Projects and Stuff

Design Checklist

& Standardization Manual

Contents

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
- ▢ 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

Content pending.

PCB Layout and Design

Content pending.

Module Component and Pad Design

Content pending.

Gerber Review

Content pending.

Physical Board Inspection

Content pending.

Firmware Design

Content pending.

Documentation

Content pending.

Project File Structure and GitHub Tips

Content pending.

References

Content pending.