGLE\projects\my_first_project\my_schematic.sch





Туре	Value	Note		
PCB Dimensions	min. 5 x 5 mm	PCB does not have to be		
	max. 425 x 425	rectangular!		
	mm			
Board thickness	1.55 <i>mm</i>	Industry standard		
	1.00 <i>mm</i>	Also a standard for ceramic PCBs Heavy components		
	2.00 mm			

Electrical layout characteristics

- Guidelines for outer layers (top/bottom), but also suitable for inner layers
- Copper base thickness: $18 \mu m$
- Suitable for PCB thickness of: $\leq 2.4mm$



PCB composition:



In a 2-layer design, the Copper Inner and Prepreg layers are left out but the base FR4 is thicker.

- The thickest, middle part of the board is a insulating substrate (usually FR4).
 Fiberglass Sheets
- FR4 is a standard defined by the NEMA (National Electrical Manufacturers Association) لمصنعي الأجهزة الكهربائية الرابطة الوطنية for a glass-reinforced epoxy resin laminate صفائح راتينج إيبوكسي مقوى بالزجاج FR stands for "flame retardant" مثبط لهب
- On either side of that is a thin layer of **copper**, where our electric signals pass through.
- To insulate and protect the copper layers, we cover them with a thin layer of lacquer-like soldermask, which is what gives the PCB color (green, red, blue, etc.).
- Solder mask is a thin lacquer-like layer of polymer that is usually applied to the copper traces of a printed circuit board (PCB) for protection against oxidation and to prevent solder bridges from forming between closely spaced solder pads.







- Finally, to top it all off, we add a layer of ink-like silkscreen, which can add text and logos to the PCB.
- Silkscreen is a layer of ink traces used to identify components, test points, parts of the PCB, warning symbols, logos, texts and marks etc.



PCB pads



- PCB pads are the exposed metals on the substrate where the component lead is soldered.
- Pads can be through-hole ones or surface mount pads.
- A hole is considered PTH (Plating-Through-Hole) if its pad has copper along with the solder stop mask.
- solder stop mask (where solder mask should not be applied) should be larger than the hole/slot with a minimum of 6 mils in width.
- Holes in which the pad's copper size is smaller than the hole or if there is no copper at all are NPTH Non-Plating-Through-Hole.





Distinguish between Plating-Through-Hole and Non-Plating-Through-Hole PCBs

- The easiest way to do that is by visually checking for traces of plating on the borehole wall in the PCB.
- Non-Plating-Through-Hole PCBs will not have any traces of copper in the borehole wall.
- Plated-Through-Holes PCBs are more expensive than non-plated-through-hole PCB.
- Also, PTH printed circuit boards are often smaller than NPTH printed circuit boards.

The Use of Non-Plated-Through-Holes in PCBs



- NPTHs are simpler PCBs. Therefore the manufacturing process for these is faster. They are frequently used as Tooling/Mounting holes, to fix the PCB to its operational location.
- They are also used to mount components in some single-sided PCBs.

The Use of Plated-Through-Holes in PCBs

These PTH holes have two purposes:

- Component holes for welding DIP components: In such cases, the hole diameter must be larger than the pin of the component so that the component can be inserted in the PTH.
- Vias can be between outer and inner layers, or inner layers only, or from surface-to-surface. They connect and conduct wiring between different layers in PCB.
- The size of these PTH—Plating-Through-Holes will be smaller than component holes.







PCB composition:

Туре	Minimal	Recomm.	Note
Trace Width	0.15 mm		Recommended values: see "PCB
			trace widths"
Edge-Trace	0.45 mm	1.0 <i>mm</i>	
Trace-Trace	0.15 mm	0.35 mm	
Trace-Pad	0.15 mm	0.35 mm	
Pad-Pad	0.15 mm	0.5 <i>mm</i>	
Pad-Hole	0.20 mm	0.5 <i>mm</i>	Also for Trace-Hole
Finished hole ø for PTH	0.25 mm		Initial drill hole ø will be +0.1 <i>mm</i> to accommodate through hole plating
Finished hole ø for NPTH	0.35 mm		
OAR PTH	0.125 <i>mm</i>	0.35 mm	
OAR NPTH	0.30 mm	0.5 mm	For non-plated hole but with pad on top/bottom
Via ø	0.45 mm	0.6 mm	A PTH to connect traces from/to different layers



PCB composition:

جامعة الم نارة

PCB trace widths

- Copper base thickness: $18 \mu m$
- + Temperature rise of trace: $\leq 15^{\circ}{\rm C}$
- Ambient temperature: $\sim 25^\circ {
 m C}$
- For outer layer traces only (top/bottom)

Signal type	Minimal	Recommended	Application
Low current	0.5 mm	1.0 <i>mm</i>	Low voltage DC power supply lines
High current	1.5 <i>m</i> m	≥ 2.5 mm	Mains, high power DC
Digital	0.2 <i>mm</i>	0.5 mm	I/O, logic, microcontroller, CPLD
Analog	0.5 <i>m</i> m	0.8 mm	ADC, low power op-amps



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

Resistance = Resistivity*Length/Area*(1 + (Temp_Co*(Temp - 25))

R =
$$(\rho * L / A') * (1 + \alpha * (^{T}_{TEMP} - 25 °C))$$

Where,

Copper Resistivity $\boldsymbol{\rho}$ = 1.72 x 10⁻⁸ Ω m L=length Area A' = Thickness*Width A copper Thickness of 1oz/ft^2 ~ 0.0035 cm وزن 1 أونصة على مساحة قدم مربع Copper Temp_Co α = +0.393 percent per degree C. This means if the temperature increases 1°C, the resistance will increase 0.393%.

Voltage Drop is Current * Resistance

Power Loss is Current^2 * Resistance





PCB

https://www.mclpcb.com/blog/pcb-trace-width-vs-currenttable/ https://www.4pcb.com/trace-width-calculator.html https://circuitcalculator.com/wordpress/2006/01/31/pcbtrace-width-calculator/

https://www.omnicalculator.com/other/pcb-trace-width

PCB Copper Thickness



The copper in a PCB is rated in ounces, It's the resulting thickness when <mark>1 oz of copper is pressed flat and spread evenly over a one square foot area. This equals 1.37 mils (1.37 thousandths of an inch).</mark>

For example a PCB that uses 1 oz. copper has a thickness of 1.4mils.

	1/2 oz.		0.7 mils	
	1 oz.		1.37 mils = <mark>0.03556 mm =35um</mark>	
	2 oz.		2.8 mils	
PCB Copper Thickness (daycour	nter.com)			
opper thickness: 0.035 mm= 35 ur opper or track width: 0.25 mm_fo	n r 1A	2.54	mm=0.1 inch=100 mil	
Mass		\$	Length	¢
1	= 28.3495		1 = 0.0254	
Ounce	≑ Gram	÷	Thousandth of an inch Millimetre Formula divide the length value by 39.37	¢

PCB layers:

EAGLE's Layers

المَـنارة The EAGLE board designer has layers just like an actual PCB, and they overlap too. We use a palette of colors to represent the different layers. Here are the layers you'll be working with in the board designer:

Color	Layer Name	Layer Number	ayer Purpose							
	Тор	1	Top layer of copper							
	Bottom	16	Bottom layer of copper							
	Pads	17	Through-hole pads. Any part of the green circle is exposed copper on <i>both</i> top and bottom sides of the board.							
	Vias	18	Vias. Smaller copper-filled drill holes used to route a signal from top to bottom side. These are usually covered over by soldermask. Also indicates copper on both layers.							
	Unrouted	19	Airwires. Rubber-band-like lines that show which pads need to be	connected.						
	Dimension	20	Outline of the board.							
	tPlace	21	Silkscreen printed on the top side of the board.							
	bPlace	22	Silkscreen printed on the bottom side of the board. Top origins, which you click to move and manipulate an individual part. Origins for parts on the bottom side of the board.							
	tOrigins	23								
	bOrigins	24								
// Hatch	tStop	29	Top stopmask. These define where soldermask st ould not be applied. Absent soldermask on the bottom side of the board. Non-conducting (not a via or pad) holes. These are usually drill holes for stand-offs or f special part requirements.							
۱۱ Hatch	bStop	30								
	Holes	45								
	tDocu	51	Top documentation layer. Just for reference. This might show the outli other useful information.	ne of a part, or						





https://manara.edu.sy/

جَامعة

PCB layers:

To turn any layer off or on, click the "Layer Settings..." button -- -- and then click a layer's number to select or de-select it.

Before you start routing, make sure the layers above (aside from tStop and bStop) are visible.

	R 1 Bo	ard C			cuponto\ E A	CLEV projectel my first proje	-
	Eilo Edit	BRI	Visib	le Layers		×	
deal á							
جامعه							
الم_نارة			Filter:	All Layers	*		
MANARA UNIVERSITY	1 2 , 3						
			۲	#	Name	×	
	() (۲	1	Тор		
	U C		۲	16	Bottom		
			۲	17	Pads		
	+ 4k		۲	18	Vias		
	↔ /\		۲	19	Unrouted		
			۲	20	Dimension		
	> -		۲	21	tPlace		
	D B		۲	22	bPlace		
	- a		۲	23	tOrigins		
			۲	24	bOrigins		
	•		•	25	tNames		
	•	' I	۲	20	bNames		
	* 7		۲	2/	tValues		
		1	۲	20	bValues		
			•	29	tStop		
	\triangle A	F	۲	21	bStop		
				32	tCream	•	
	* -	· [] -		32 💟	bCream tCipich		
		1		34	bEinich		
				35 📈	tChuo		
	≣∖≣ ∿	·		36 N	bGlue		
	A990 -	-4		37	tToct		
	₽ + ⁰	.		38	hTest		
	R2 R2			39	tKeepout		
	10k 10k	-		40	bKeepout		
	‡⊅ ∂				ſ		
			New	Layer	l	Show Layers Hide Layers	
	~ ~ <u>~</u>		Laver	Sets	*	New Set Remove Set	
	7 .3		20,01				
	e c					OK Cancel	









A 0.05" grid, and 0.005" alternate grid is a good size for this kind of board.

EAGLE forces your parts, traces, and other objects to "snap" to the grid defined in the *Size* box.



2.54 mm=0.1 inch=100 mil



Elle Edit Draw View

). E 🖯

Types

5_ #_ T

۲

Α

|→ |/ ≧≡ ∿

Ĵ

R2

7

ß

<Preset

<Preset :

<Preset

Angle:

0

Moving Parts: +

Our first job in this PCB layout will be <mark>arranging the parts</mark>, and then <mark>minimizing the area of our PCB dimension</mark> outline. PCB costs are usually related t <mark>board size</mark>, so a <mark>smaller board is a cheaper board</mark>.

you can start to move parts within the dimension box. While you're moving parts, you can **rotate** them by either right-clicking or changing the angle in the drop-down box near the top.

•Don't overlap parts: All of your components need some space to breathe. The green via holes need a good amount of clearance between them too. Remember those green rings are exposed copper on both sides of the board, if copper overlaps, streams will cross and short circuits will happen.

•Minimize intersecting airwires: While you move parts, notice how the airwires move with them. Limiting crisscrossing airwires as much as you can will make routing *much* easier in the long run. While you're relocating parts, hit the RATSNEST button to get the airwires to recalculate.

Part placement requirements: Some parts may require special consideration during placement. For example, you'll probably want <mark>the insertion point of the barrel jack connector to be facing the edge of the board</mark>. And make sure that decoupling capacitor is nice and close to the IC. Tighter placement means a smaller and cheaper board, but it also makes routing harder.





Moving Parts:

<mark>Group all</mark> parts and <mark>move them</mark> inside the yellow borders near the <mark>origin corner</mark>



Adjusting the Dimension Layer



Then use the line tool 🖌 to draw manually a new outline. Before you draw anything though, go up to the options bar and set the layer to 20 Dimension. Also up there, you may want to turn down the width a bit (we usually set it to 0.008").

Layer:	20 Dimension	Ŧ	٣	Width:	6	Ŧ	Style:	continuous	٣	Radius:	0	• 77

Then, starting at the origin, draw a box around your parts. Don't intersect the dimension layer with any holes, or they'll be cut off! Make sure you end where you started.

ed.	Properties			×
• You can also right click	Line			
on the each yellow	From	0	100	
border and set values	To	0	0	
from to	Angle	270		
• Adjust the unit from	Width	0		Ŧ
Grid	Style	continuous		Ŧ
	Сар	round		٣
	Layer	20 Dimension	n	T
	Curve	0		
	Locked			

Adjusting the Dimension Layer





You can draw it based on your requirements



Adjust the parts location





- Adjust the parts location by move and rotate
- Use ratsnest to redraw "unrouted" airwires





Adjust the **Isolate** setting which defines how much clearance the ground pour gives other signals, 0.012" for this is usually good.



جَامعة

Adding Copper Pours

POLYGON

Using EAGLE: Board Layout Routing the Board



- turning each of those gold **airwires** into top or bottom copper traces.
- At the same time, you also have to make sure not to overlap two different signals.





× C

Using the Route Tool:



' Width: 0.016 ∨ 🖪 💿 O Diameter: auto ∨ Drill: 0.02362205

🖱 Grid		×	
Display	Style		
🔾 On 💿 Off	◯ Dots	Lines	
Size: 0.05	inch 🔻	Finest	
Multiple: 1			
Alt: 0.025	inch 🔻	Finest	
Default	ОК	Cancel	

•Layer: On a 2-layer board choose whether you want to start routing on the top (1) or bottom (16) layer. •Width: This defines how wide your copper will be. Usually 0.01" is a good default size. You shouldn't go any smaller than 0.007" (or you'll probably end up paying extra).

Wider traces can allow for more current to safely pass through.If you need to supply 1A

through a trace, it'd need to be much wider.

•to find out how much, exactly, use a trace width calculator.

•Via Options: You can also set a few via characteristics here. The shape, diameter, and drill can be set, but usually the defaults (round, auto, and 0.02" respectively) are perfect.

- you start a route by left-clicking on a pin where a airwire terminates.
- The airwire, and connected pins will "glow", and a red or blue line will start on the pin.
- You finish the trace by left-clicking again on top of the other pin the airwire connects to.
- Between the pins, you can left-click as much as you need to "glue" a trace down.

While routing it's important to avoid two cases of overlap: copper over vias, and copper over copper. Remember that all of these copper traces are basically bare wire. If two signals overlap, they'll short out, and neither will do what it's supposed to.

If traces do cross each other, make sure they do so on opposite sides of the board. It's perfectly acceptable for a trace on the top side to intersect with one on the bottom. That's why there are two layers!







Placing Vias



- Vias are really tiny drill holes that are filled with copper.
- We use them mid-route to move a trace from one side of the board to the other.
- To place a via mid-route, first left-click in the black ether between pins to "glue" your trace down.
- Then you can either change the layer manually in the options bar up top, or click your middle mouse button to swap sides.
- And continue routing to your destination. EAGLE will automatically add a via for you.



use the RIPUP tool -- -- to remove traces. This tool turns routed traces back into airwires. You can also use UNDO and REDO to back/forward-track.



Checking for Errors

• Ratsnest -- Nothing To Do!

⊁։ ⊷ 21 of 21 shown (0 selected) Devices **|→**〉 Q Search ... Name ≣∖≣ Մ Footprint C1 E2-5 10uf Ĵ **∔**⊅, C2 E2-5 10uf C3 22pf C025-024X044 **R2** 10k R2 10k C4 C5 22pf C025-024X044 C025-024X044 0.1uf +.D-് IC1 79XXS J1 POWER_JACK_PTH_LOCK POWER_JACKP1 1 JP1 1X05 JP2 1X06 70 0 of 0 shown (0 selected) Items 0 ζΞ Q Search ... Ŧ ++ Туре Name Signal ⊢ + \approx

Ratsnest: Nothing to do! Left-click to select signal object to route

جَـامعة المَـنارة



Right click om it then properties



🕮 Properties			\times
Wire			
From	21.59	24.13	
То	41.91	24.13	
Length	20.32		
Angle	0		
Width	0.4064		•
Style	continuous		•
Сар	round		*
Layer	16 Bottom		•
Curve	0		
Locked			
Signal			
Name	D8		
Net Class	0 default	•)
Airwires hidden			
	OK Can	cel Appl	ly]









- the most common errors:
- **Clearance**: A trace is too close to either another trace or a via. You'll probably have to nudge the trace around using the MOVE tool.
- Overlap: Two different signal traces are overlapping each other. This will create a short if it's not fixed. You might have to RIPUP one trace, and try routing it on the other side of the board. Or find a new way for it to reach its destination.
- **Dimension**: A trace, pad, or via is intersecting with (or too close to) a dimension line. If this isn't fixed that part of the board will just be cut off.
- **Stop mask errors**: the silkscreen overlapping. **Ignore them**

tPlace	21		
bPlace	22		



Add a Hole (NPTH)



* •

1.5.15





ULP user language Programs	
2 Board - E:\softwares\Engineering Aprilelectronic software\SCH schematic layout\eagle\MYexamples\0.my File Edit Draw View Tools Library Options Win, 'w Help Image: Ima	کےامد المَــنا
Image: Design Manager Inspector Selection Filter 1 mm (-29 73) Click or press Ctrl+L key to activate command li Image: Types Image: Design Manager Inspector Selection Filter Image: Design Manager Inspector Selection Filter Image: Types Image: Design Manager Inspector Selection Filter Image: Design Manager Inspector Selection Filter Image: Types Image: Design Manager Inspector Selection Filter Image: Design Manager Inspector Selection Filter Image: Types Image: Design Manager Inspector Selection Filter Image: Design Manager Inspector Selection Filter Image: Types Image: Design Manager Inspector Selection Filter Image: Design Manager Inspector Selection Filter Image: Types Image: Design Manager Inspector Selection Filter Image: Design Manager Inspector Selection Filter Image: Types Image: Design Manager Inspector Selection Filter Image: Design Manager Inspector Selection Filter Image: Types Image: Design Manager Inspector Selection Filter Image: Design Manager Inspector Selection Filter Image: Types Image: Design Manager Inspector Selection Filter Image: Design Manager Inspector Selection Filter Image: Types Image: Design Manager Inspector Selection Filter Image: Design Manager Inspector Selection Filter Image: Types Image: Design Manager Inspector Selection Filter <	ne mode X
	D:/EAGLE 9.6.2/examples/ulps/examples
 Hole Hole Line Polygon Edge Rectangle Spline Text 	$ \begin{array}{c} es \\ es \\ \leftarrow \rightarrow \lor \uparrow I & Tools-master & \checkmark & \circlearrowright & \mathcal{P} & Search Tools-master \\ \hline & \bigcirc & Organize & New folder & & & & & & & & & & & & & & & & & & &$
Via Wre Layers A B	een layers. It Amme Date modified # Quick access mormalize-text.ulp 10/28/2020 10:0 Image: Desktop Image: Desktop Image: Desktop Image: Desktop Image: Desktop <td< th=""></td<>
GitHub - moderndevice/ I ools: Tools	File name: normalize-text2.0.ulp V User Language Programs (*.ulp) V
https://m	Den Cancel



Manufacturing Visualize





Add Silkscreen TEXT









D12 digital pin	
_D11 digital pin	
_D10 digital pin	
_D9 digital pin	

Adding Dimension









CAM processor-Generating Gerbers

• Gerber files are open ASCII vector format files that contain **information on each physical board layer of your PCB design**. Circuit board objects, like copper traces, vias, pads, solder mask and silkscreen images, are all represented by a flash or draw code, and defined by a series of vector coordinates.



CAM processor-Generating Gerbers

	_						
m	CAM Processor		— q				
		_					
1	template_2_layer.cam	Export as ZIP Export to Project Directory	Units:				
Inspe	Output Files	Name Bill of Material					
	▼ Gerber						
	Top Copper	BOM Filename %ASSEMBLYPREFIX/%N					
	Bottom Copper						
	Soldermask Top	Filename Resolution CAMOutputs/Assembly/arduino_uno.txt					
	Soldermask Bottom	List Types					
	Solderpaste Top						
	Solderpaste Bottom	O Parts CAM Processor	🕮 Select CAM file				
	Silkscreen Top	Values					
	Silkscreen Bottom	tamehta 2 huar ann	$\Box \leftarrow \rightarrow \checkmark \land \downarrow \land$ Third Party \rightarrow OSH Park $\checkmark \land$				
	▼ Drill 503	List attribute					
	▼ Assembly						
	Bill of Material	Ouput Type Output Files	Organize New folder				
	Pick and Place	▼ Gerber					
	Drawings	O HTML Top Copper	lools-master n Name				
	Legacy	O CSV Bottom Copper					
		Profile Caldermark Tap	OneDrive OSHPark_2_layer.cam				
		Soldermask Top	CSHPark 4 laver cam				
		Solderpaste Top	This PC				
		Solderpaste Bottom					



https://manara.edu.sy/

OneDrive

CAM processor-Generating Gerbers



Select all and drag them to be opened by gerbv





PCB Prototype & PCB Fabrication Manufacturer - JLCPCB



Save as pdf

Save as pdf				
From Print	×	ج <u>امعة</u>	🕮 Print	×
Printer: Microsoft Print to PDF • Setup: colored, 1 copy • Output file: • Paper: A4 (210x297 mm, 8.3x11.7 inch) • Orientation: Portrait • Alignment: Center • Area: Full • Options Scale Options Scale factor: 0 9age limit: 0 • Page limit: 0 0 • Calibrate Border X 1 V 1 0 0inch	<image/>	Ratsnest	Printer: Microsoft Print to PDF • Setup: colored, 1 copy Output file: Paper: A4 (210x297 mm, 8.3x11.7 inch) Orientation: Portrait Patrea: Full Area: Full V Area: Full • Options Scale Options Scale Options Scale Options Scale factor: 3.5 Page limit: 0 0 Calibrate Border X 1 V 1 Border X 1 V 1 Calibrate Calibrate Direction Calibrate Direction Calibrate Calib	Atual scaling 3.50
	OK Cancel]		OK Cancel

PIC Development Board Learning Programmer Experiment based on Microchip PIC16F877A



Amazon.com: Balance World Inc New PIC Development Board Learning <u>Programmer Experiment + Microchip PIC16F877A: Electronics</u>



Amazon.in: Buy Embeddinator's PIC Microcontroller Development Board Online at Low Prices in India | Embeddinator Reviews & Ratings



(2143) DIY Digital PCB Exposure Box - YouTube



online tour in the JLCPCB factory

one of the largest prototype PCB manufacturers in Shenzhen, China.

• (2021) Inside a Huge PCB Factory - in China - YouTube



نهاية المحاضرة