PCB Design: Creating and Using Pattern Libraries and Schematic Layouts within DipTrace

Cecilia Acosta

November 11, 2011

Design Team 1

Abstract

This paper will focus on the use of pattern libraries and schematics layout using DipTrace software. These are two key steps needed in the design of the PCB that will hold the PSOC First Touch Kit and the Arduino Ethernet Shield. At the end of the design the user will be ready to layout the PCB for fabrication.

Keywords

DipTrace, Pattern Libraries, Schematic Layout, PCB Design, PCB

Introduction

DipTrace is a software application for creating schematics and printed circuit boards (PCB) [1]. This document will walk the user through creating and using pattern libraries and schematics layouts in DipTrace. These are two of the four steps needed to create a PCB design using this software. The four steps needed are Component design, Pattern Library design, Schematics Layout and the PCB layout. Throughout this paper there is the assumption that the components layout have already been created and that the user is familiar with the procedure. There are some references and appendix information at the end of this document that can help if further assistance is needed with the techniques not focused on in this paper.

Objective

The objective of this manuscript is to create the libraries and schematics for the PCB that will hold and interface the PSoC First Touch Kit and Arduino Shield. This PCB will allow easy access to both components, in addition, the entire final product will be easy to package. This final product includes the PSoC, the Arduino Shield and the PCB. The communication between the PSoC and Arduino Shield will transpire without further hardware interaction. After the two components are added to the PCB the only changes needed for flexibility in application will be based on software design instead of hardware.

Steps

Pattern Editor :

Open DipTrace and choose the Pattern Editor option as shown in Figure 1to open the pattern editor session of DipTrace.

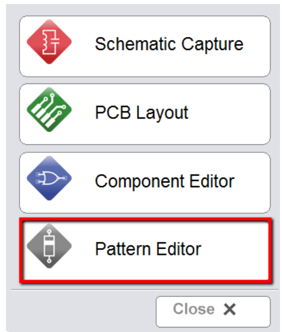


Figure 1: Open Patter Editor

The first step to the design is to create one of the hardware pattern headers. It is recommended that several patterns are used and create each header of the components individually. After having all the headers added, combine them to have the finalized pattern. This will reduce the steps in case the redesign of a component within the pattern is needed. In order to add each header to the pattern click on Pattern, and choose Get from Library as shown in Figure 2 below.

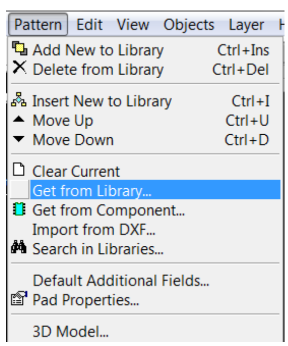
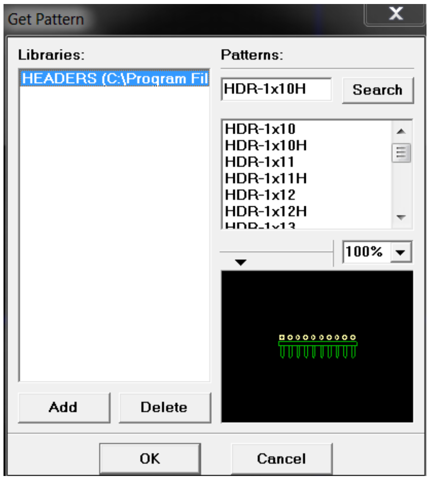


Figure 2: Pattern from Library

The windows shown in Figure 3 will pop up, here is where a library can be chosen, and in this case the headers library within DipTrace is needed. If the library needed does not show up under libraries, click on Add and search for it under Program Files in the DipTrace folder. After the library is added the desired pattern is chosen. In this case the Arduino Shield and the PSoC are composed of several 1x6, 1x8 and 2x3 headers. This process is repeated until all the needed headers have been added. To create the pattern separate from each other right click on the previous pattern on the left and choose Insert New Pattern as shown in Figure 4.



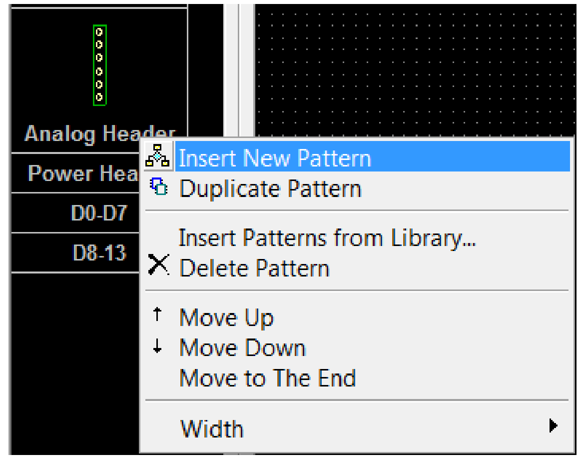


Figure 3: Headers from Library

Figure 4: Pattern Creation

Using this method all the headers for each component are created. For the Arduino Shield the following headers need to be created individually: ICSP (3x2), Analog Header (1x6), Power Header (1x6), D0-D7 (1x8) and D8-D13 (1x8). To name each header component use the Pattern Property window displayed in the upper right hand side corner of the screen. There could be cases where the pattern of the design is not already a part of DipTrace library. In these cases new patterns will need to be drawn out and saved as libraries components for future use. All pattern library are saved with an extension of *.lib.

Finally, to have the entire Arduino Shield Pattern, all the headers previously created are added to a last pattern with each header in a different layer. The list of layers are displayed in the right hand side pane of the screen, by simply clicking on the next layer, adding pattern from library and choosing your previously created pattern you can have the design shown in Figure 5 below. For those components that need to be rotated simply right click and rotate.

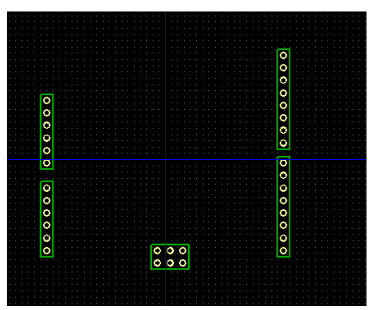
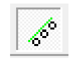


Figure 5: Arduino Shield

For the design shown above a grid of 5mils was used. Between the two headers on the left there are 200 mils, between the left lower header and the middle one there are 890 mils and between the middle header and the right one there are 810 mills. The outer headers are 1900 mils from each other measuring from the inside lines. The two headers on the right are separated by 160 mils. This is very useful information in this section as well as PCB layout section not included in this paper.

The same procedure as described above is used to create the PSoC header components, except that this time there was not an available 1x14 header in the library. In order to create a 1x14 header first a padline was placed in the first layer. This is done by clicking on this figure  on the tool menu, followed by clicking and dragging the mouse to draw as many pads as are needed, in this case 14. On the second layer, a rectangle was drawn around the padline to simulate the headers. Lastly this was added to the library as the Arduino Shield pattern. The final pattern is shown in Figure 6 below. The right and left headers are separated by half of an inch from each other measuring from the inside lines.

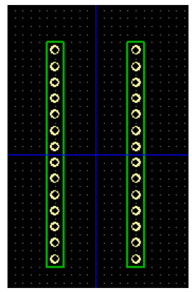


Figure 6: PSoC

Schematic Capture:

To start the schematic's design, choose Schematic Capture when opening DipTrace as shown in Figure 7.

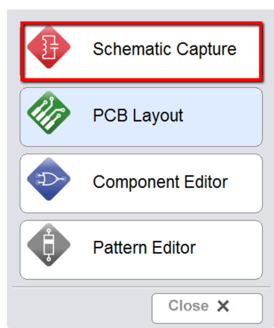


Figure 7: Schematic Capture

To begin the schematic access to already designed components is needed. These files have an extension *.eli. To add components click on Objects on the Menu bar at the top of the screen and select Place Component as shown in Figure 8. A pop up window will appear as shown in Figure 9. In this window if your library is selected as shown the desired component will be chosen. In this case the component is the Arduino Shield, select it and click Place. The component will now appear on the DipTrace Window and by moving the mouse it can be placed anywhere in the window.

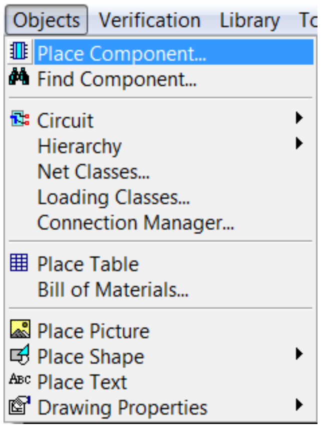


Figure 8: Place Component

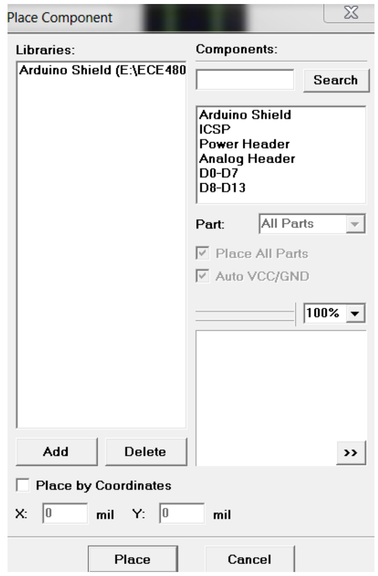


Figure 9: Place Component pop up

In order to add the PSoC First Touch Kit we follow the same procedure as above with a minor modification. The difference is that the component library needs to be added to the list and then PSoC can be chosen from the list of components within the library. Again, click place and the component will be placed wherever desired in the window, in this particular case the components are 2440 mils from each other. After the steps above are followed the schematics should look as shown in Figure 10 below.

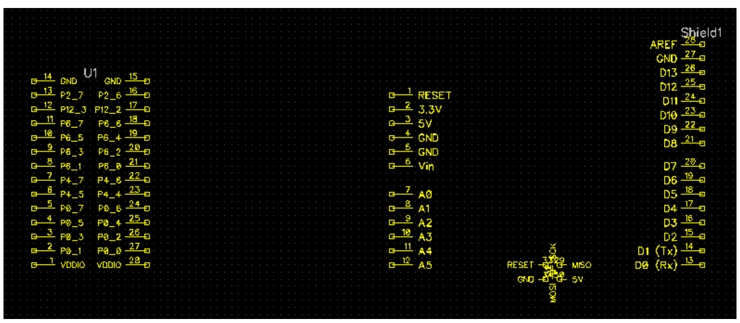
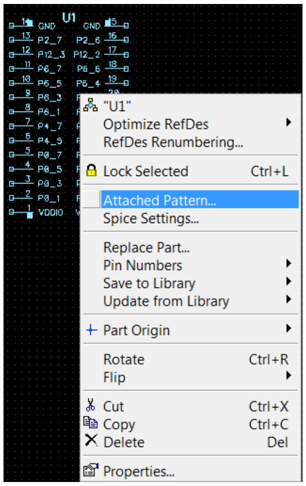


Figure 10: Components placed in schematic

The last step of the schematics is to attach the pattern to the components. In order to attach the pattern on each component, right click on the component and choose Attached Pattern as shown in Figure 11. The window shown in Figure 12 will pop up, on this window the right pattern library needs to be chosen. If the needed library is not shown in the list, click add and select the correct library. Remember pattern libraries have a file extension of *.lib. After the correct library is selected click ok. Proceed to repeat the same steps to attach the pattern to the Arduino Shield.



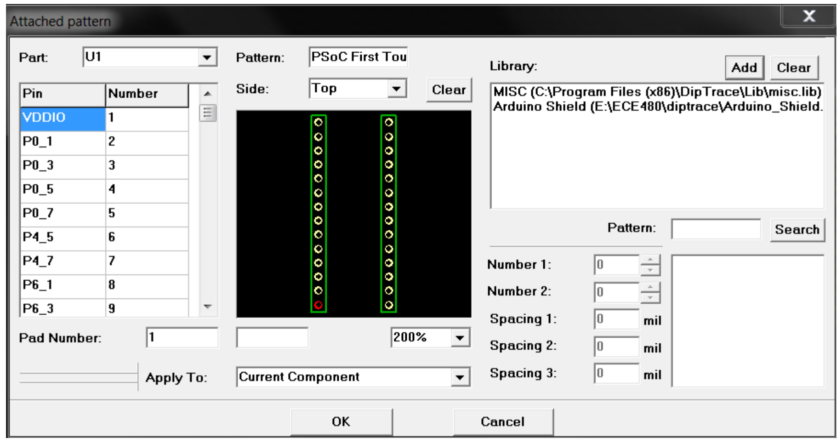
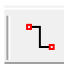


Figure 11: Attach Pattern

Figure 12: Attach Pattern pop up

The final step of the schematic capture is to wire the pins. The wiring configuration is shown in the appendix on this document. Following this configuration choose a pin, for example pin P0_0 of the PSoC connects to Pin D13. Click on  on the toolbar at the top of the screen. Next select pin P0_0 and move the mouse to pin D13, click on it and the two pins are now connected. Follow these steps for all the pin connections. At the end of this task, all pins are attached as shown in the schematics layout in Figure 13 below. To confirm the right connections are being made, hover over a line connector or a particular pin and either the entire line will change color or the pin that it is attached to will be highlighted. This is the best way to confirm all the connections.

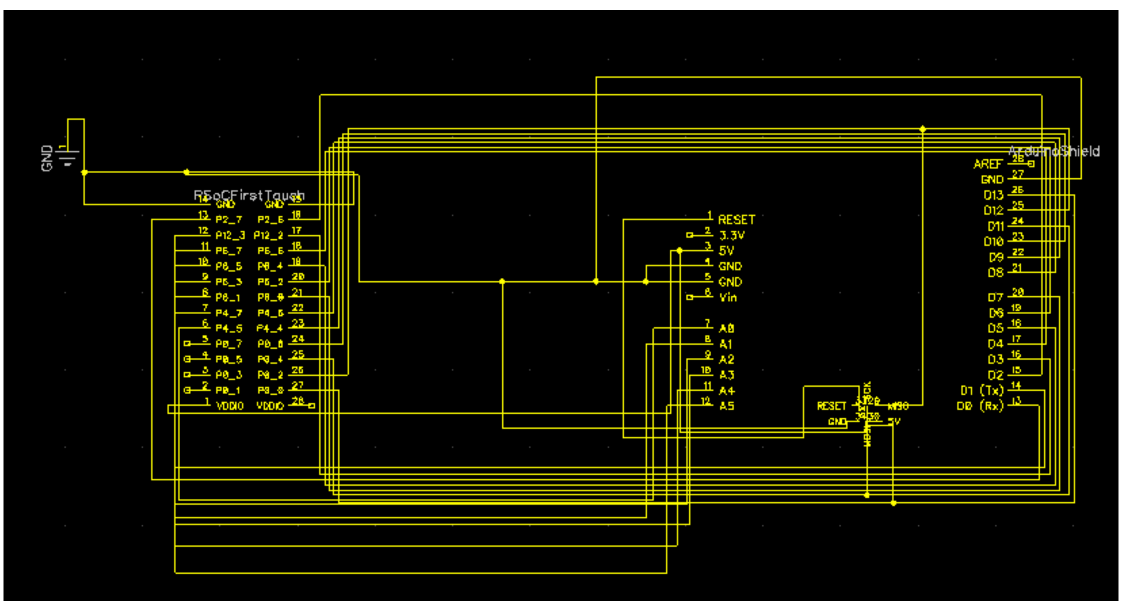


Figure 13: Schematic's Layout

This concludes the schematic capture. The next step to the PCB design is to have the PCB layout and the pins routed according to the schematic capture. In the Appendix section information can be found on how to route the pins of the PSoC and Arduino Ehternet shield as well as more help using DipTrace in general.

Conclusions and Recommendations

After having the components, pattern libraries and the schematic layout done the user is ready to layout the PCB for fabrication. These patterns and layouts are easily modifiable if needed due to the fact that the parts within the pattern libraries were created separately. This is an optimal design method to practice in order to minimize work and increase flexibility of the design.

References

1. DipTrace Professional PCB Design Tool, www.diptrace.com

Appendix

PIN Routing

Corresponds to →

PSoC	**Arduino**
P0_0	D13 (Led, SCK)
P0_2	D12 (MISO)
P0_4	D11 (MOSI)
P0_6	D10 (PWM, SS)
P4_4	D9 (MWN)
P4_6	D8
P6_0	D7
P6_2	D6 (PWM)
P6_4	D5 (PWN)
P6_6	D4
P12_2	D3 (PWR)
P2_6	D2 (External Interrupt)
P2_7	D0 (Serial RX)
P12_3	D1 (Serial TX)
P6_7	A5 (SCL)
P6_5	A4 (SDA)
P6_3	A3
P6_1	A2
P4_7	A1
P4_5	A0

In the case of this PCB design the connections are done using the ICSP as shown below.

PSoC	**Arduino ICSP**
P0_2	Pin 29 (MISO)
VDDIO	Pin 30 (5V)
P0_0	Pin 31 (SCK)
P0_4	Pin32 (MOSI)
	Pin33 (Reset) Routed to Reset Pin on Arduino
GND	Pin34

DipTrace Help

For further information on how to use DipTrace, Component creation and PCB layout follow the tutorials within DipTrace by clicking on the Help Menu options as shown in Figure X below.

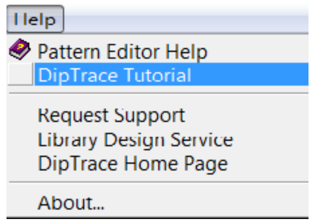


Figure 12: DipTrace Help