

Intro to KiCad

KiCon 2019 (based on KiCad 5.1.0)

Worksheet by Shawn Hymel (License: CC BY 4.0)


Name: _____

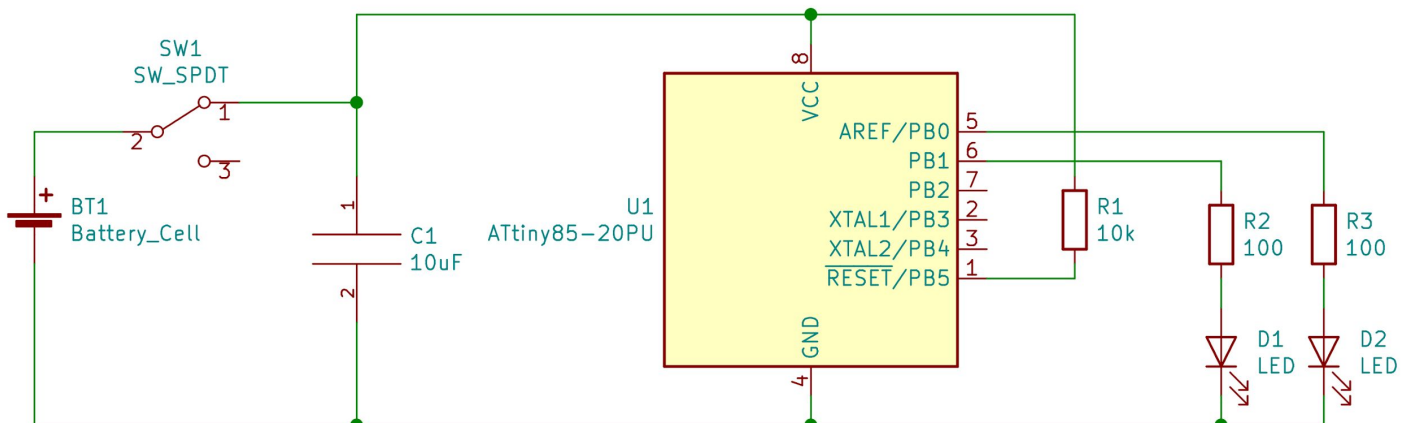
Date: _____

Installation


- Navigate to **kicad-pcb.org/download/** and download the installer for your OS
- Run the installer, accepting all defaults
- Wanna see video tutorials of KiCad? Go here: **bit.ly/LearnKiCad**
- Project files can be found here: **github.com/ShawnHymel/kicon19-blinky**

Schematic Capture

- Start KiCad; you should be looking at the Project Manager window
- File > New > Project, create a new project named **kicon19-blinky.pro**
- Click **Schematic Layout Editor** 
- Place > Symbol, click anywhere in work area
- Search for **ATtiny85-20PU**, click OK
- Place anywhere within schematic sheet
- Repeat for:
 - 1x **Battery_Cell**
 - 1x **SW_SPDT** (for “switch”)
 - 1x **CAP** (for “capacitor”)
 - 3x **R** (for “resistor”)
 - 2x **LED**
- Update component values (Right-click on symbol > Properties > Edit Value...)
 - Change capacitor value from *CAP* to **10uF**
 - Change 2x resistor values from *R* to **100**
 - Change 1x resistor value from *R* to **10k**
- Move parts around to match the schematic in the image below
 - Right-click on symbol > Move to move symbols
 - Right-click on symbol > Orientation > Rotate Clockwise to rotate
- Connect the components as shown in the schematic
 - Place > Wire
 - Click on symbol end to start wire, click on another symbol end to stop drawing wire
- Add reference designators to symbols
 - Tools > Annotate Schematic
 - Keep defaults, click *Annotate* then click *Close*
 - The symbols should have reference designators (e.g. R1, R2)



Assign Footprints to Symbols

- This is where we tell KiCad to associate a PCB part footprint with each schematic symbol
- Click Tools > Assign Footprints...
- Make sure only the *Pin Count* and *Library* filters are selected → 
- Assign **BatteryHolder_Keystone_103_1x20mm** footprint to *Battery_Cell* symbol
 - In *Footprint Libraries* pane, select the *Battery* library
 - In *Footprint Assignments* pane, select *Battery_Cell* symbol (BT1)
 - In *Filtered Footprints* pane, double-click *BatteryHolder_Keystone_103_1x20mm*
- Assign **C_Disc_D5.0mm_W2.5mm_P2.50mm** to *10uF* capacitor symbol
 - In *Footprint Libraries* pane, select the *Capacitor_THT* library
 - In *Footprint Assignments* pane, select *10uF* symbol (C1)
 - In *Filtered Footprints* pane, double-click *C_Disc_D5.0mm_W2.5mm_P2.50mm*
- Repeat this process for the rest of the symbols:
 - From the *LED_THT* library, assign **LED_D5.00mm** to both *LED* symbols (D1, D2)
 - From the *Resistor_THT* library, assign **R_Axial_DIN0207_L6.3mm_D2.5mm_P10.16mm_Horizontal** to the 10k and 100 Ohm resistors (R1, R2, R3)
 - From the *Button_Switch_THT* library, assign **SW_Slide_1P2T_CK_OS102011MS2Q** to the *SW_SPDT* switch (SW1)
 - From the *Package_DIP* library, assign **DIP-8_W7.62mm** to the *ATtiny85-20PU* (U1)
- Make sure your footprint assignments look like the ones below, and click **OK**



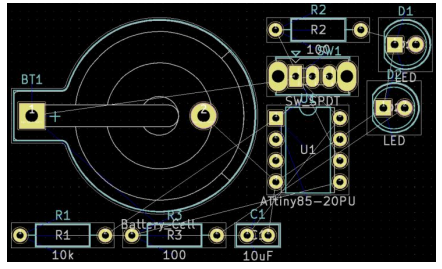
Symbol: Footprint Assignments

1	BT1 -	Battery_Cell : Battery:BatteryHolder_Keystone_103_1x20mm
2	C1 -	10uF : Capacitor_THT:C_Disc_D5.0mm_W2.5mm_P2.50mm
3	D1 -	LED : LED_THT:LED_D5.0mm
4	D2 -	LED : LED_THT:LED_D5.0mm
5	R1 -	10k : Resistor_THT:R_Axial_DIN0207_L6.3mm_D2.5mm_P10.16mm_Horizontal
6	R2 -	100 : Resistor_THT:R_Axial_DIN0207_L6.3mm_D2.5mm_P10.16mm_Horizontal
7	R3 -	100 : Resistor_THT:R_Axial_DIN0207_L6.3mm_D2.5mm_P10.16mm_Horizontal
8	SW1 -	SW_SPDT : Button_Switch_THT:SW_Slide_1P2T_CK_OS102011MS2Q
9	U1 -	Attiny85-20PU : Package_DIP:DIP-8_W7.62mm

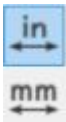
Generate Netlist

- A netlist tells the PCB layout software where everything is connected, according to the schematic
- Click Tools > Generate Netlist File...
- Click **Generate Netlist**
- Save .net file in your project directory
- Close Schematic Layout Editor

Load Netlist into PCB Layout Editor

- In the Project Manager, click **PCB Layout Editor** → 
- Click Tools > Load Netlist...
- Click the *folder* button in the top-right of the window → 
- Select your netlist file (.net), and click *Open*
- Click **Update PCB**
- All your components should be on your cursor
 - Zoom in/out with your mouse wheel
 - Click to place them in the center of the work area → 

Configure Design Rules

- Change units to *inches* by clicking on this button on the left bar → 
- Click File > Board Setup...
- In *Design Rules*, enter the following minimums →
- In *Net Classes*, enter the following:
 - Clearance: 20 mils
 - Track Width: 20 mils
 - Via Size: 32 mils
 - Via Drill: 16 mils
 - μ Via Size: 32 mils
 - μ Via Drill: 16 mils
 - dPair Width: 20 mils
 - dPair Gap: 20 mils

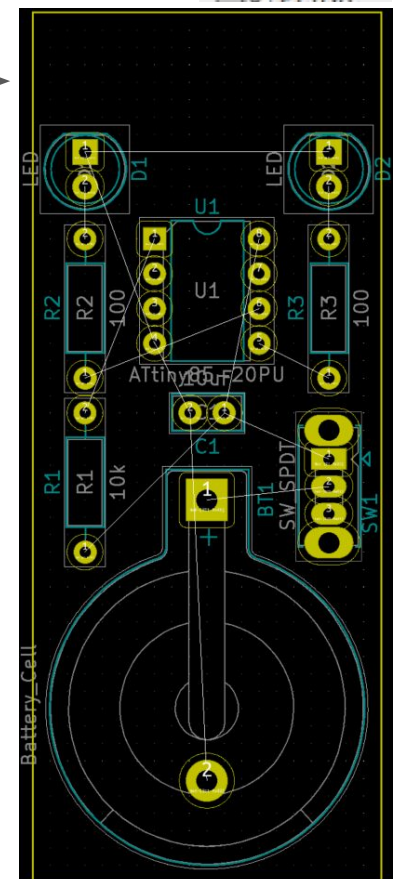
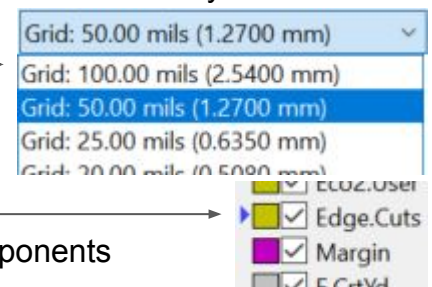
Minimum track width:	20	mils
Minimum via diameter:	32	mils
Minimum via drill:	16	mils
Minimum μ Via diameter:	32	mils
Minimum μ Via drill:	16	mils
Minimum hole to hole:	20	mils

Name	Clearance	Track Width	Via Size	Via Drill	μ Via Size	μ Via Drill	dPair Width	dPair Gap
Default	20 mils	20 mils	32 mils	16 mils	32 mils	16 mils	20 mils	20 mils

- Click **OK**. This sets our PCB trace and drill parameters, and it should match our manufacturing process (you can get these rules from your PCB manufacturer's website)

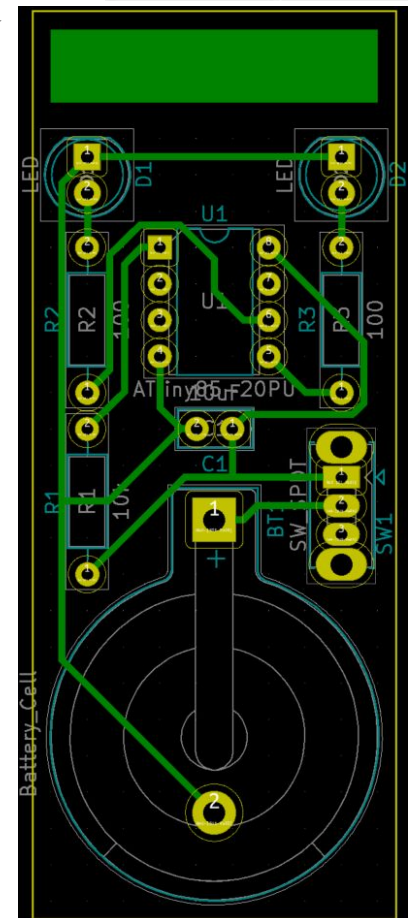
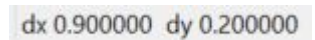
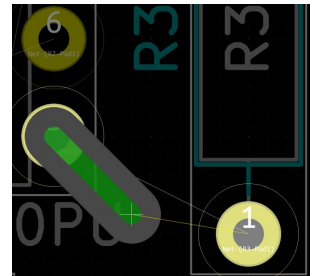
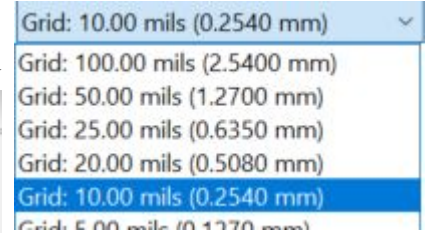
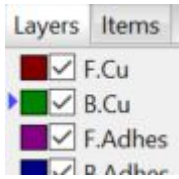
Organize Components

- NOTE: This is a *single-sided PCB*, so our components will be on the *Front* layer and our traces will be on the *Back* layer
- At the top, change the grid to *50.00 mils (1.2700 mm)* →
- Organize the components
 - Right-click on the components, select *Move* or *Rotate*
 - Try to avoid *airwire* crossings as much as possible
- On the right bar, click the *Edge.Cuts* layer to select it →
- Click Place > Line, and draw a board outline around your components
 - Leave 0.3 inches at top to allow for pin back
- Check your board
 - It should look like this →
 - Outline should measure 1.0 x 2.5 inches
 - Check outline dimensions with Inspect > Measure



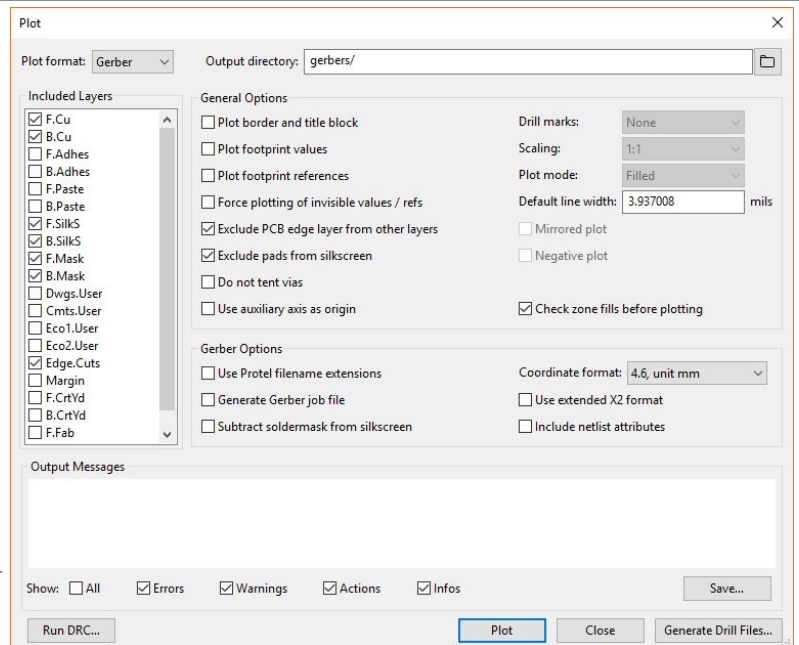
Route Traces

- At the top, change the grid to *10.00 mils (0.2540 mm)*
- Select *B.Cu* layer
- Route traces
 - Click Route > Single Track
 - Click on component through-hole to start
 - Click to place bend in track
 - Click on end component through-hole to end
 - You can also click on parts of a track to connect traces together
 - Where each trace goes will be highlighted by its airwire
 - Let the airwires guide you!
 - Do not cross traces from different nets.
 - Stay on the B.Cu layer
 - Keep connecting component leads together until the airwires are gone
- Add a copper “filled zone” for the pin back
 - Click Place > Zone
 - Click 0.05 in. down and 0.05 in. to the right of the top-left corner of the PCB outline
 - A pop-up will appear, keep all defaults, and click **OK**
 - Press spacebar to set a temporary origin point
 - Draw a 0.9 x 0.2 in. rectangle.
 - Use the dx/dy markers in the bottom-right to help
- Your final board should look like this:
- Check your design
 - Click Inspect > Design Rules Checker
 - Click **Run DRC**
 - You should see 4 errors pointing at the switch footprint, as the holes are too close together (but still OK)
 - You should also see 1 error showing the battery footprint has an incorrect courtyard (this is also OK)
 - Find and fix any other errors you might see
 - Click **List Unconnected**
 - You should see no unconnected items
 - Click **Delete All Markers**
 - Click **Close**

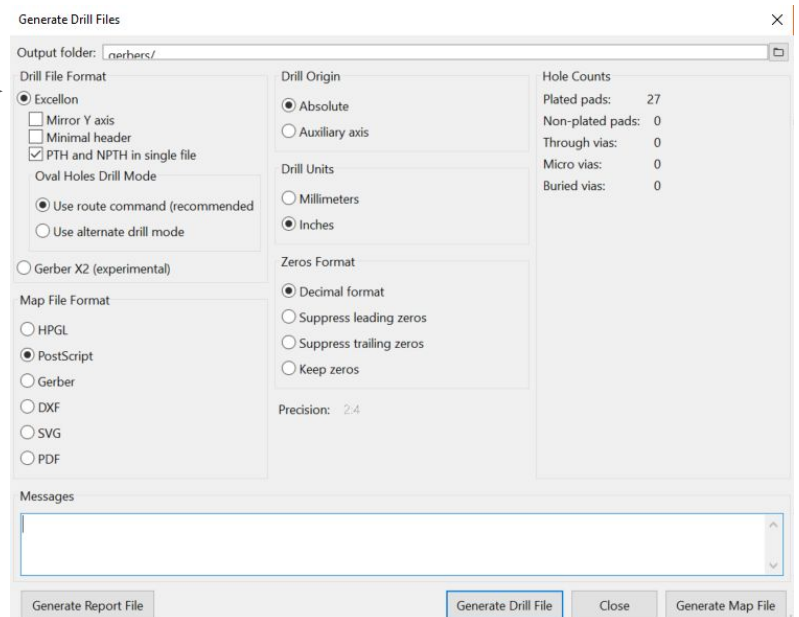


Generate Gerbers

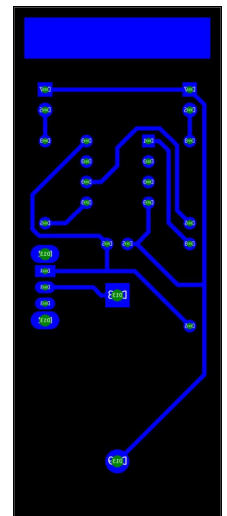
- Create Gerber files (ready for manufacturing!)
 - Click File > Plot...
 - Change *Output directory* to **gerbers/**
 - In *Included Layers*, make sure just the following are selected: **F.Cu, B.Cu, F.SilkS, B.SilkS, F.Mask, B.Mask, Edge.Cuts**
 - In *General Options*, deselect **Plot footprint values** and **Plot footprint references**
 - Make sure your options are as follows →
 - Click **Plot**



- Click **Generate Drill Files...**
- Leave everything as default →
- Click **Generate Drill File**
- Close out of both windows




- Save and exit out of the *PCB Layout Editor* program
- Inspect your drill file and Gerber files
 - Open the **Gerber Viewer** from the Project Manager window
 - Click File > Open Excellon Drill File(s)...
 - Open the .drl file in your *gerbers* directory
 - Click File > Open Gerber File(s)...
 - Open the B_Cu.gbr file
 - Click File > Open Gerber File(s)...
 - Open the Edge_Cuts.gbr file
 - Your drill holes, traces, and board outline should all line up →
 - Feel free to repeat for the other layers (if you plan to have your board professionally manufactured)



Make Your Boards

- Send your boards off for fabrication!
 - One good place is oshpark.com
 - Note that this is a *single-sided board*, which might save you on costs
- Another option: mill boards yourself (if you have a CNC mill)
 - The Bantam Tools Desktop Milling Machine is a good option

Assemble Board

- Place components on the front side (the side without copper)
 - Refer to this assembly guide 
 - Note that U1 should be the DIP socket
 - Solder component leads on back side (side with copper)
 - Hold the safety pin to the copper rectangle on the back side a soldering iron and feed solder into the pin's holes
- Program the ATtiny85 with the fade-blinky Arduino program
 - Code found in GitHub project repository
 - This can be done with another Arduino
 - Alternatively, this board can help: sparkfun.com/products/11801
- Test it!
 - Insert the ATtiny85 into the socket
 - Insert the battery
 - Flip the switch
 - Things should blink! 