# KiCad Style Guide

A brief style guide for KiCad and general electronics design.

## Project

### Creating KiCad Project

1. KiCad projects should be created in their own folder (usually within the Hardware folder of a project). Note that KiCad will create the project folder for you.
2. Once KiCad creates the project, Add two more

## Schematic

### Starting Schematic Capture

1. When starting your schematic, start with the *default\_layout.kicad\_wks* worksheet at size A4. Increase size of paper incrementally as need be up to maximum size A2. Anything larger than A2 should be done as a hierarchical, multi-sheet schematic.
2. Set dimensions to use imperial units.
3. Use only Grid Select sizes of 100 mil, 50 mil, or 25 mil.

### Schematic Capture

1. TODO

### Library Management

1. TODO

### Creating Schematic Symbols

1. TODO

## Bill of Materials

### BOM Generation

1. TODO

### BOM Format

1. TODO

## Board Layout

### Component Placement