eightolives.com

Using the Raspberry Pi B Hat Templates in Schematic Mobile and Layout

William_Kaupinis@eightolives.com
Mar 4, 2016

About the Templates

- A Raspberry Pi Hat is an add-on circuit board that plugs onto a Raspberry Pi B+ CPU board.

- It is used to add custom peripheral circuits

- Eightolives provides two generic templates

  - An "empty" schematic that contains only the single 40 pin connector with signal assignments for use in Schematic Mobile

  - A circuit board template with mounting holes and connector for use in Layout

- You can use both templates to start your design.

Template for Schematic Mobile
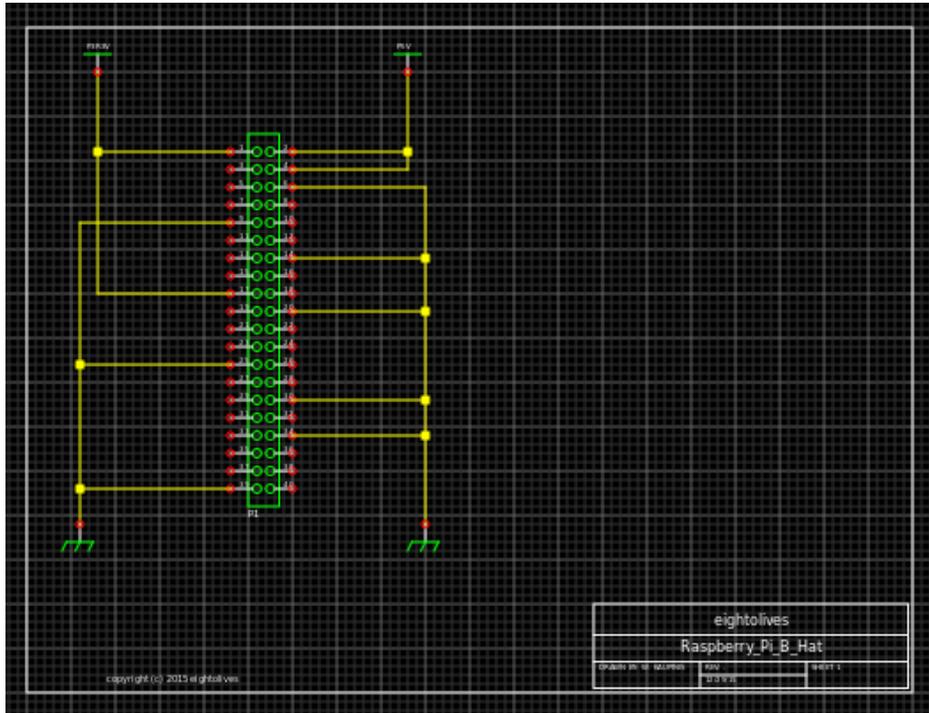
- In Schematic Mobile, select **Design > Open Project**.

  - Select **Server** then, from the drop-down menu, select "Raspberry Pi B Hat Board"

- Select **Design > Open Project File**

  - From the drop-down menu, select "**Schematic: Raspberry Pi Hat**"

- You can then edit and save your design in your own project storage area.

See http://www.eightolives.com/docs/papers/UsingSchematicMobile.pdf for details on how to use Schematic Mobile.

Template for Layout
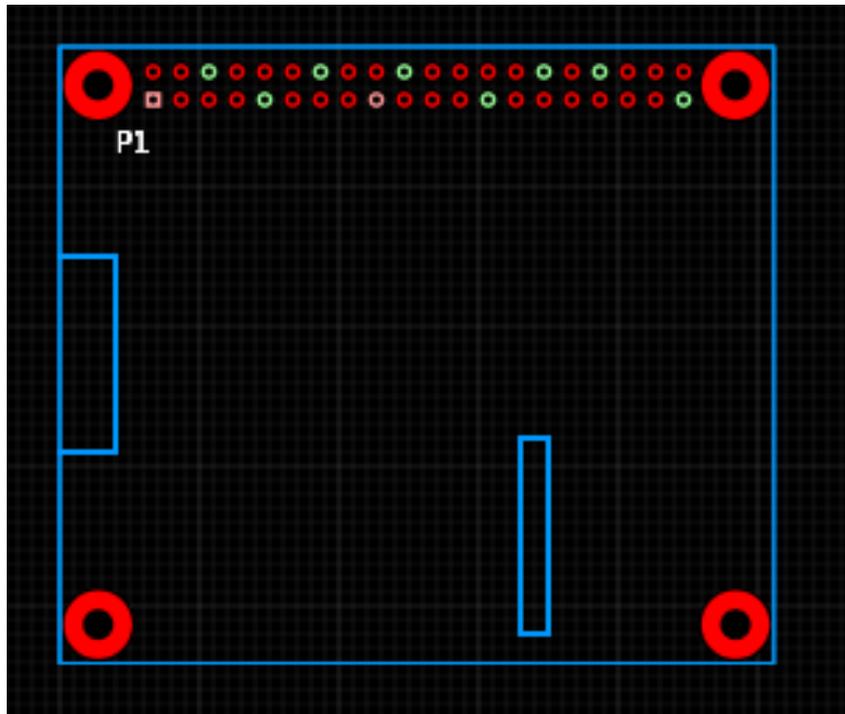
- For the corresponding layout:

- In Layout, select **Design > Open Project**.

  - Select **Server** then, from the drop-down menu, select "**Raspberry Pi B Hat Board**"

- Select **Design > Open Project File**

  - From the drop-down menu, select "**Layout: Raspberry Pi B Hat.pcb**"

- You can then edit and save your design in your own project storage area.

See http://www.eightolives.com/docs/papers/UsingLayout.pdf for details on how to use the Layout tool.

Raspberry Pi Hat Dimensions
Converted to inches

eightolives.com

Origin 0, 0

2.559
65

58 2.283

R = 3mm 0.118

29
1.142

0.039
R=1mm

DISPLAY FLEX CUTOUT

5
0.197

17 0.669

0.079

1.929

49 56.5

2.205

56.5mm FOR SMT STYLE GPIO HEADER
OTHERWISE 56.0mm FOR THROUGH HOLE HEADER

2
CAMERA FLEX SLOT

0.768
19.5

3.5
0.138

11.5

17 0.669

0.453

3.5

45
1.772

6.2 0.244

2.75 0.108

0.138

4x M2.5 MOUNTING HOLES
DRILLED TO 2.75mm +/- 0.05mm

Copyright © 201

5
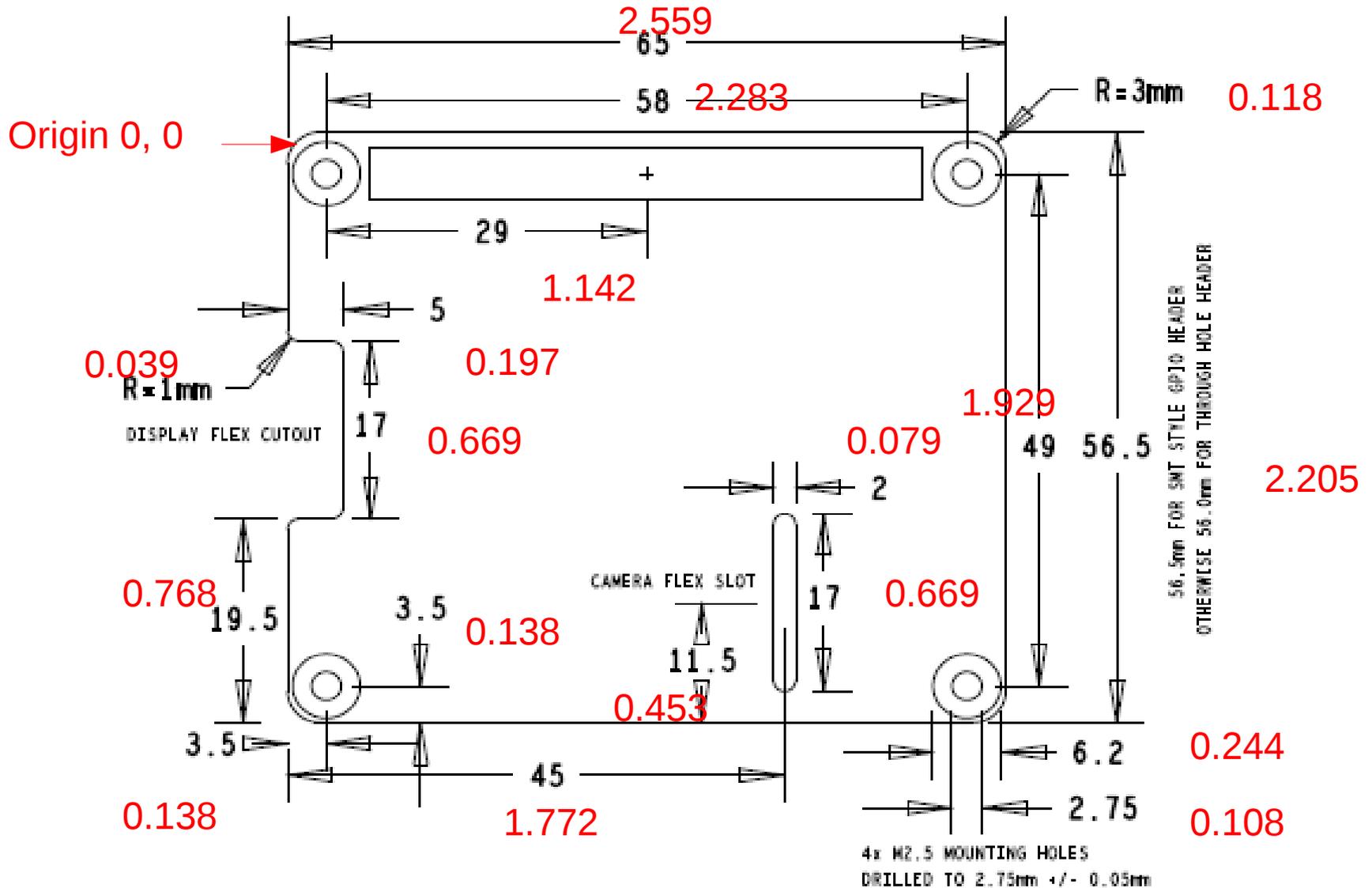
Raspberry Pi Hat Notes

MINIMUM HAT REQUIREMENTS:

- BOARD MUST BE 65x56mm or 65x56.5mm AS PER THIS DRAWING
- BOARD MUST HAVE 3mm RADIUS CORNERS AS PER THIS DRAWING
- BOARD MUST HAVE 4 MOUNTING HOLES IN CORNERS AS PER THIS DRAWING
- BOARD MOUNTING HOLES MUST FOLLOW MOUNTING HOLES SPECIFICATION
- BOARD MUST HAVE FULL 40W GPIO CONNECTOR

MOUNTING HOLES SPECIFICATION:

MOUTING HOLES SHOULD IDEALLY BE NON-PLATED. IF PLATED, HOLE
AND LAND MUST BE ELECTRICALLY ISOLATED (DO NOT CONNECT THESE TO GND)

MOUNTING HOLE LAND SHOULD BE MIN. 6.2mm AND EITHER ISOLATED
COPPER OR BARE BOARD (OPEN SOLDER MASK)

MOUNTING HOLES SHOULD BE DRILLED TO 2.75mm +/- 0.05mm

FURTHER NOTES:

IT IS RECOMMENDED TO PROVIDE SLOTS OR CUTOUTS FOR CAMERA AND DISPLAY
FLEXIS SO CAMERA AND DISPLAY CAN STILL BE USED WITH HAT ATTACHED

CAMERA FLEX SLOT AND DISPLAY FLEX CUTOUT ARE EXAMPLES OF HOW TO DO THIS

Layout Notes:

- Layout does not currently support the specification of rounded corners and milling specifications

- Layout tool is still in development. Check open issues before use.

Starting Layout from Schematic Mobile
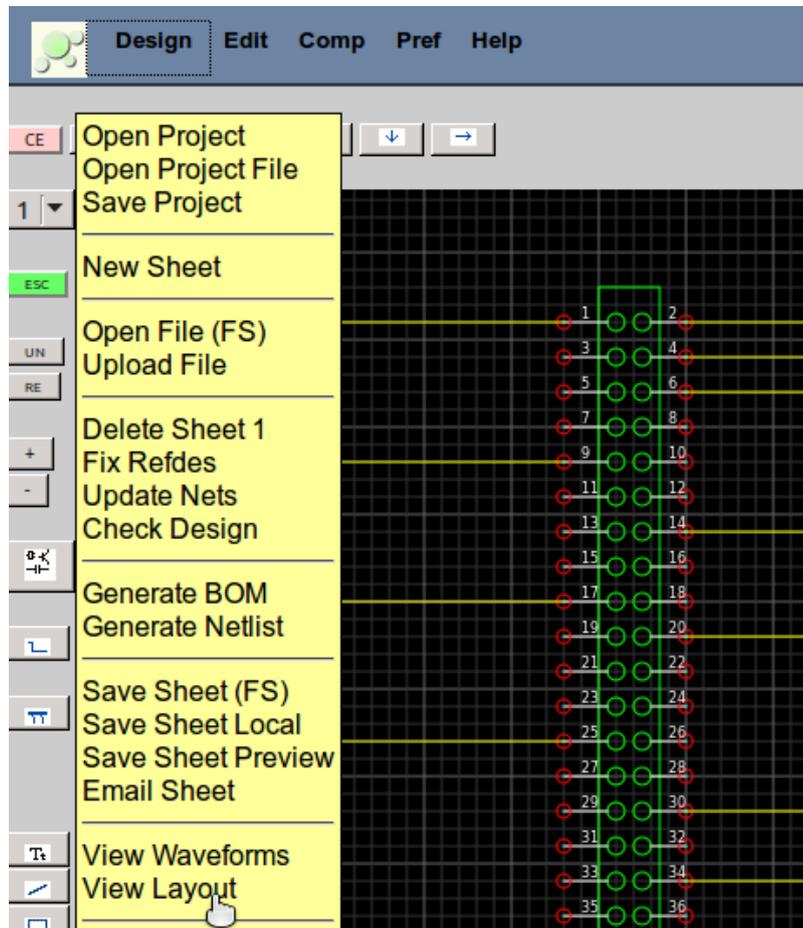
- When you have created your project area and modified your schematic, you can quickly invoke Layout by selecting **Design > View Layout**

Design Data is linked using Hardware Models (.js files)

- Schematic Mobile creates a Hardware Model of the schematic which contains component, pin, footprint and net connect information. (DesignName.js)

- Schematic Mobile can also create default, template models for all components used

- Layout uses the Hardware Models for footprint, netlist and routing instructions. The default Layout file should be named DesignName.pcb

In Schematic Mobile

- All components should be assigned the Attribute **footprint** with the value being the footprint file name

  - e.g. Header40.fp

  - Use the Edit Attributes options or from the Component popup menu select **Set Footprint**

- For items like bypass capacitors, assign the **placenear** attribute e.g. **placenear** U1-14 (used by Autoplace)

- Top level attributes **gnd_plane_signal** and **power_plane_signal** should specify the signal names of the power and ground planes.

In Layout

- DesignName.pcb should be your baseline file. Keep backups of the board template version, the version with good placement data, a version with critical routes and other work-in-process versions.

- Designs saved as .pcb files can be read by the gEDA PCB tool.

For more information

- Visit