# Panel Specifications

## Panel Dimensions

* Panel height shall be 5.060"
* Panel width will be an integer multiple of 0.2"
* Mounting holes shall be 340 mils from horizontal edge and 120 mils from vertical edge

## Panel Text

* Panel text shall be KICAD FONT ALL CAPS
* Panel text shall always be below any knob, switch, jack, or other component
* Text shall be center justified both vertical and horizontally and placed in center of rectangle on user drawing for below
* KNOB text shall be 90MILx90MIL 13MIL thickness as space allows
* THONKICONN text shall be 70MILx70MIL 10MIL thickness
* IN and OUT shall be used instead of input and output
* Module title shall be placed at the bottom of the panel spaced evenly between both edges

## Knob Specifications

* Knobs shall have minimum 1" spacing between each other as measured from the center of each knob.
* The knobs shall be spaced 0.5" from the panel edge.

# PCB Specifications

## PCB Dimensions

* PCB height shall be 4.31"
* PCB width shall be 80mils less than the width of the panel with 40mil space on each side.
* The top of the PCB shall be 0.430" from the top of the panel

# Library Specifications

## Footprint Layer Definitions

The below table outlines the use of various layers when creating footprints.

|  |  |
| --- | --- |
| Layer Name | Definition |
| User.Drawings | Shall Contain Drill map, stackup, board characteristics, and any comments. |
| User.Comments |  |
| User.Eco1 |  |
| User.Eco2 |  |
| F.Fab | Entirety of Front Assembly Drawing. Footprint name shall not be visible. Reference designators shall be visible as needed. |
| B.Fab | Entirety of Back Assembly Drawing. Footprint name shall not be visible. Reference designators shall be visible as needed. |
| User.1 |  |
| User.2 | Guide for multiple connector submodules. Place exact location of each connector needed. |
| User.3 | Miscellaneous notes |
| User.4 |  |
| User.5 |  |
| User.6 | For PCB mounted components that require a panel hole. The outline of the panel hole shall be drawn and a “+” shall be placed in the center of the hole. This should match the User.9 layer of the associated panel component. |
| User.7 |  |
| User.8 | Used for Panel Component Only. Shall indicate the outline of any knobs or caps |
| User.9 | Used for Panel Components only. All panel holes shall be outlined on this layer. The center point of the hole shall be marked with a “+” indicator |

Drill map is generated by exporting the drill map to a DXF. Choose option for PTH and NPTH in same file. Then import the DXF as graphics with the Fix Discontinuities box unchecked.

Stackup drawing is generated by selecting Place>Add Stackup Table.

Board Characteristics are generated by selecting Place>Add Board Characteristics

## Component Fields

All symbols shall have the following fields:

|  |  |
| --- | --- |
| **Title** | **Description** |
| Reference | Reference Designator |
| Value | Part number for ICs or similar and value of passives or similar |
| Footprint | Must be from custom synthesis library |
| Datasheet | Optional |
| Description | Part Description as seen on Digikey |
| MFR | Manufacturer Name |
| MPN | Manufacturer Part Number |
| LCSC | LCSC Part Number if available |

# Production Files

## Revisions

Revisions shall have two digits. The first is a letter representing the revision of copper starting with A. The second digit shall be a number and represent any changes to the population of components starting with 1.

## Titleblocks

Company name shall be DIALTONE SYNTHESIS and display revision with part number.

## Output Files