*Projects and Stuff*

Design Checklist

& Standardization Manual

Contents

[General Notes 3](#_Toc333437037)

[Schematic Design - Page 4](#_Toc333437038)

[Schematic Design - Drawing 5](#_Toc333437039)

[Schematic Component Design 7](#_Toc333437040)

[PCB Layout and Design 8](#_Toc333437041)

[Module Component and Pad Design 9](#_Toc333437042)

[Gerber Review 10](#_Toc333437043)

[Physical Board Inspection 11](#_Toc333437044)

[Firmware Design 12](#_Toc333437045)

[Documentation 13](#_Toc333437046)

[Project File Structure and GitHub Tips 14](#_Toc333437047)

[References 15](#_Toc333437048)

# General Notes

This document is intended to serve as a reference for standardizations, and as a set of checklists to guide projects from the beginning stages through the finished product.

# Schematic Design - Page

* Schematic drawing has Date, Project Name, “Projects and Stuff”, and Revision Number
* All Drawings are on sheet sizes A (11”x8.5”) or B (17”x11”)
* Layout notes have been added in order to improve clarity when laying out the PCB, including, but not limited to, the following situations:
  + Specific trace widths are needed
  + Components are socketed, rather than soldered
  + Specific placement location for a particular component
  + Specific signal length requirements
  + Noting labels that should be placed on the PCB
* The *Projects and Stuff* logo and *Open Hardware* logo have been added to the schematic drawings
* Every component in the schematic is accounted for on the Bill of Materials (BOM)

# Schematic Design - Drawing

* Power nets (VCC, GND, etc) are global across entire design and not unique per sheet
* All parts have Reference Designator values annotated as follows:
  + R – Resistor
  + C – Capacitor
  + L – Inductor
  + D – Diode or rectifier
  + Q – Transistor, FET, SCR
  + U – Integrated Circuits
  + X – Crystal
  + S – Switch
  + F – Fuse
  + FL - Filter
  + J – Jack
  + P – Plug
  + VR – Voltage Regulator
  + BT – Battery
  + W – Wire, Jumper, Specific Traces
  + T – Transformer
  + K – Relay
  + TP – Test Point
* All parts have values assigned as appropriate
* If a component value is not yet known, use the appropriate prefix, as above, and append “SEL”. For instance CSEL for a capacitor of unknown value, and RSEL for a resistor of unknown value
* Connector pin-outs are verified
* All outside world I/O signals are filtered for RFI
* All ICs have appropriate decoupling capacitors at power input
* Pull-up resistors are placed on all open collector outputs
* All unused inputs on integrated circuits are terminated
* Sufficient power rails (generally including 0.1uF and 10-22uF Capacitors in power block)
* Analog blocks are separated from Digital blocks
* Indicator LEDs are pulsed or seriously current limited to reduce wasted power
* All polarized parts have the polarization clearly shown on the schematic, and the polarization shown in the schematic has been verified
* Extra pins on microcontrollers are run to an extra jumper/connector to ease in future modifications, and pull-ups/pull-downs are used. As noted later, using spare IO for at least one switch and one indicator light is highly recommended.
* Mounting holes and fiducials are annotated in the schematic
* The Electric Rules Check (ERC) has been completed and thoroughly verified
* The Netlist has been updated from the most current version of the schematic
* Unless it affects schematic readability, signals flow from left to right, and from top to bottom.
* For large designs, the first page of the hierarchal schematic serves as a block diagram
* Lines are added around blocks of components that serve a specific function in order to distinguish circuit operation
* Busses are used when it improves schematic readability
* All important Nets are explicitly named

# Schematic Component Design

* When designing microcontroller applications, it’s a good idea to use spare IO lines to include at least one button and one indicator light. This way you can test that the microcontroller is addressable, and can respond to inputs and provide output. The switch and indicator can then be used for other applications if needed.
* Include a power-on indicator light
* Include an on-off switch, where applicable

# PCB Layout and Design

Content pending.

# Module Component and Pad Design

Content pending.

# Gerber Review

* An ERC check does not guarantee that there will be no unconnected traces. Double-check the Gerber for traces with no end/termination

# Physical Board Inspection

Content pending.

# Firmware Design

Content pending.

# Documentation

Content pending.

# Project File Structure and GitHub Tips

General Structure:

**MAIN FOLDER**

**./Notes**

Contains the Project Log, Bill of Materials (BOM) and other notes

**./Electronics**

Contains the schematics and PCB design

**./Firmware**

Contains the code required for programming the project

**./Hardware**

Contains CAD drawings and photos of hardware such as enclosures, gears, inserts, etc.

**./GerberOutputs**

Contains the Gerber files for manufacturing PCBs

# References

Content pending.