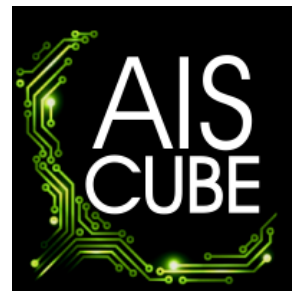


BUILD YOUR OWN ARDUINO UNO SHIELD

SESSION 02

12
GEEKS





LAST WEEK

You learnt to:

- Breadboard & connect modules to the Arduino Uno
- Basic program to control the LEDs on the dot matrix
- Set up EAGLE directories
- Add components in EAGLE

Schematic
Drawing

Preparing
the board

Part
Placement

Preparing
to Route

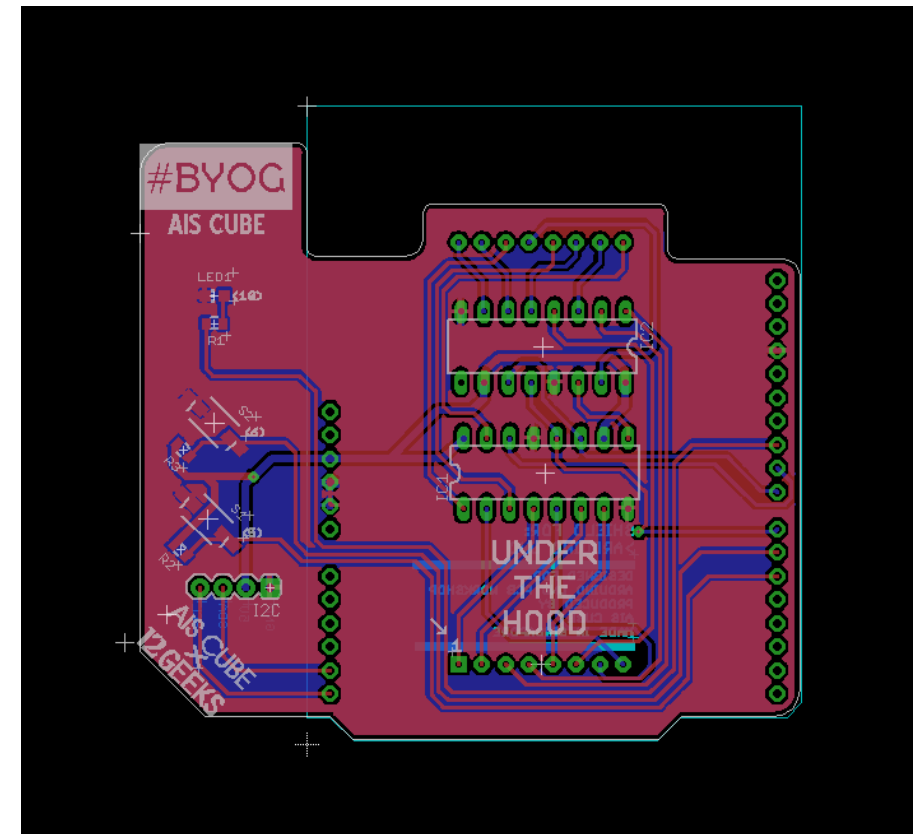
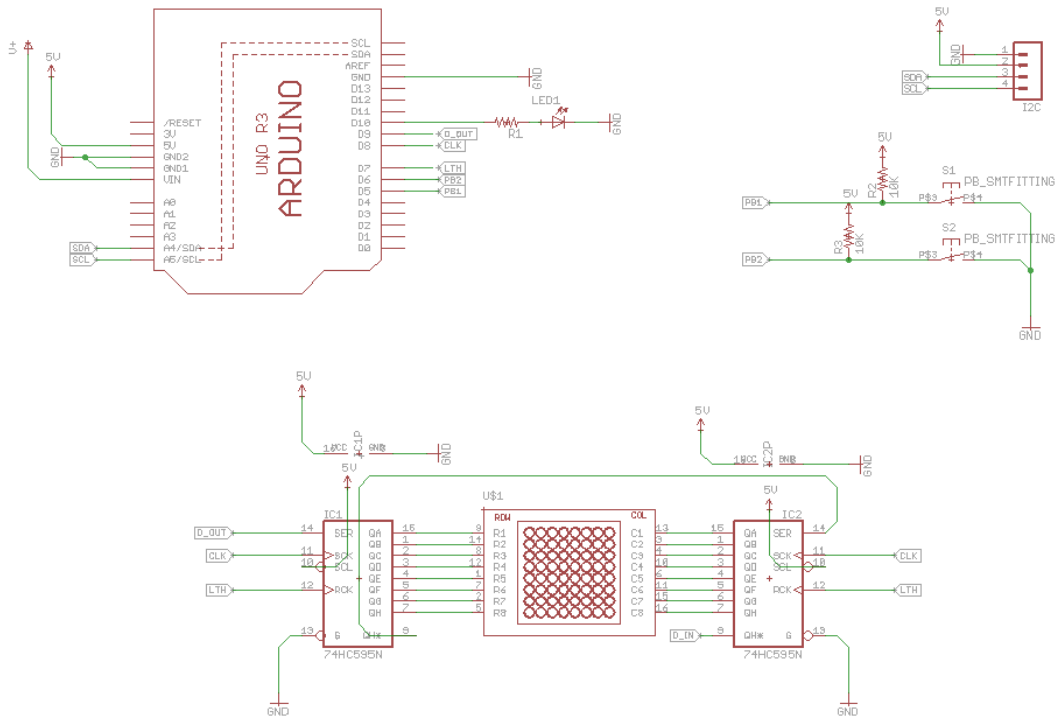
Power
Lines

Signal
Lines

GND
Planes

Silkscreen
Items

THIS WEEK



WAYPOINTS

Schematic
Drawing

Preparing
the board

- Board Dimensions
- Change grid units to mm

Part
Placement

- Good parts placement is half the battle won

Preparing
to Route

- Change grid units to 10 mil
- Setting up route parameters
- Eg. (Bend Style, Width, Via shape, via drill size)

Power
Lines

- Trace width of 32 mil
- Route lines in parallel on bottom layer

Signal
Lines

- Trace width of 16 mil
- Route lines on top layer

GND
Planes

- Polygons, signals and ratsnest

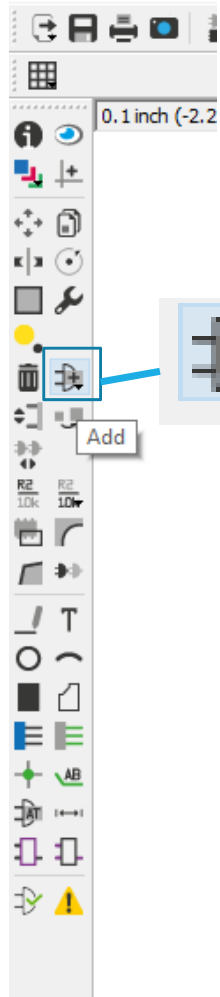
Silkscreen
Items

- Logo
- Labels

ADDING COMPONENTS

SCH 1 Schematic - D:\A

File Edit Draw



ADD

Name	Description
> AISCube_BYOG	

Symbol Preview

Footprint Preview

Attribute	Value
-----------	-------

☒ Pads ☒ Smds ☒ Description ☒ Preview

Search

Attributes

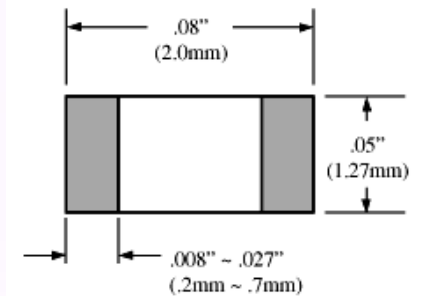
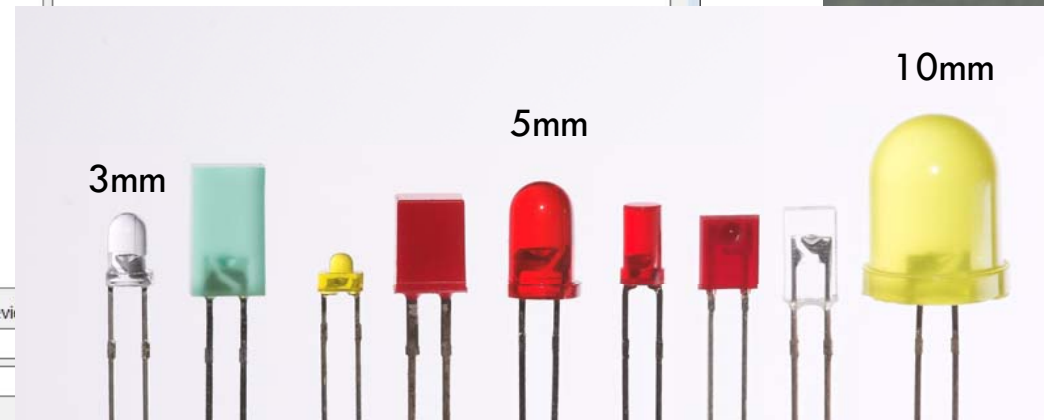
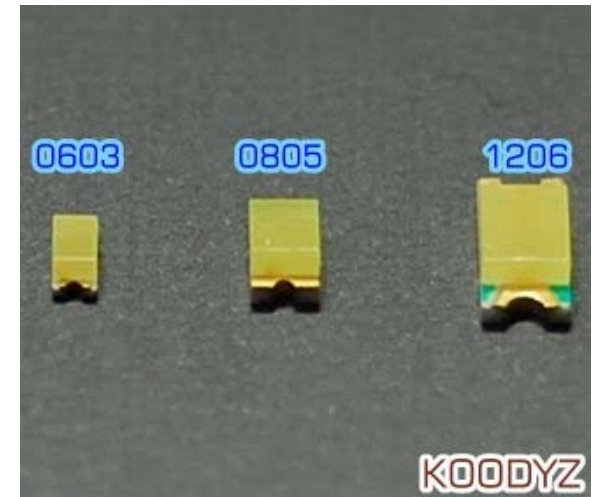
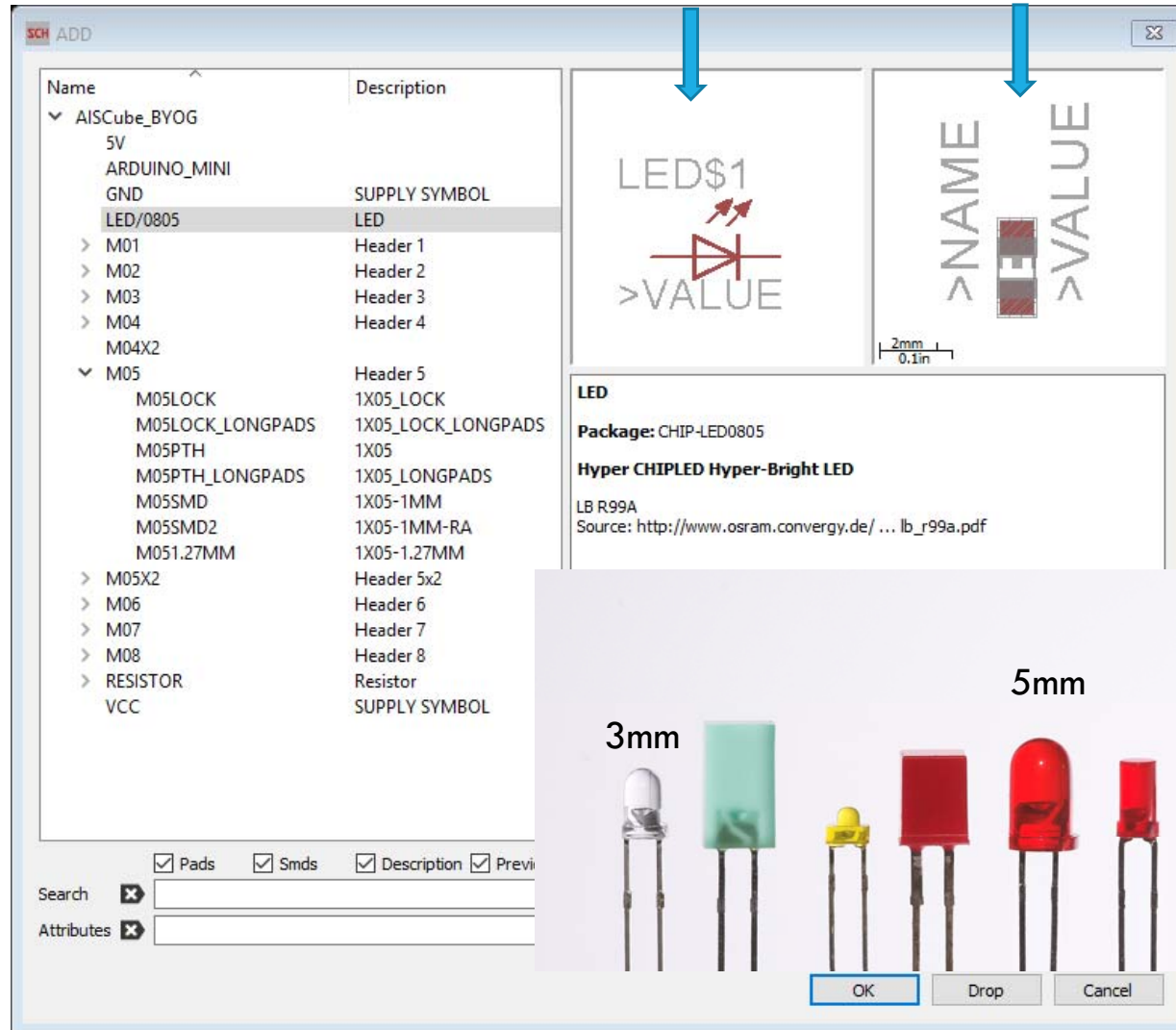
OK Drop Cancel

Symbol Preview

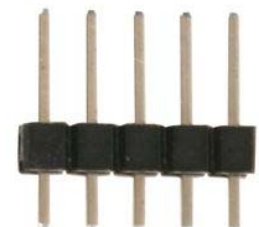
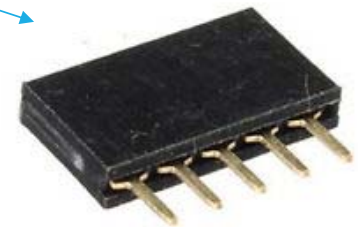
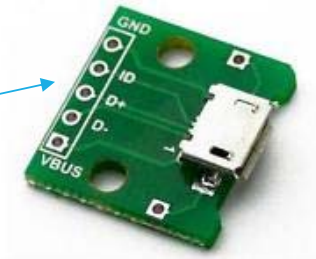
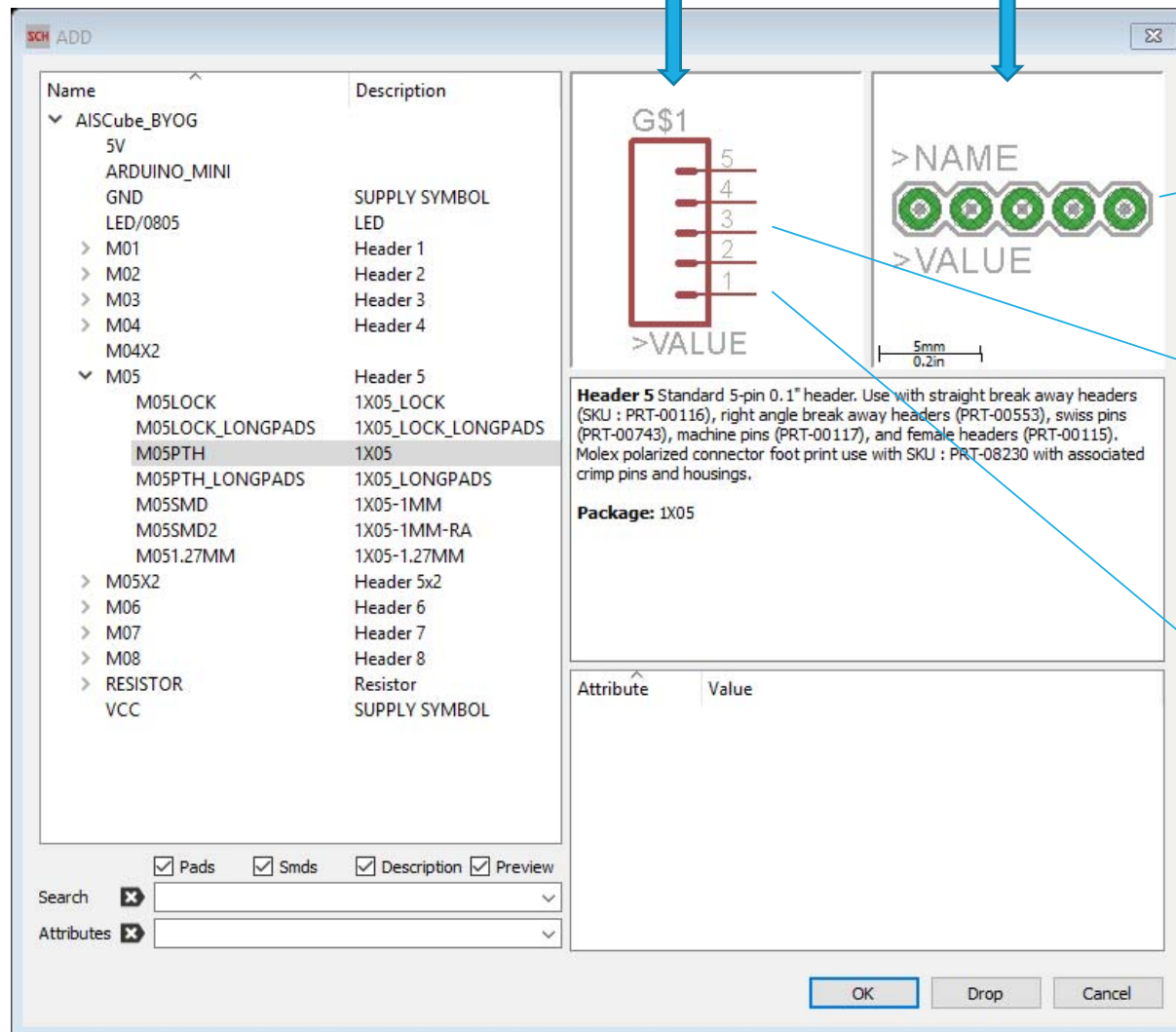
(DOES NOT CHANGE)

Footprint Preview

(CHANGES)



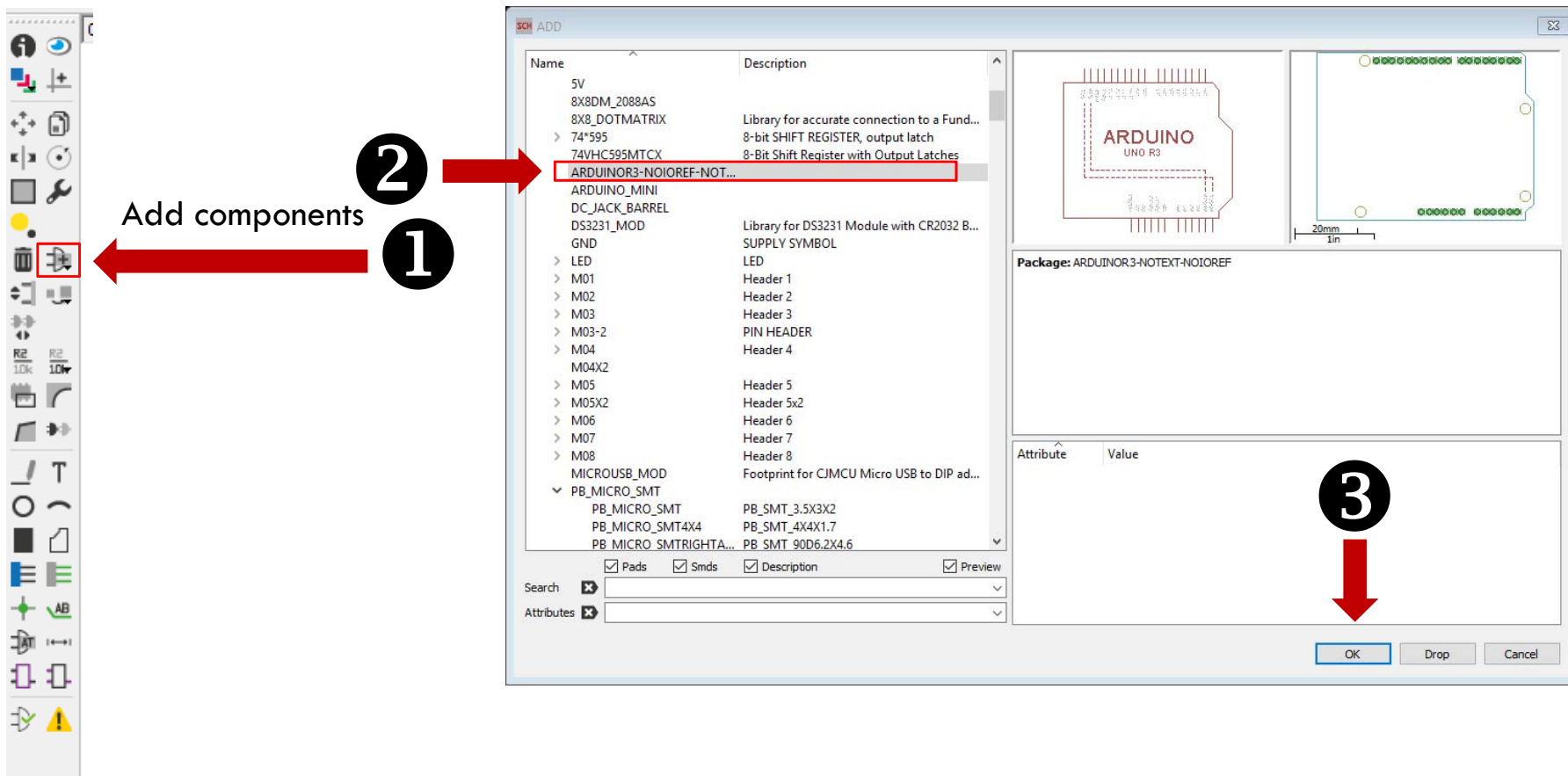
Symbol Preview Footprint Preview





ADD COMPONENTS

Let's start with the Arduino Uno Reference





ADD POWER SYMBOLS

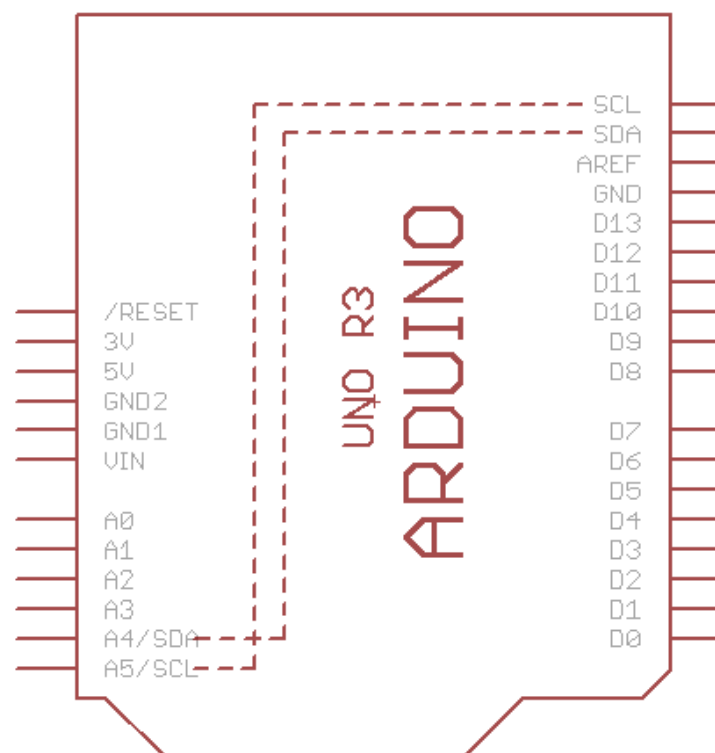


Add components

Add 5V and GND

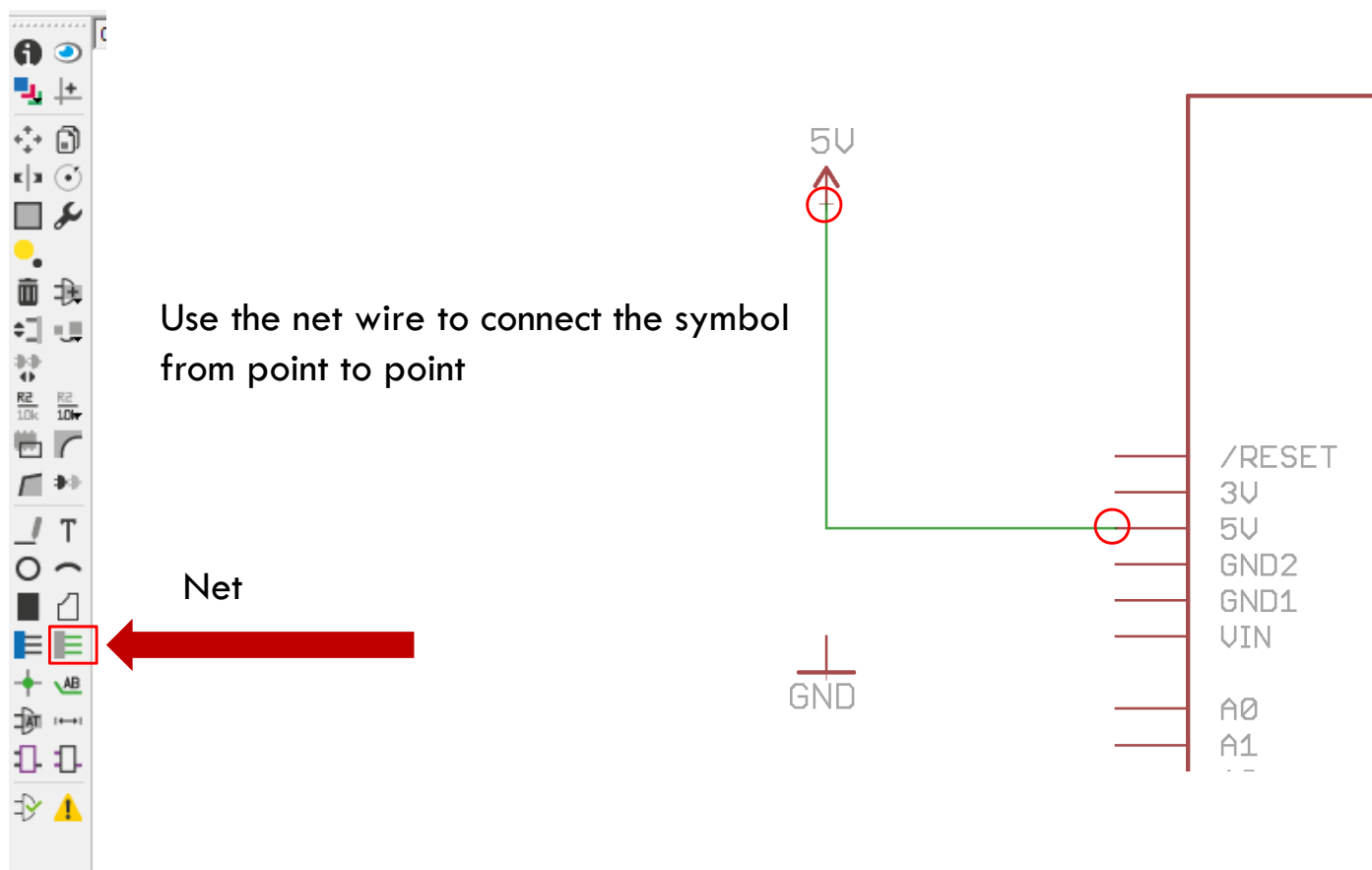
5V
↑

GND
⊥





CONNECT THE POWER LINES TO THE ARDUINO

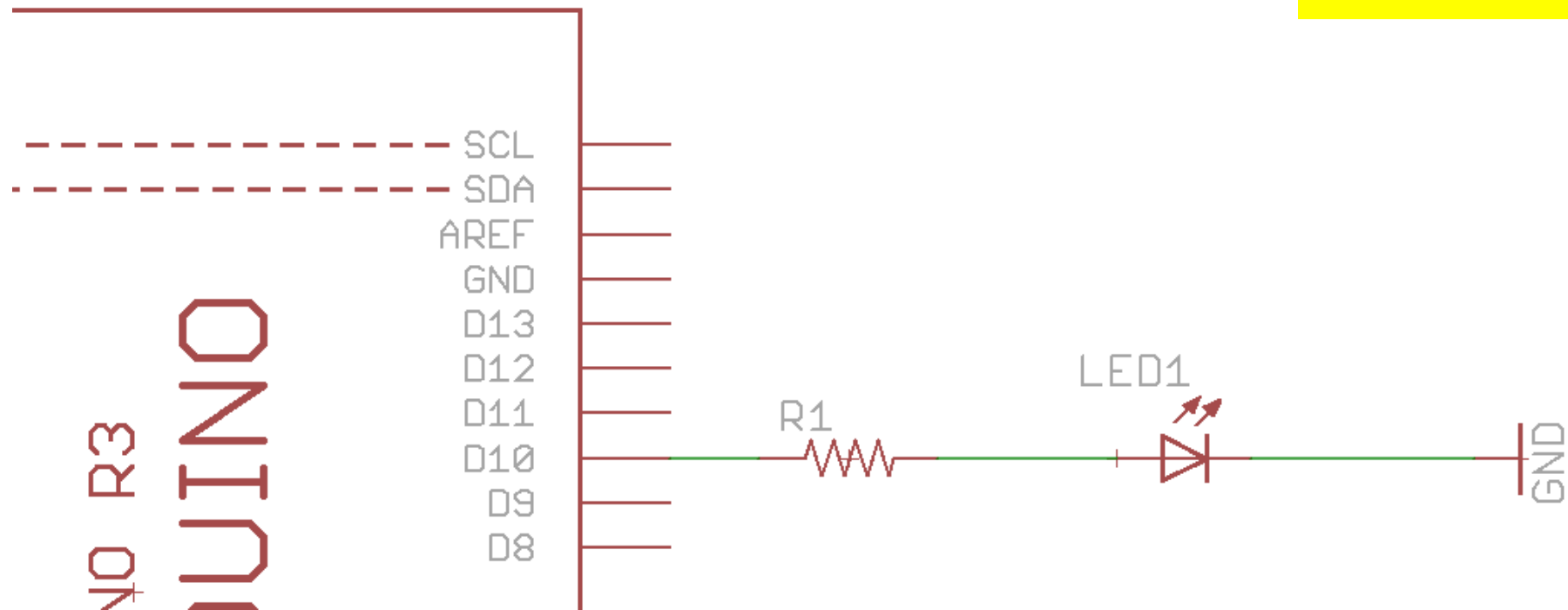




YOUR FIRST EAGLE CIRCUIT

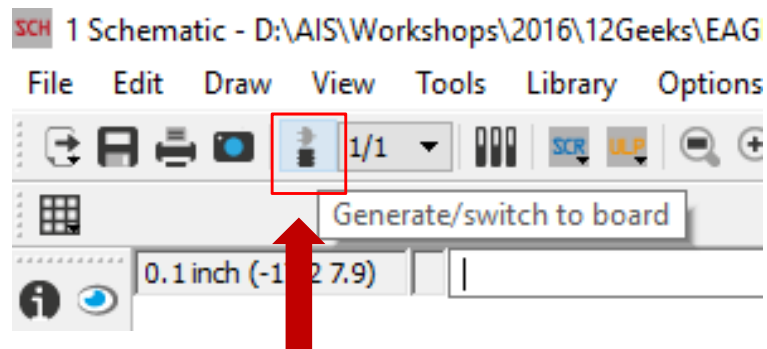
“Hello World” EAGLE style

**TRY DOING THIS
BY YOURSELF**





UNDERSTANDING HOW .SCH & .BRD ARE CONNECTED



EAGLE WALKTHROUGH

LIVE DEMO TIP :

Using **move** to auto-magically generate net wire connections.

Using **copy** to make copies of existing components (eg. GND) to add to the circuit.

Net wire in .sch → air wire in .brd

Schematic
Drawing

Preparing
the board

Part
Placement

Preparing
to Route

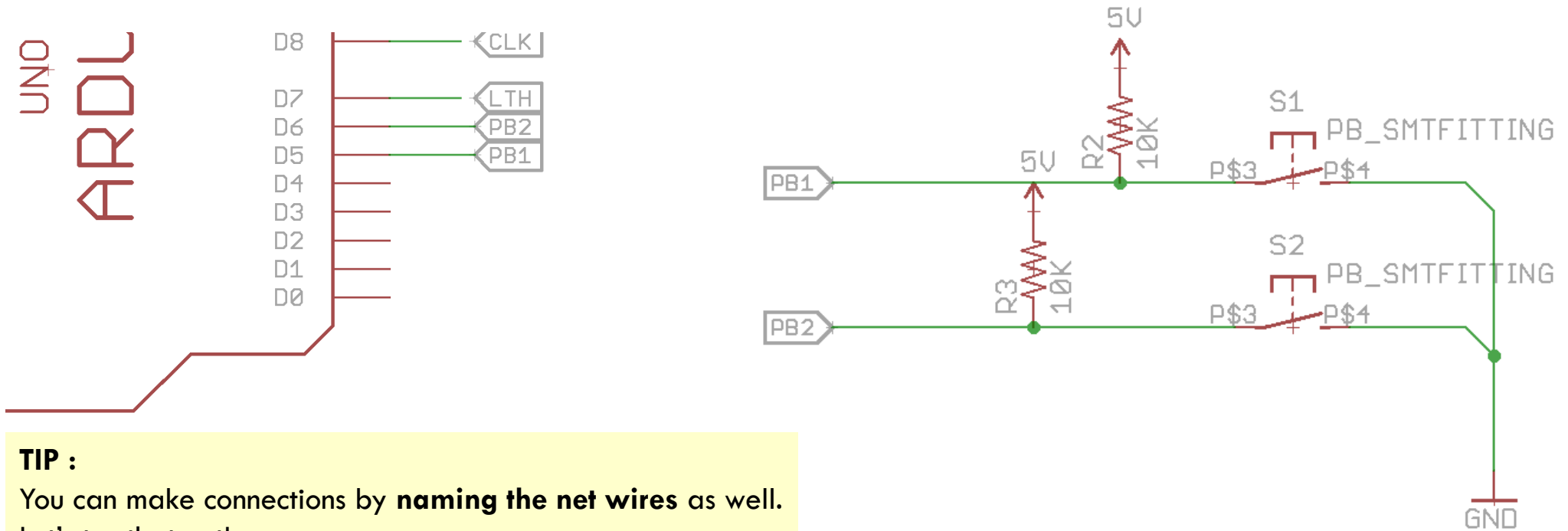
Power
Lines

Signal
Lines

GND
Planes

Silkscreen
Items

LET'S TRY THAT OUT WITH THE PUSH BUTTONS

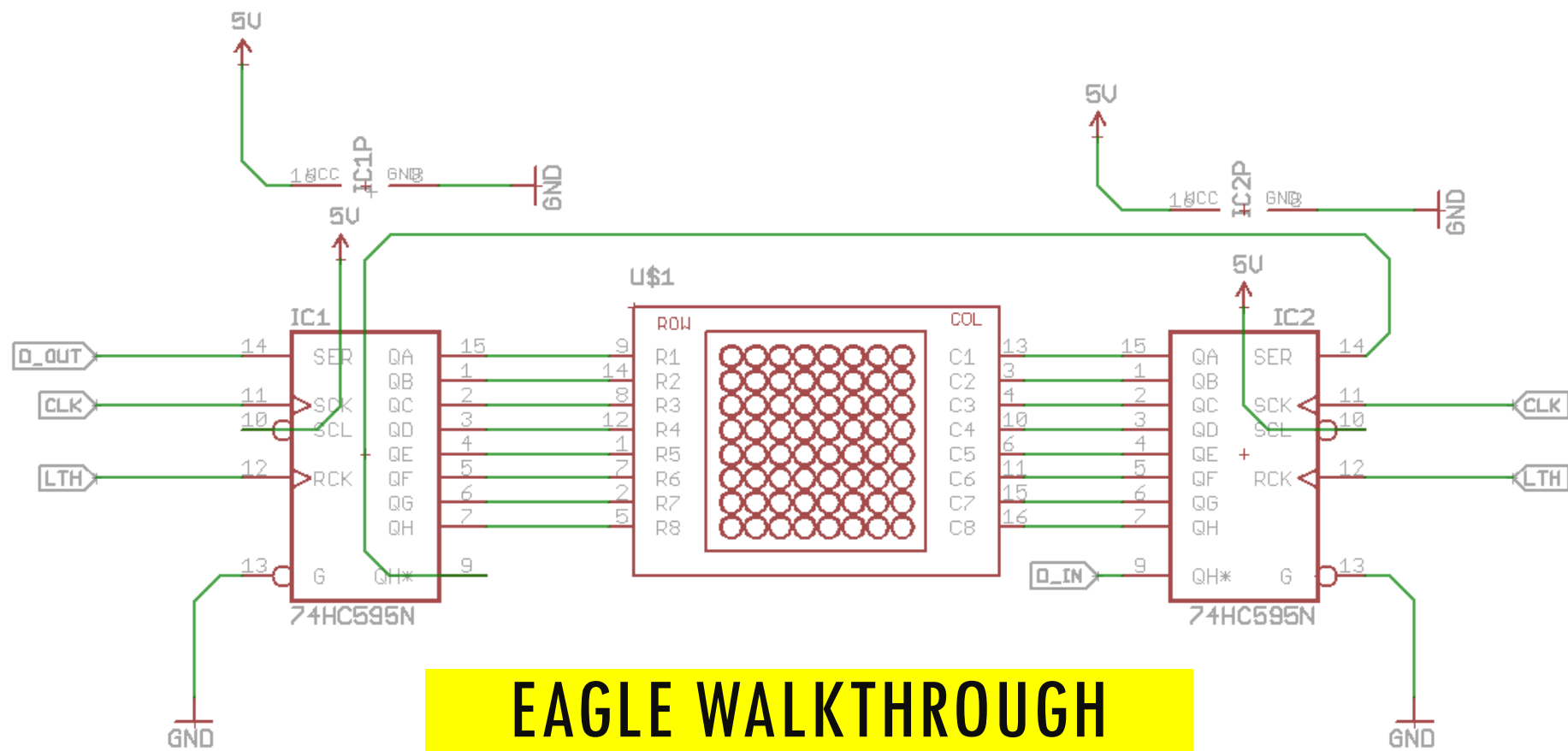


TIP :

You can make connections by **naming the net wires** as well.
Let's try that out!

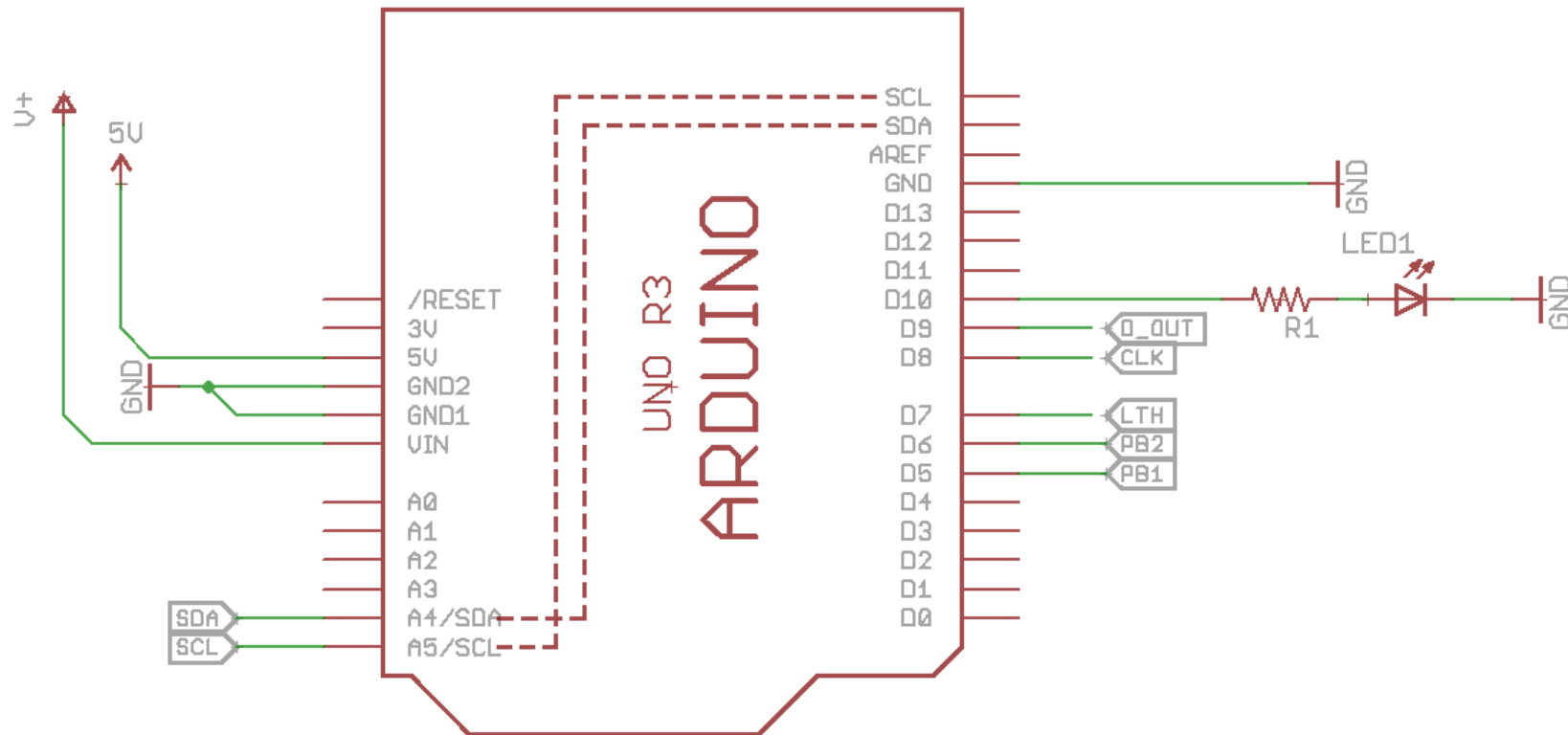


NEW TIPS, AND HERE COMES THE MATRIX





BY THIS TIME, YOUR ARDUINO SHOULD HAVE THE FOLLOWING PIN OUTS



Schematic
Drawing

Preparing
the board

Part
Placement

Preparing
to Route

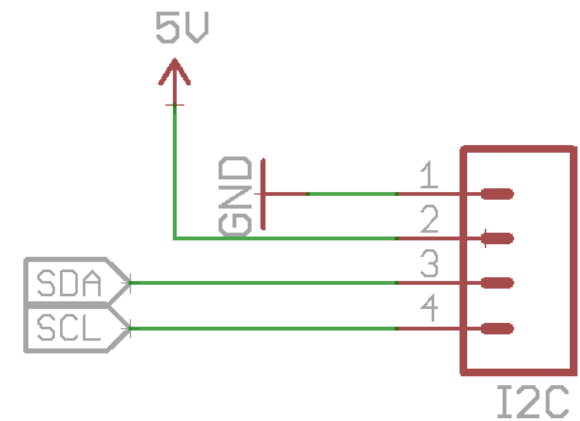
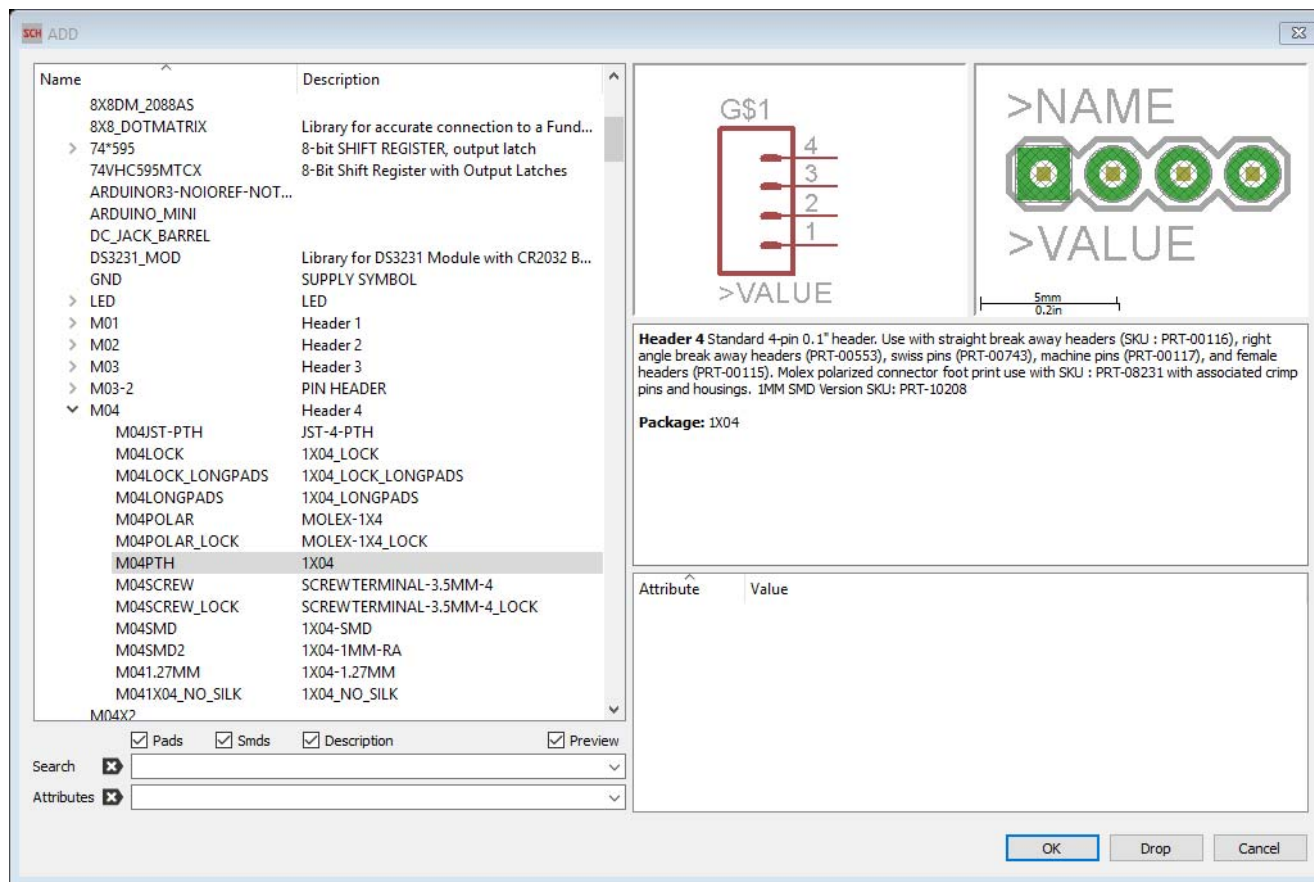
Power
Lines

Signal
Lines

GND
Planes

Silkscreen
Items

~~PANDEMIC~~ PINDEMIC BREAKOUT



(OPTIONAL)
FOR RTC



ERROR CHECKING (ERC)

- Common Mistakes
- Error Descriptions, what they mean, and how to solve it

ERC = Electrical Rule Check

This command is used to test schematics for electrical errors.





**GREAT JOB
GETTING HERE**

LET'S TAKE A BREAK

WAYPOINTS

Schematic
Drawing

Preparing
the board

- Board Dimensions
- Change grid units to mm

Part
Placement

- Good parts placement is half the battle won

Preparing
to Route

- Change grid units to 10 mil
- Setting up route parameters
- Eg. (Bend Style, Width, Via shape, via drill size)

Power
Lines

- Trace width of 32 mil
- Route lines in parallel on bottom layer

Signal
Lines

- Trace width of 16 mil
- Route lines on top layer

GND
Planes

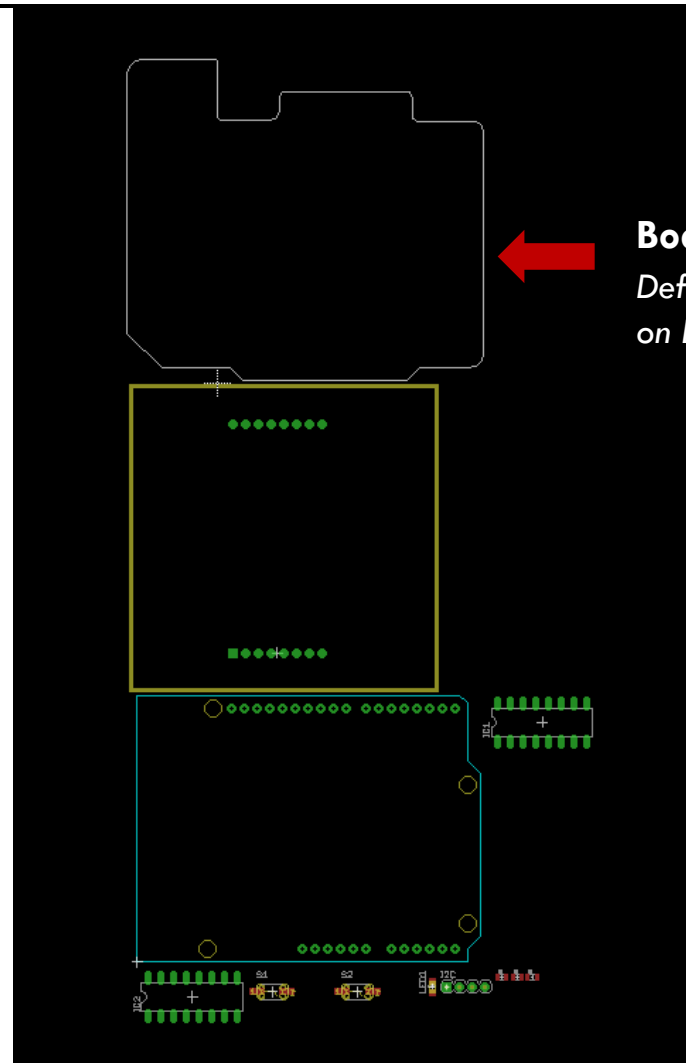
- Polygons, signals and ratsnest

Silkscreen
Items

- Logo
- Labels



Components situated
OUTSIDE of the board



Board Dimension

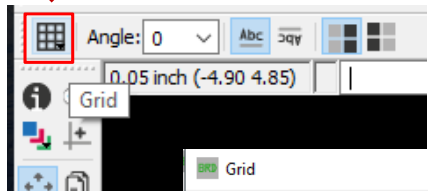
*Defined with Line of width 0
on Dimension Layer*



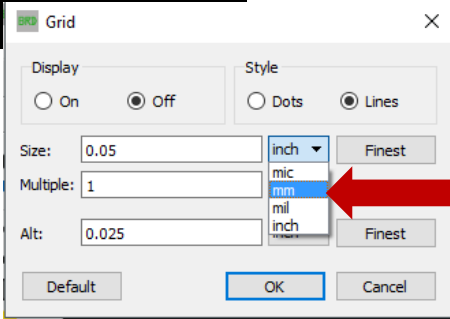
TIP : Working with grid units in mm is great for parts placement to have a sense of measurement

CHANGE GRID UNITS TO MM


1 Select Grid




2 Select mm




3 Change size to 1



4 Select "On" to display grid lines



5 Click OK



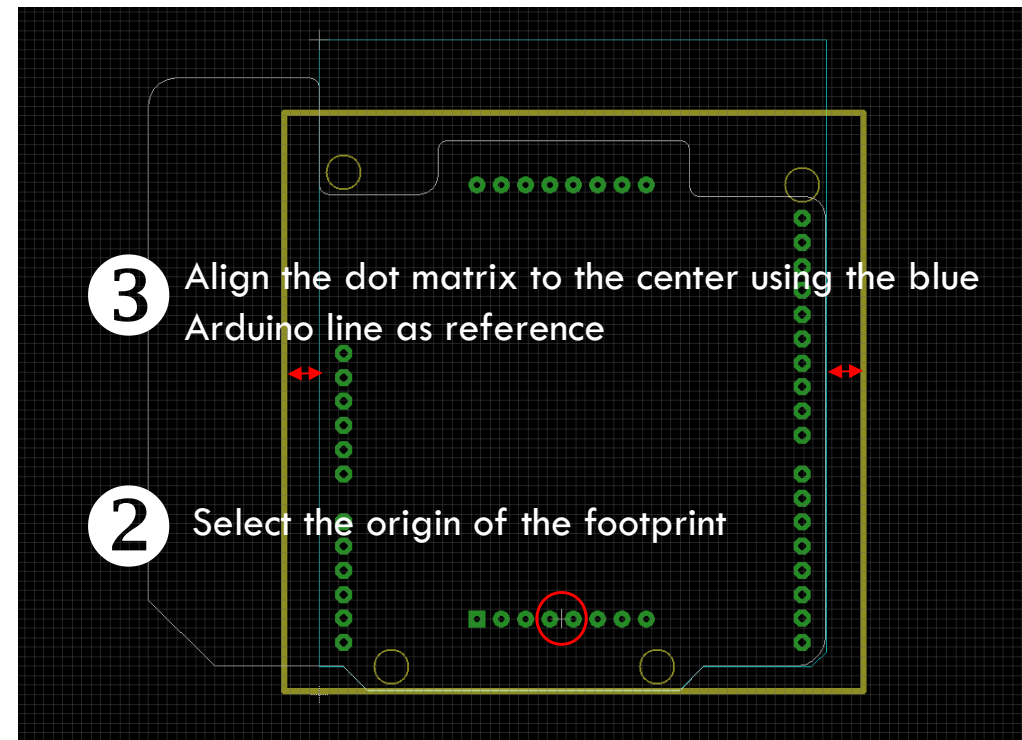
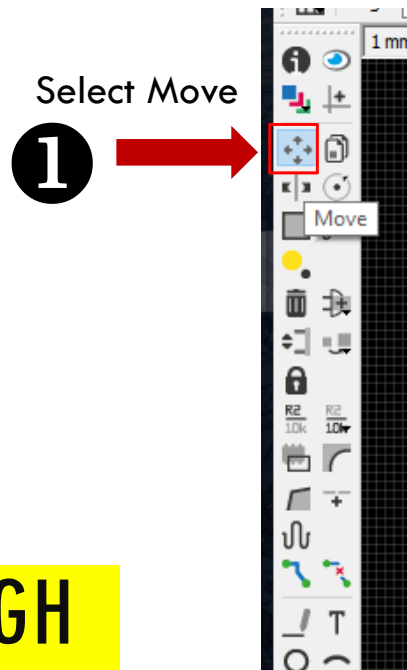


PARTS PLACEMENT :

Good parts placement is half the battle won

TIP : Use air-wires to help indicate the orientation and location of parts that provides the shortest path to route between each other. Click on “Route” if the air-wires are not showing up. Alternatively, enable the unrouted (19) layer.

Since we've already helped you with the quick start for the Arduino layout & board dimensions, the next thing to place would be the dot matrix.



EAGLE WALKTHROUGH



TIP : While deciding on where to place the parts, keep in mind that it should not block the connection to other components/ other components are not blocking it.

Eg. OK to place shift registers underneath the dot matrix module.

NOT OK to place the push buttons or LEDs underneath the dot matrix module.

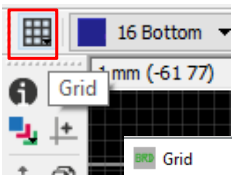


TIP: Working with grid units in **mil** is great for routing (dealing with a number like 10 mil is so much friendlier than 0.25mm)

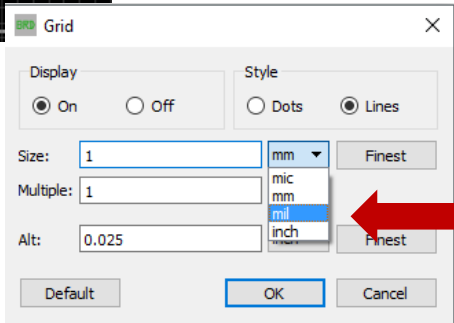
CHANGE GRID UNITS TO MIL

(Unit of measurement for very thin objects - Equal to 1/1000 of an inch)

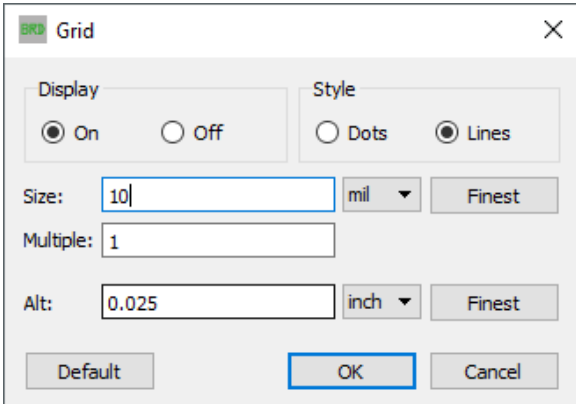
1 Select Grid




2 Select mil



3 Change size to 10



4 Click OK





PREPARING TO ROUTE

Some things you need to know

ROUTING PARAMETERS

Size of drill hole for vias
(unit according to what is set in grid)
Min: 14mil



Select Route

1



Select layer on which
to route trace



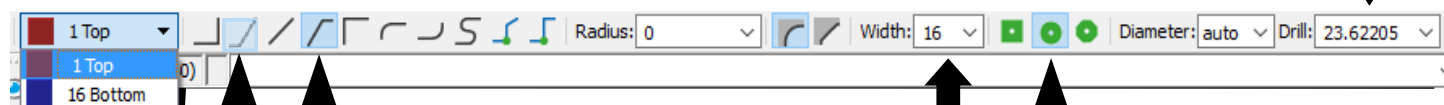
Alternate between these
2 styles to route traces,
they automatically create
45 degree bends



Width of trace
(unit according to what is
set in grid)
Min: 8 mil
*Typical: 16mil



Shape of Via
(square/circle/octagon)
select **circle**





ROUTE POWER LINES FIRST

TIP : Route power lines on the bottom layer of the board, route all signal traces for components on the top layer.

Always run the power lines (+ and -) in parallel to each other.

ROUTING PARAMETERS

1 Select Route

2 Select Bottom Layer

Select layer on which to route trace

Alternate between these 2 styles to route traces, they automatically create 45 degree bends

3 Width of trace (unit according to what is set in grid)

Select 32

Size of drill hole for vias (unit according to what is set in grid)
Min: 14mil

4 Shape of Via (square/circle/octagon)
select circle

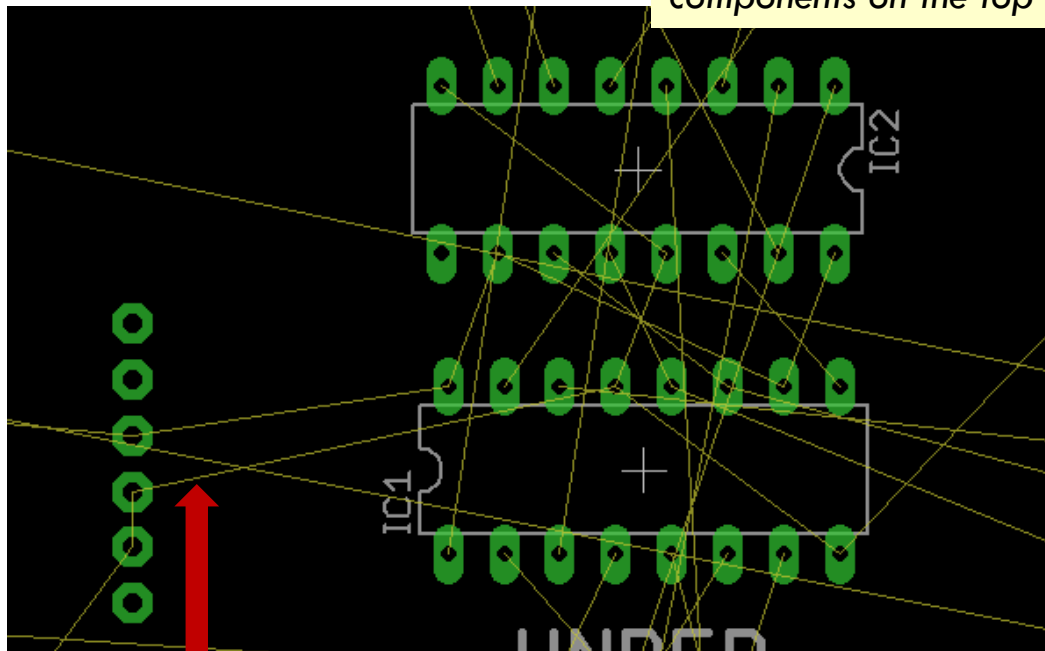
IPC Recommended Track Width For 1 oz cooper PCB and 10 °C Temperature Rise

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

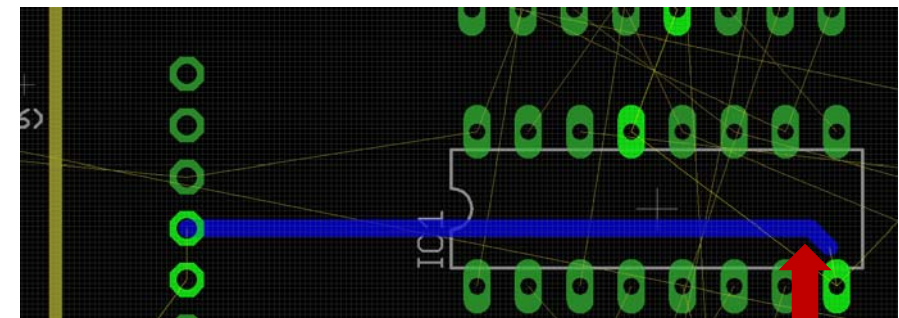


ROUTE POWER LINES FIRST

TIP : Route power lines on the bottom layer of the board, route all signal traces for components on the top layer.

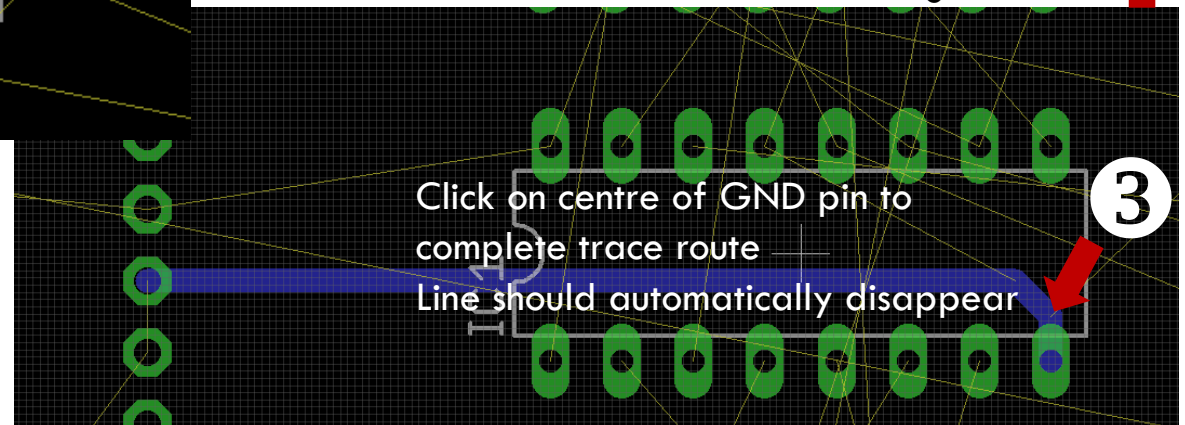


1 Select air wire at GND of Arduino Board



Click to place a bend point before continuing

2



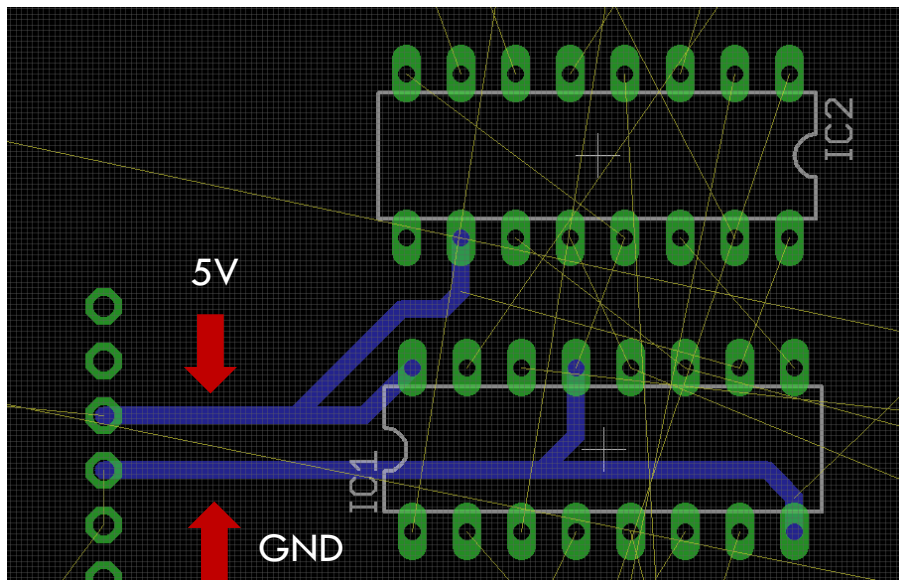
Click on centre of GND pin to complete trace route
Line should automatically disappear

3



ROUTE POWER LINES FIRST

TIP : As much as possible, avoid daisy chaining the power lines from part-to-part.



“Always run the power lines (+ and -) in parallel to each other.”

REPEAT THE STEPS THE OTHER 5V AND GND LINES.

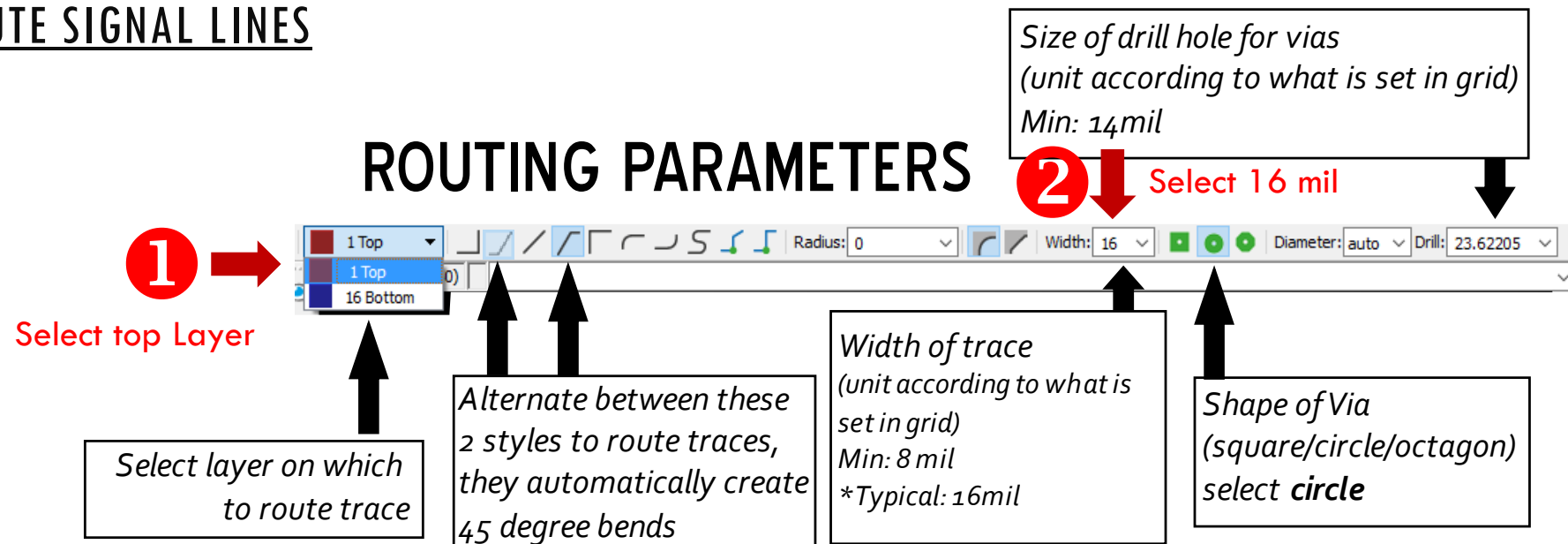
TIP :

*Use the grid lines to gauge **minimum clearance** between the traces.
(That's the reason why we use 10mil)*



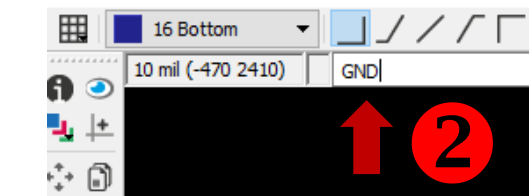
ROUTE SIGNAL LINES

ROUTING PARAMETERS

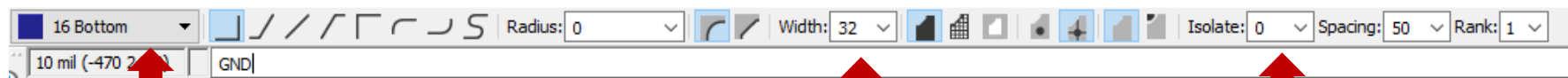


TIP : Is another line blocking? Use Vias!

Eg. SMD on top layer of board needs to connect to a GND on the bottom layer of the board, route a top layer trace from the LED, and select Bottom Layer and continue routing to a GND pin. A via will automatically appear!



1. Type GND
2. Hit Enter



3. Select Bottom Layer
4. Change to 10 mil
5. Change to 16 mil

GROUND PLANES

Large area of copper connected to a circuit's ground point.

- Serves as a return path for current from different components
- Reduces electrical noise, interference, crosstalk
- ..and other functions depending on the circuit

Live Demo

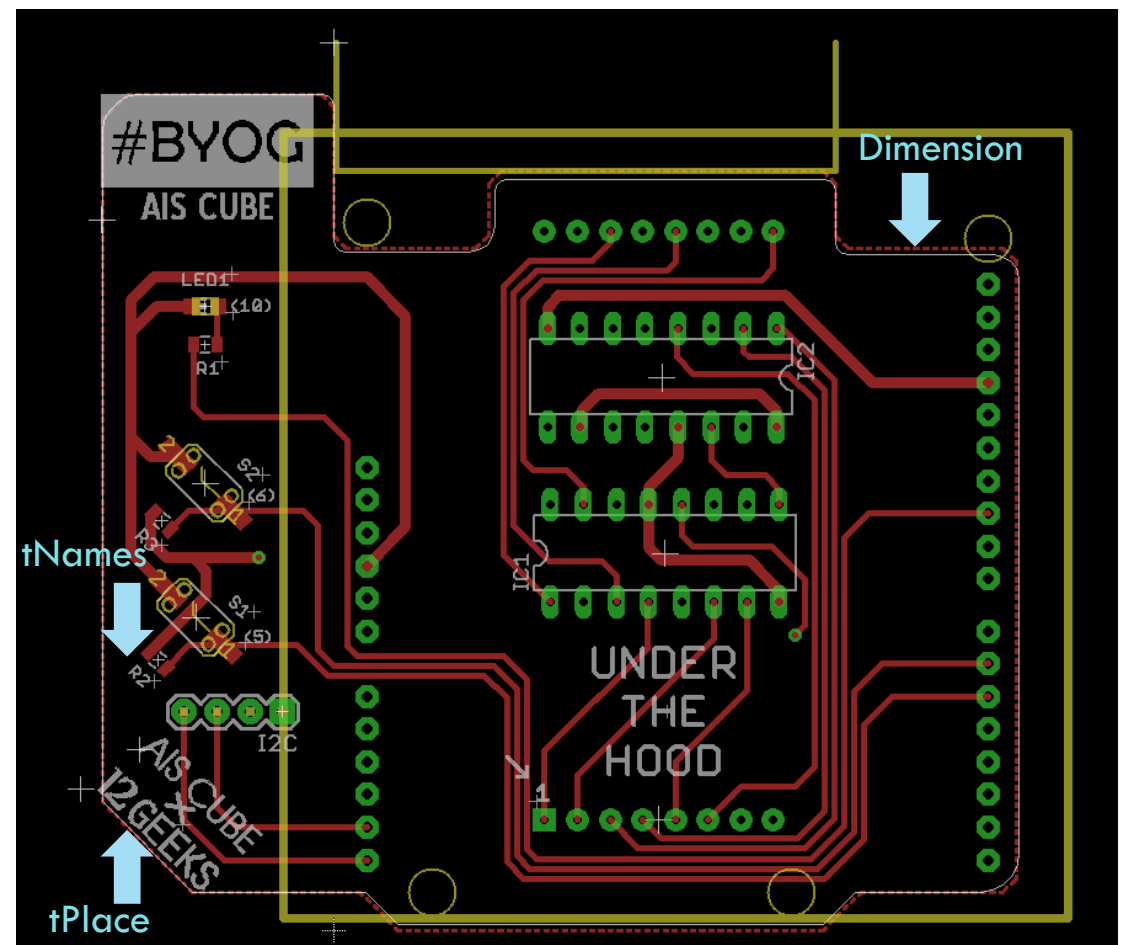
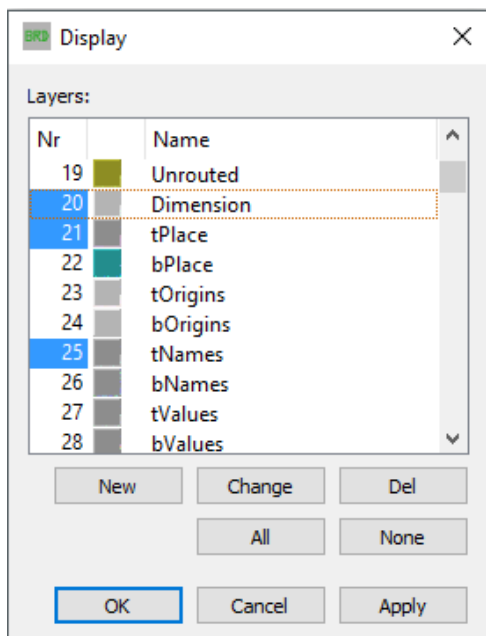
- A. Click and follow the edges of the board to form a polygon
- B. The polygon will turn into a dotted line when closed
- C. Use show on the polygon to check that the signal is GND.
(Your GND traces should be highlighted together with the polygon, if it doesn't, use "NAME" on the polygon to name it as GND.)
- D. Do the same for the Top Layer
- E. Click on Ratsnest to generate solid ground planes



WHAT GOES ON TO THE SILKSCREEN LAYER?

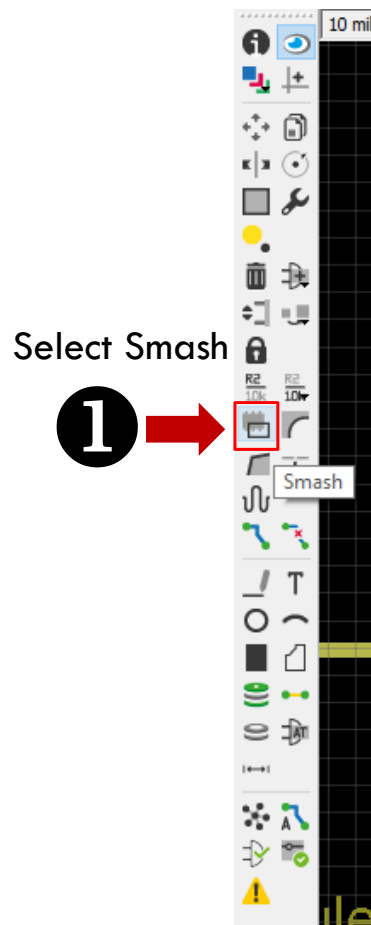
Top Layer: tPlace(21), tNames(25)

Bottom Layer: bPlace(22), bNames(26)



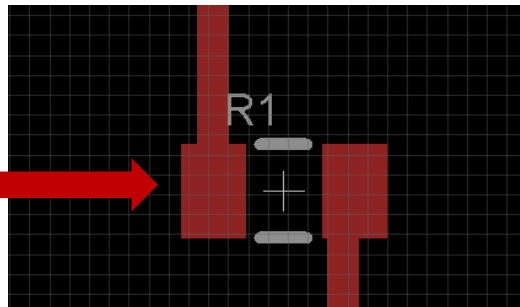


MAKING CHANGES TO THE LABELS



Select R1

2



A cross hair for R1 will appear and make R1 editable.

Right Click R1 > Properties

Under Size, Select 50

3

4

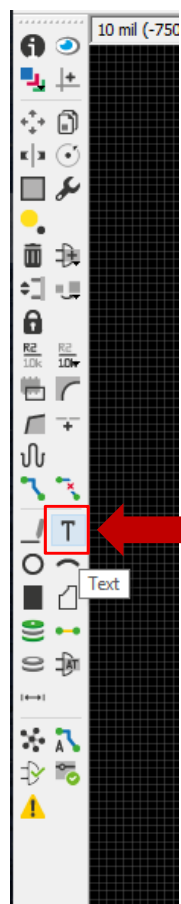
You can now move the Labels as well.

Do the same for the others where required.



WRITING TEXT ON TO SILKSCREEN LAYER

ALWAYS USE VECTOR FONT

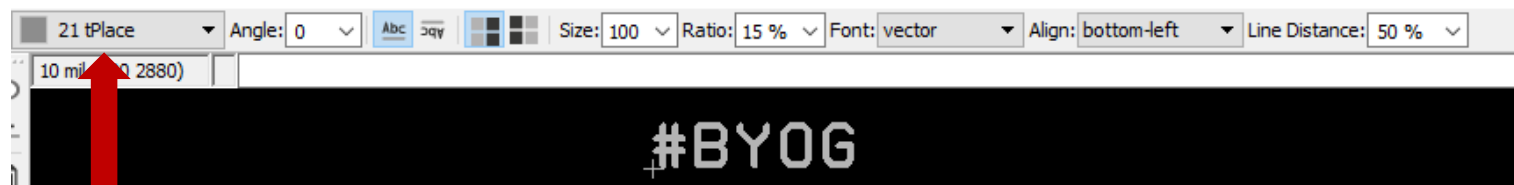


1

Select Text > Write text > Enter

2

Select **tPlace** for **Top** Layer
Select **bPlace** for **Bottom** Layer



3

Play around with size, ratio and font till happy
(Remember: Ratio only works with Font: Vector!)

NOTE:

You may choose to have just text, or just an image, or both on the silkscreen layer.
If you're just placing text, follow the instructions here, and when you're done, go straight to the last slide.
If you wish to place an image, continue till the last slide.

Schematic
Drawing

Preparing
the board

Part
Placement

Preparing
to Route

Power
Lines

Signal
Lines

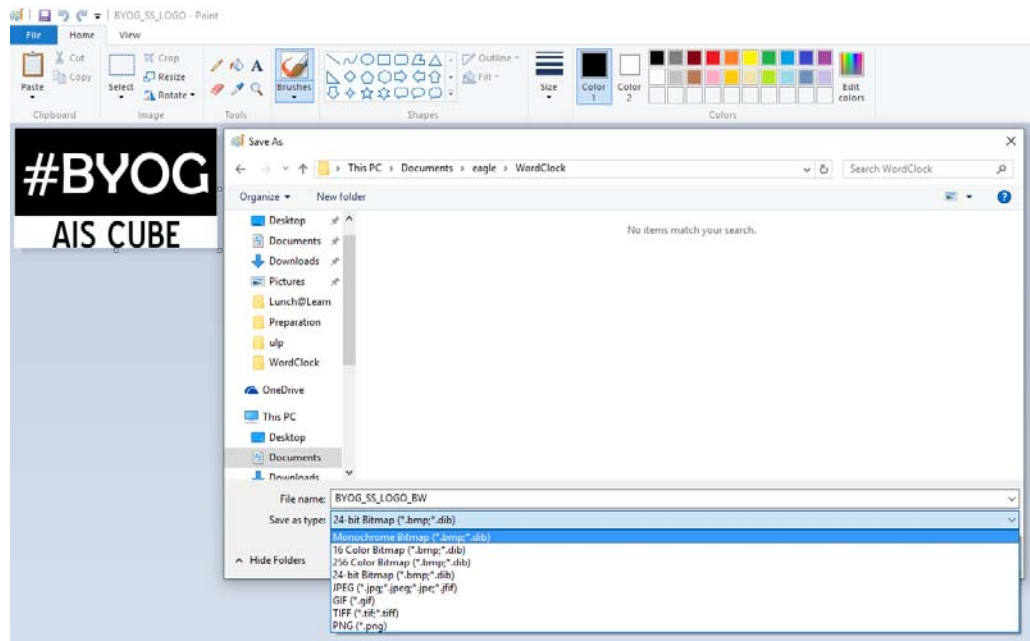
GND
Planes

Silkscreen
Items

PREPARING YOUR IMAGE FOR SILKSCREEN (USING PAINT)

WINDOWS USERS

1. Make sure your image is in .bmp Format, Monochrome (1 bit)
2. (If your image is already in Black & White but not in the above format, use Paint to Save As .bmp Monochrome)



(If you have already done the conversion in PAINT, skip this slide)

Schematic
Drawing

Preparing
the board

Part
Placement

Preparing
to Route

Power
Lines

Signal
Lines

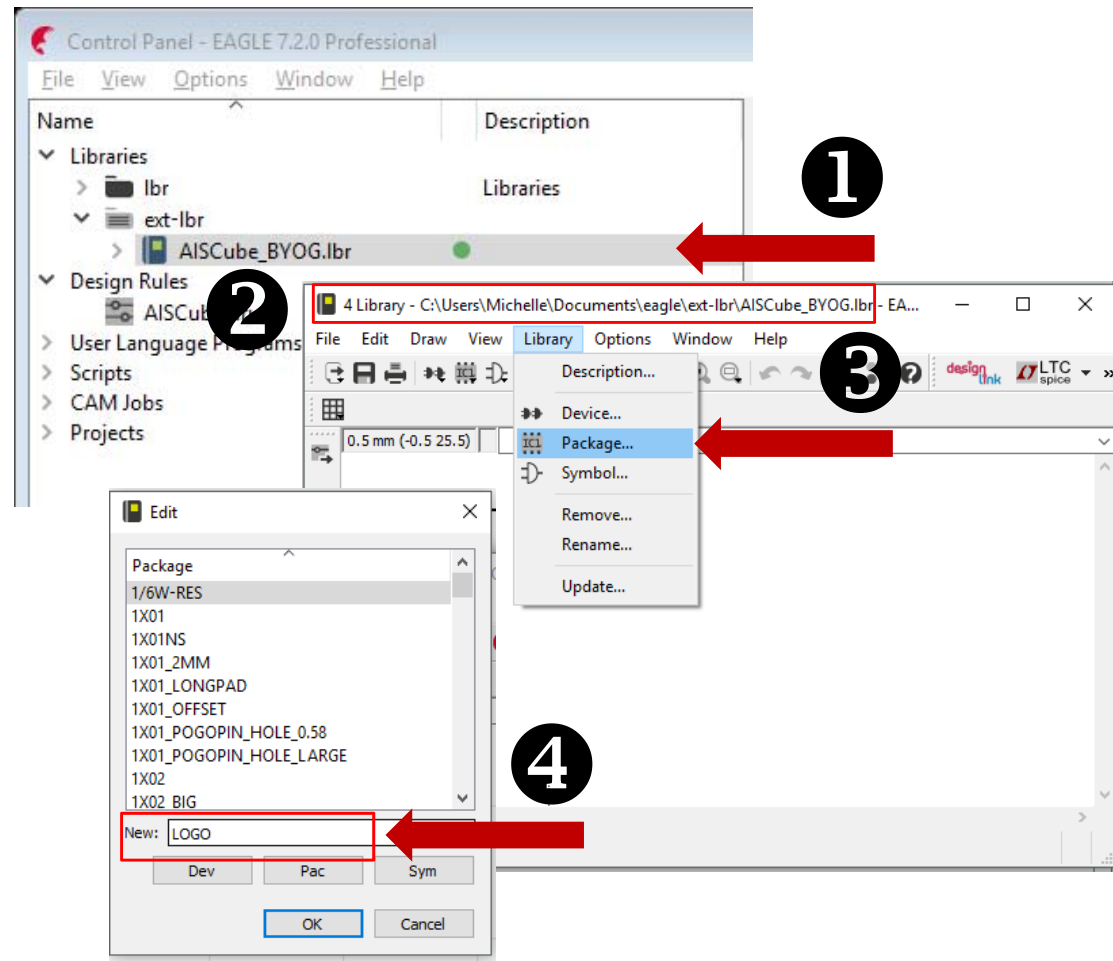
GND
Planes

Silkscreen
Items

ADDING YOUR IMAGE INTO THE LIBRARY

Let's put your image into the library for easy usage

1. Under EAGLE's control panel, Double click on AISCube_BYOG.lbr
2. The library window will pop up
3. Select Library > Package
4. In the textbox next to "New", Name your image as "Logo" (or whatever makes you happy)
5. Click OK
6. A package editor window (that looks like a board layout window) will pop up
7. Proceed to import bmp (next slide)



Schematic
Drawing

Preparing
the board

Part
Placement

Preparing
to Route

Power
Lines

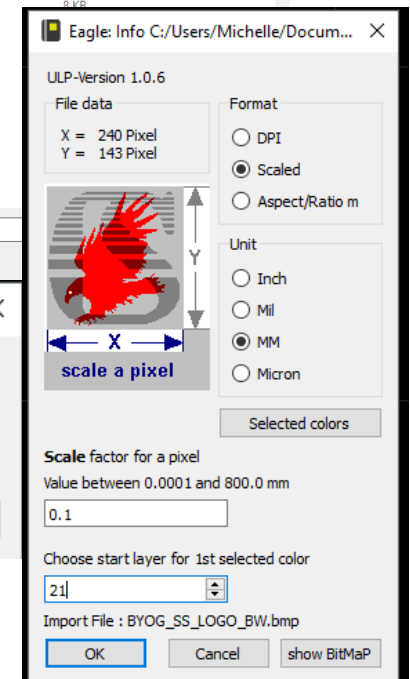
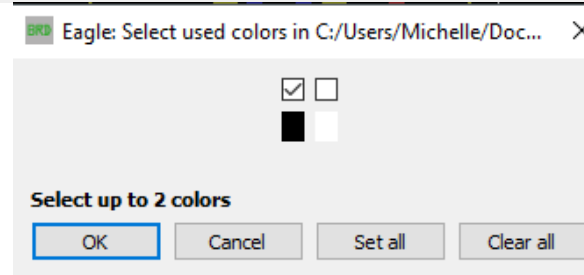
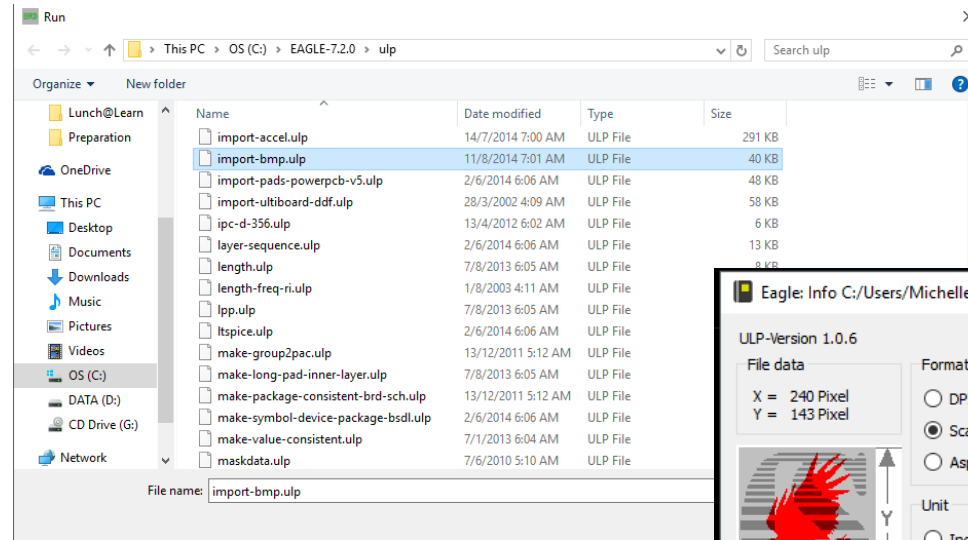
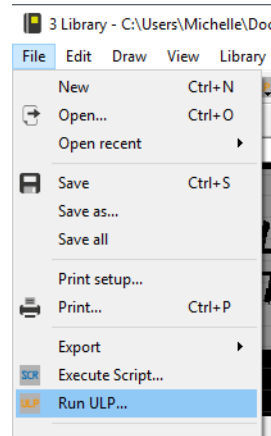
Signal
Lines

GND
Planes

Silkscreen
Items

IMPORT BMP

1. File > Run ULP
2. Select import-bmp.ulp
3. Select your image
4. Info Box > Click OK
5. Select Black > OK
6. Ensure your settings are as follows:
 - Format: Scaled
 - Unit: MM
 - Scale Factor: 0.1
 - Layer: 21 (tPlace)
7. OK > Run Script
8. **Change the Grid** to a value like 10mm to check sizing
9. If you would like to redo the image at a different size, **Delete** the entire **group**, and repeat from Step 1 while playing around with the **scale factor** till you're happy with the sizing
10. Once you're happy with the size, **Save** the library.
11. Go to EAGLE's control panel
12. Update AISCube_BYOG.lbr (right click> update)
13. In your board design, Click on **ADD** > LOGO
14. Place logo onto board



DONE!

Please email your **entire project folder** to
FAZLI@12GEEKS.COM

By **Sunday (13/11/2016) Night 2359 hours**