

Introduction to PCBA Design

Before starting the design of the PCB, the schematic has to be checked at a schematic review meeting. It is a critical step because unnoticed errors will go to PCB and/or BOM and further. Sometimes these errors go to products, and this becomes a big problem for the company. Let me give you some examples from my practice. Once, on a schematic released to manufacturing I discovered that a signal from a sensor goes to a Schmitt trigger IC, powered by 5V, and the output of Schmitt trigger goes directly to the input of the microcontroller, powered by 3.3V. Quick check of the data sheet showed that the microcontroller did not have 5V-tolerant inputs. It is a recipe for disaster – the PCBA was released, manufactured, working...

Example of One Board Errors

1. The return path for +15, +5, +3.3, -15V was missing in the power supply.
2. A MOSFET, receiving signal from a remote sensor, did not have a Gate to Source resistor and switched randomly.
3. The crystal for microcontroller was 16MHz, although it has to be 25MHz.
4. A logical gate (Schmit trigger) in CANOpen interface did not have its power supply pins connected.
5. COM40 and COMA have not been connected together near the microcontroller. They have not been connected at all.
6. The values of the capacitors for the crystal oscillator were wrong.
7. Many unconnected and unused pins of the microcontroller.

Electronics Design Checklist

- all unused inputs terminated (see slide 2)
- all not-connected pins on IC's should be labeled N/C or X
- mating connectors on assemblies checked for the same pinout
- all outside world I/O lines filtered for RFI (see slide 2)
- all outside world I/O lines protected against static discharge (slide2)
- filtering (“decoupling” or bypass) cap for each power pin of each IC
- voltage ratings of components checked
- each IC has predictable or controlled power-up state after reset
- file name on each sheet (page) of schematic

Electronics Design Checklist, 2

- A dot placed on each intended connection
- minimum number of characters in values (1k, 1.0k, 1.00k versus bad: 1kOhm, 1.0kOhm, 1.00kOhm)
- consistent character size in each category for better readability
- schematics printed at a readable scale
- all components have reference designators, values and footprints
- check pin numbers/types - of all custom-generated parts
- link to part's data sheet entered for each component in BOM
- check polarized components for correct connections
- electrolytic and tantalum capacitors checked for no reverse voltage

Electronics Design Checklist, 3

- power and ground pins are shown for each component (see sl.2)
- check hidden power and ground connections and make them visible
- unused sections of multi-symbol parts have to be present in schematic
- wires with off-page connection must have Global labels
- pullups on all open collector/drain outputs (I2C) exist in the circuit
- sufficient power rails for analog circuits clean (noiseless) power
- amplifiers checked for stability
- oscillators checked for reliable startup (see slide 2)
- consider signal rate-of-rise and fall for noise radiation prevention

Electronics Design Checklist, 4

- check for possibility of input voltages being applied before power -it may cause latchup in CMOS chips
- reset circuit design is reliable, both glitch-free and consistent; tested with slow power supply fall time
- separate analog signals (traces on PCB) from noisy or digital signals
- calculate sufficient capacitance on low dropout voltage regulators
- check the data sheet fine print and appnotes for weird IC behaviors
- determine effect of losing each of multiple grounds on a connector
- automotive powered devices must withstand 60 to 100 volt surges
- check maximum power dissipation at worst-case operating temperatures

Electronics Design Checklist, 5

- check opamp input over-drive response for unintended output inversion
- check maximum common mode input voltages on opamps
- estimate total worst case power supply current
- for buses, ensure bus order matches device order
- ensure resistors are operating within their specified power range plus safety factor
- electrolytic/tantalum capacitor temperature/voltage de-rating sufficient for MTBF
- check for low impedance sources driving tantalum caps which can cause premature failure

Electronics Design Checklist, 6

- hold ROHS compliance requirement review
- hold part obsolescence review
- replacement parts compatibility with original parts has to be done based on the following requirements: same functionality, same footprint, same or better speed and temperature range, ROHS.
- text should not overlap wire or symbol graphics on schematics
- off board connectors identify all signals even if not used on this design
- unpopulated parts annotated and enclosed by dashed-line box on schematics

Electronics Design Checklist, 7

- page title is present and consistent on all pages if not in title block.
- wires exist between all connected pins/ports (no direct pin/pin connections - some capture packages do not like such connections)
- connect wires with one junction dot for three wires max (not four) - look how it is done for U14, power symbol LDO_3V3 and Cap C41 on sheet 8 of IoT board schematic.
- diagnostic resources incorporated in design (LEDs, serial ports, etc.), even if unpopulated by default
- pin names and attributes on symbols with multi-function pins should match actual design usage (I/O/Bi, Name)
- obey preferred component reference designators.

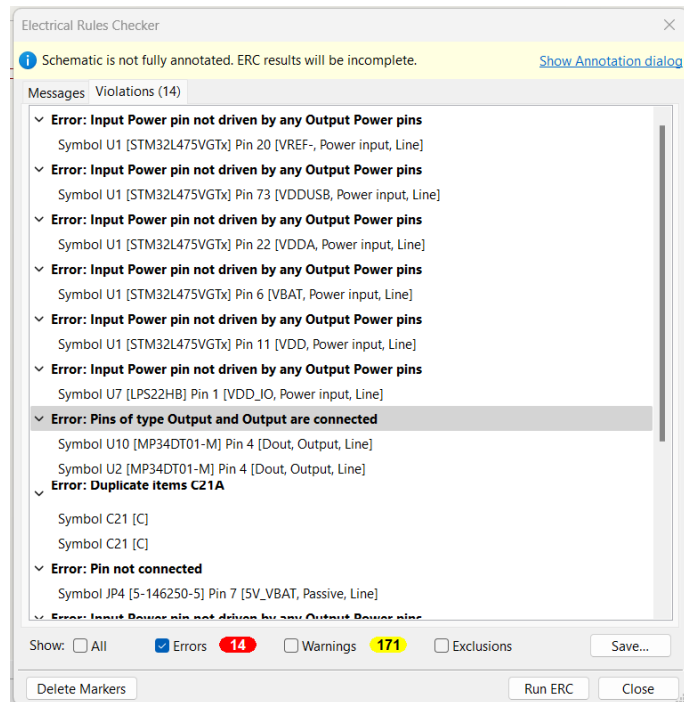
Annotating Components

When all components are placed, connected, and the design is checked, you can annotate the components. Go to Tools→Annotate Schematic, select: Entire Schematic, Sort by Y position, Use first free number after 0; and click on Annotate. Ideal case, if you get a message “Annotation Complete” and no warnings/errors. If you get errors, find and fix them.

Now you can use KiCAD’s utility called Electrical Rules Checker. This checker can be found from the menu bar under Inspect tab.

Go to ERC tab and run the checker. See the following slide displaying the errors. Click on error to view it on the schematic. Errors are red arrows, warnings – yellow.

Electrical Rules Checker



Assigning Footprints

Every part must have a footprint assigned to it – to be placed on a PCB. To check that every part has a footprint go to Tools → Edit Symbol Fields. This is a convenient tool to assign footprint to similar parts: resistors and capacitors - simply Copy-and-Paste. Vast majority of footprints are standardized and can be found in appropriate libraries. If you cannot find the footprint, we can create it using Footprint Editor. I will create one for microphone, so you know how to do it. Notice, that **every multisymbol part has a single footprint**, and all symbols of multisymbol part have to be present on the schematic (see slides 2, 5). Footprints names (to enter as properties) for RF modules can be taken from snapeda.com. You do not need to create footprints for them.

Creating Footprints

From the Project Manager menu Tools, click on Edit PCB Footprints. In the Footprint Editor window click on Preferences → Manage Footprint Libraries. Click on Project Specific tab, Append Library, type the name MyFpLib and select the project subdirectory. To make a footprint HCLGA-4LD 3x4x1 for microphone MP34DT01:

From Project window, click on Tools → Edit PCB Footprints. From the Menu File select New Footprint. Enter the name HCLGA-4LD 3x4x1. On the left side of the new window click on Grid and mm icons. From the menu View → select Grid Settings and change sizes X and Y to 0.1mm, then change Grid1 to 0.1mm and Grid 2 to 1.0mm. Click OK. See the next slides for reference.

Specifying Grid Settings

Grid Settings

Grid Origin

X: 0 mm

Y: 0 mm

User Defined Grid

Size X: 0.1 mm

Size Y: 0.1 mm

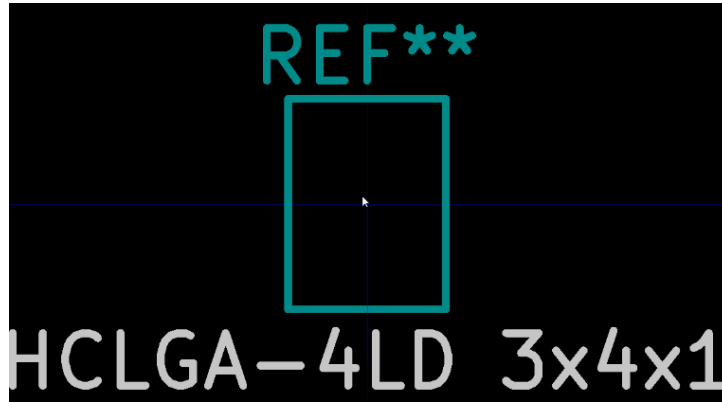
Fast Switching

Grid 1: Grid: 0.1000 mm (0.0039 in) (Alt+1)

Grid 2: Grid: 1.0000 mm (0.0394 in) (Alt+2)

Reset Grid Origin OK Cancel

Outline of Microphone's Footprint



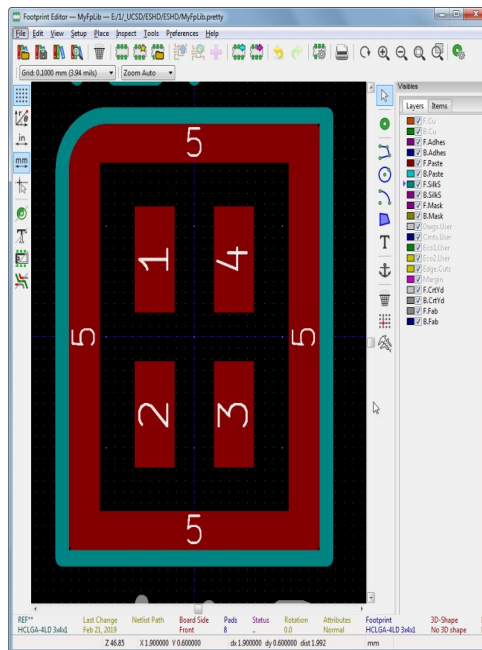
Make sure you are drawing outline on Front Silk Screen layer (see a stack of layers on the right side of the window)

Creating Outline and Pads

From Menu select → Place → Line. Looking at the bottom of the window, select a starting point $X=-1.5$, $Y=-2.0$ mm and draw a rectangle 3mm wide and 4mm high. Move REF** above the rectangle, HCLGA – below. Menu → Place → Pad, move cursor to $X=-0.5$, $Y=-0.7$ mm and click. Press Esc key to end Pads placement.

Double click on the pad and change Pad type to SMD, Shape – to Rectangular, Position X – to -0.45, Size X – to 0.45, Size Y – to 0.95. Click OK. Right click on the pad and select Duplicate, move the copy down to $X=-0.4$, $Y=0.7$. Double click on the new pad, change the number to 2, verify X position= -0.45. Continue above mentioned steps to create pads 3 and 4. Compare your footprint with next slide.

Footprint of HCLGA-4LD



Generating BOM

Fortunately, new KiCAD generates csv (comma separated values) files, so let's try to create the BOM for the Project.

From Menu: Tools click on Generate Bill of Materials. This will open a small window where you can select: bom_csv_gouped_by_value.

Now click on Generate button, and the file will be created in your project folder. You can open this file with Excel, or LibreOffice Calc.

Put "Item#" in A1 cell, "Reference" in B1, add a new column C - "Quantity", next column D will be "Value, then E - "Footprint", then F - "Manufacturer", G - "MFG Part#", H - "Datasheet".

You may see in some BOMs such properties as "Price", "Price on PCBA", and so on – added for convenience of MFG people.

Fragment of Real Life BOM

BOM, 30kV Power Supply, 3-05-2008					
Item	Qty	Reference	Description	PCB Footprint	MANUFACTURER
1	14	C1,C3,C14,C16,C17,C32,C33,C34,C35,C36,C37,C38,C39,C45,C26,C27,C28	CAP, CER, X7R, 0.1uF, 20%, 50V, 2012(0805)	0805	KEMET ELECTRONICS CORP.
2	6	C2,C18,C29,C30,C41,C44	CAP, CER, X7R, 1.0uF, 20%, 25V, 3216(1206)	1206	KEMET ELECTRONICS CORP.
3	2	C4, C24	CAP, POLY FILM, 0.068UF, 10%, 200V, DISC	DISC / L 12 MM / LS 7.5 MM / 0.6MM	PANASONIC ECG
4	16	C5,C6,C7,C8,C9,C10,C11,C12,C19,C20,C21,C22,C25,C26,C27,C28	CAP, CER, B, 1000pF, 20%, 10kV	DISC / L 14 MM / LS 10 MM / 0.8MM	MURATA
5	2	C31,C13	CAP, CER, C0G, 1000pF, 5%, 50V, 2012(0805)	0805	KEMET ELECTRONICS CORP.
6	1	C15	CAP, ALCL, 100uF, 20%, 50V, TH, RAD / DIA 8.0MM / LS 3.5MM / 0.6 MM	RAD / DIA 8.0MM / LS 3.5MM / 0.6 MM	ILLINOIS CAPACITOR

An example of a BOM for a HV Power Supply

Generating Netlist

From any schematic page go to Menu → File → Export -> Netlist. In the small window, named Export Netlist, click on KiCAD tab. Then click on Export Netlist button. By default, it will go to your Project subdirectory but can be placed anywhere you want.

In most of the companies, this is all you have to do to transfer your design (schematic, BOM, Netlist, Datasheets) to a layout person.

The layout person will draw the PCB outline, select number of layers, place the components, make the layout, and generate Gerber and Assembly files.

The set of Gerber files (including drill files, silkscreens, etc) will go to a PCB house to manufacture PCBs.

Some Remarks on PCB Name

You can find a definition PWB – Printed Wiring Board – applied to a PCB. However, modern PCBs have the following elements of circuits (thus the name “Circuit”): inductors (including power transformers of PS), made of the PCB’s copper; capacitors, made from copper on neighbor layers, divided by the substrate material (FR4, for example); transmission lines with predefined impedance; and antennas, etched in the copper. In fact, our IoT board has a Wi-Fi Module with an antenna etched on its small PCB. And the antenna for Near Field Communication radio is etched on the main IoT PCB.

PWB is a 60-year old technology, when manufacturers used solid transformer wires molded into plastic boards – to connect mounting holes for components. Think about it and do not use PWB for PCB.