

Printed Circuit Board

Design:

Using Dip Trace

ECE 480: Application Note
Design Team 6

Peter Deacon
Fall 2011



Table of Contents

Abstract.....	2
Introduction.....	2
Keywords.....	2
Procedure: Schematic Design	3
Procedure: PCB Auto-Layout.....	8
Exporting Files.....	12
Conclusion	12
References.....	13

Abstract

This application note serves as a step by step guide for creating a printed circuit board using Dip Trace 2.2 Professional PCB design tool. This document will use examples to demonstrate the basic functions that can be performed using Dip Trace. The functions demonstrated include schematic layout and automatic PCB layout. This guide is not meant to be a complete manual for Dip Trace but rather a guide to help new users evaluate and use this PCB design tool.

Introduction

Dip Trace is a user friendly PCB design tool. There are several editions of Dip Trace that have built in limitations on the number of layers a circuit board can have, as well as the number of pins allowed on the board. The full version of Dip Trace allows for unlimited pins and unlimited layers. Dip Trace offers a 30 day evaluation period on the full version.

There are several reasons that Dip Trace was chosen as the software of choice for this guide. First, Dip Trace has an extensive library of components and pads. Furthermore, the controls used in Dip Trace are simple and intuitive. These characteristics make the software easy to learn. The simplicity of creating custom pads for individual components is another benefit of using Dip Trace. Perhaps the most important feature is auto routing. In the PCB created in this guide the importance of auto routing may not be apparent, but on a complex circuit board auto routing is very important.

Keywords

PCB: Printed Circuit Board.

Dip Trace: PCB creation software.

Schematic Capture: Portion of Dip Trace software used to create schematics.

Library Pane: list of Libraries in which components are organized.

Schematic Pane: pane of the Dip Trace window where the schematic is laid out.

Procedure: Schematic Design

First, open Dip Trace; to do this click Start/All Programs/DipTrace Launcher. The window in Figure 1 will open. In the launcher, select Schematic Capture; a welcome window will appear. If you have not purchased any version of Dip Trace select evaluate.

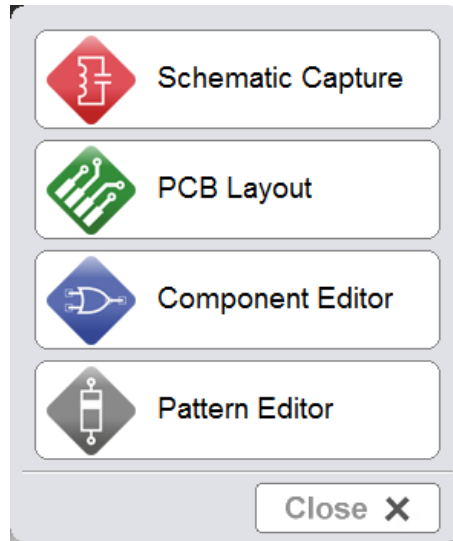


Figure 1

The Dip Trace Schematic Capture program is now open. Select File/Titles and Sheet Setup. The window in Figure 2 will open up. Apply the settings as show in Figure 2. Next click the Title Blocks tab and apply the options in Figure 3 and click close.

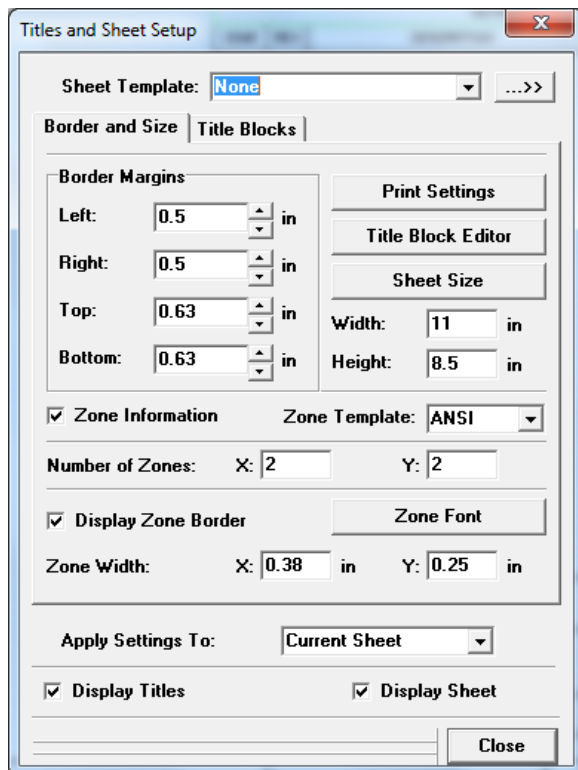


Figure 2

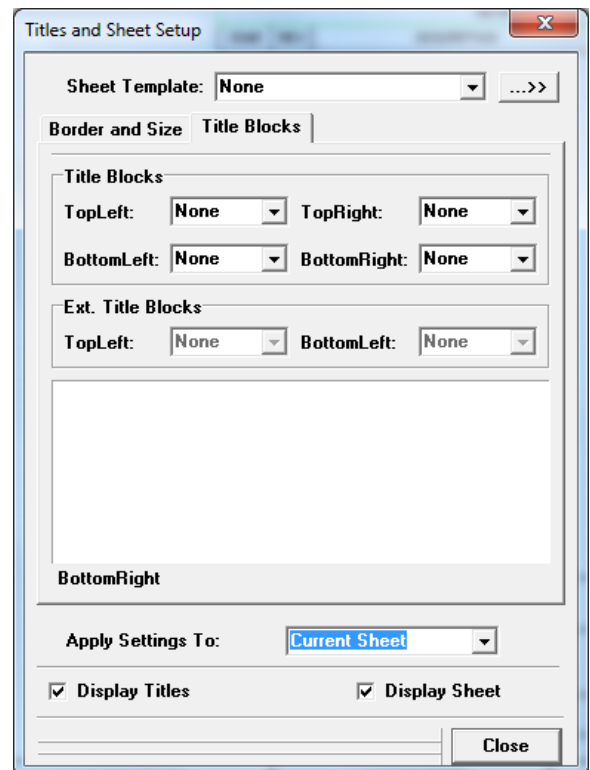


Figure 3

Now we have created an empty schematic. We are going to design a low pass filter with a gain stage. To do this we will use a chip from the directory, some resistors, and capacitors. First, we will add an integrated circuit that is found in a library provided by Dip Trace. The library pane is located just above the schematic; select the Maxim library. To scroll, click the right and left arrows on the right side of the pane. In Figure 4, the Maxim library is boxed in red in the libraries pane. The other red box in Figure 4 is the search bar; type Max414CPD into the search bar. This will bring up the precision op-amp integrated circuit that is used in this example. Throughout the design of this circuit zoom adjustments can be made by scrolling with the mouse wheel. Rolling the mouse wheel forward will zoom in centered around the cursor location. To zoom out, roll the wheel backward. Zoom can also be adjusted using the magnifying glasses above the libraries pane. The centering of the schematic can be adjusted by scrolling on the vertical and horizontal scroll bars around the perimeter of the schematic.

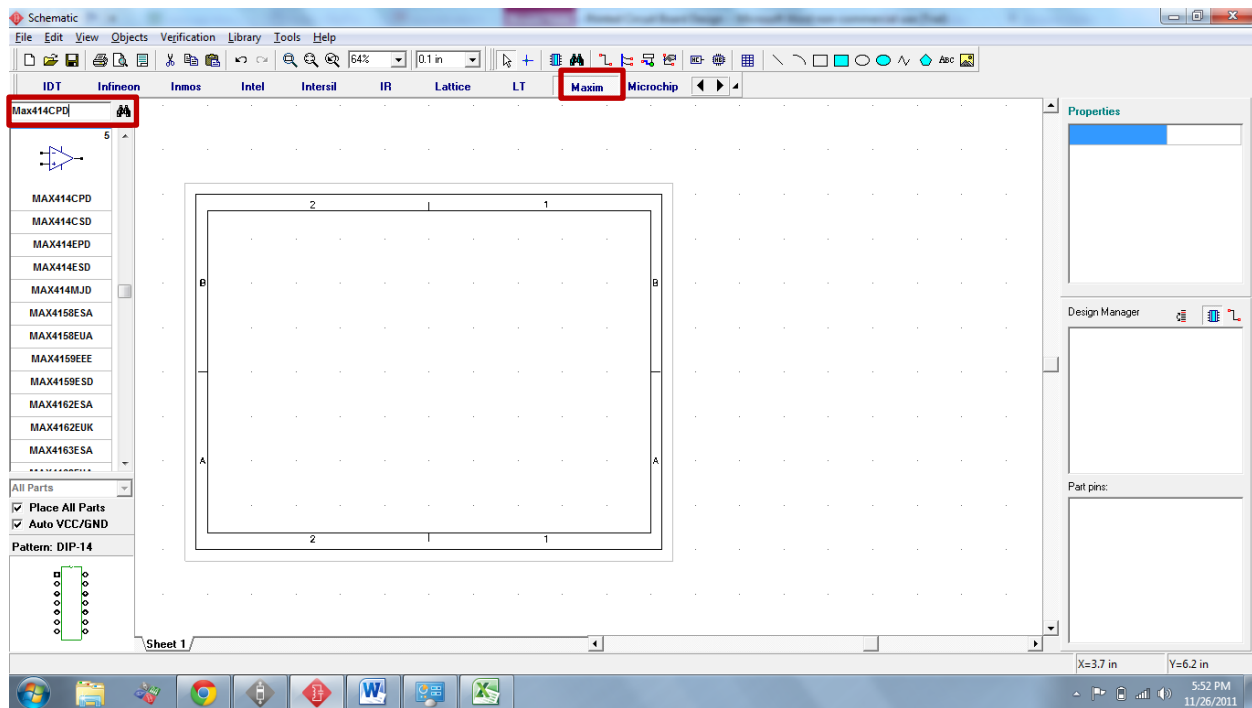


Figure 4

Left click on the MAX414CPD icon just below the search bar and move the mouse over to the schematic. Left clicking on the schematic will drop the integrated circuit onto the board. Clicking more than once will drop more IC's onto the schematic. Once one IC is placed onto the schematic right click to exit the placement mode. Once an IC has been placed, individual components can be moved by clicking and holding down the left mouse button. When the button is released the component will be dropped back onto the schematic. Rearrange the components as show in Figure 5. Delete the op-amp labeled U1.3, as it is not used in the circuit being designed. To do so left click op-amp and press the delete button.

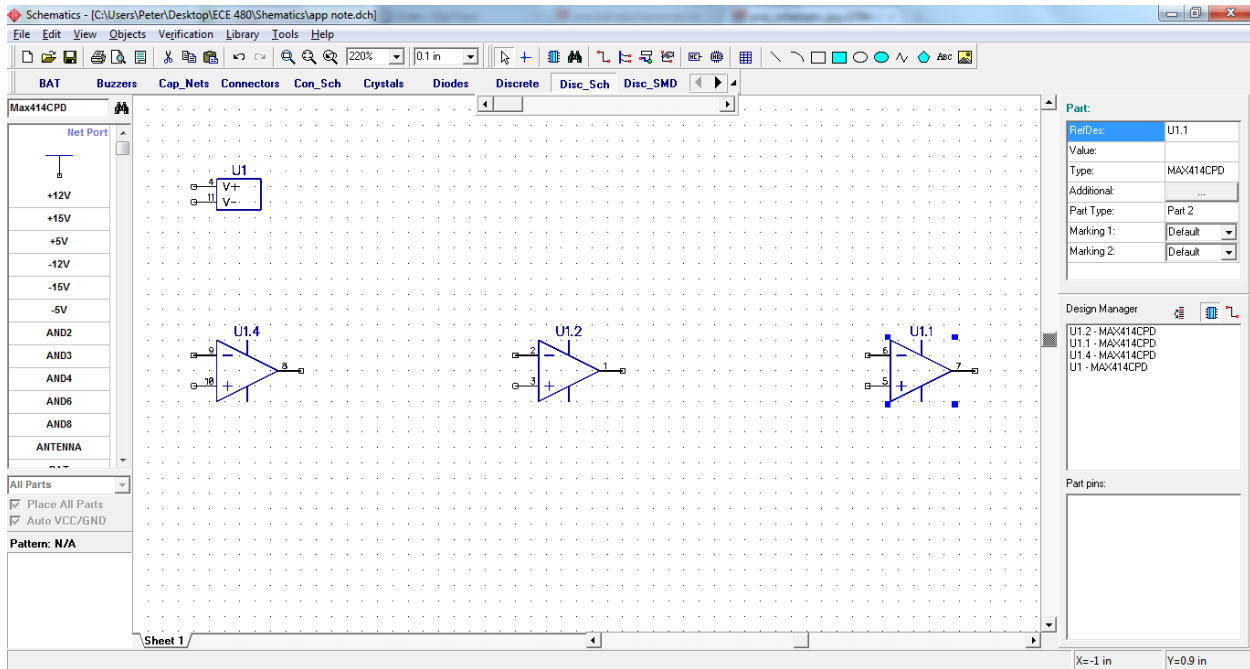


Figure 5

Next we will connect power and ground to the IC. Locate and select the Disc_Sch library. Within the library search for GND and +15V place one of each of these components by left clicking the object and then left clicking on the schematic. After placing one object right click the schematic pane to exit from place mode. Now left click the place wire button to enter connection mode. Click the square located on the +15V component and then click on pin 4 of U1 labeled V+. Next Click on the square located on the GND component and connect it to pin 11 of U1 labeled V-. Now right click to exit connection mode. The connections for power and ground are shown in Figure 6. We will also add four +5V components for later use. Do this by selecting the +5V component and right clicking on the schematic 4 times. Add an additional GND component as well.

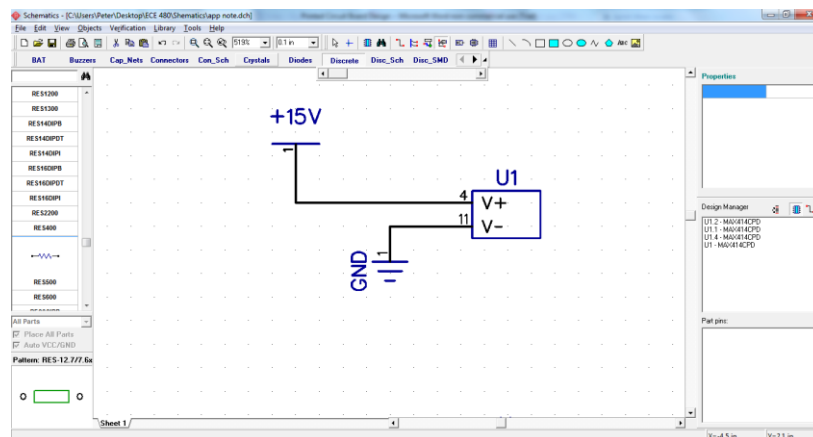


Figure 6

Next we will place several discrete components. Select the discrete library. In that library select RES600. This corresponds to a through-hole resistor with 600 mils spacing between pins. We need to place 7 of these resistors horizontally and 6 vertically. The default placement is horizontal. Place 7 resistors horizontally on the bottom half of the schematic by left clicking 7 times. To rotate the components press the space bar while they are selected. Now click 6 more times to place the 6 vertically oriented resistors. The schematic should now contain resistors labeled R1 through R13. Right click to exit placement mode. Select R1 by left clicking it. In the upper right part of the window a pane will open up called the Part Pane. The Part Pane is outlined by a red box in Figure 7. Enter 8.45K as the “Value” and change “Marking 2” to Value. R1 will now have the label 8.45K below it.

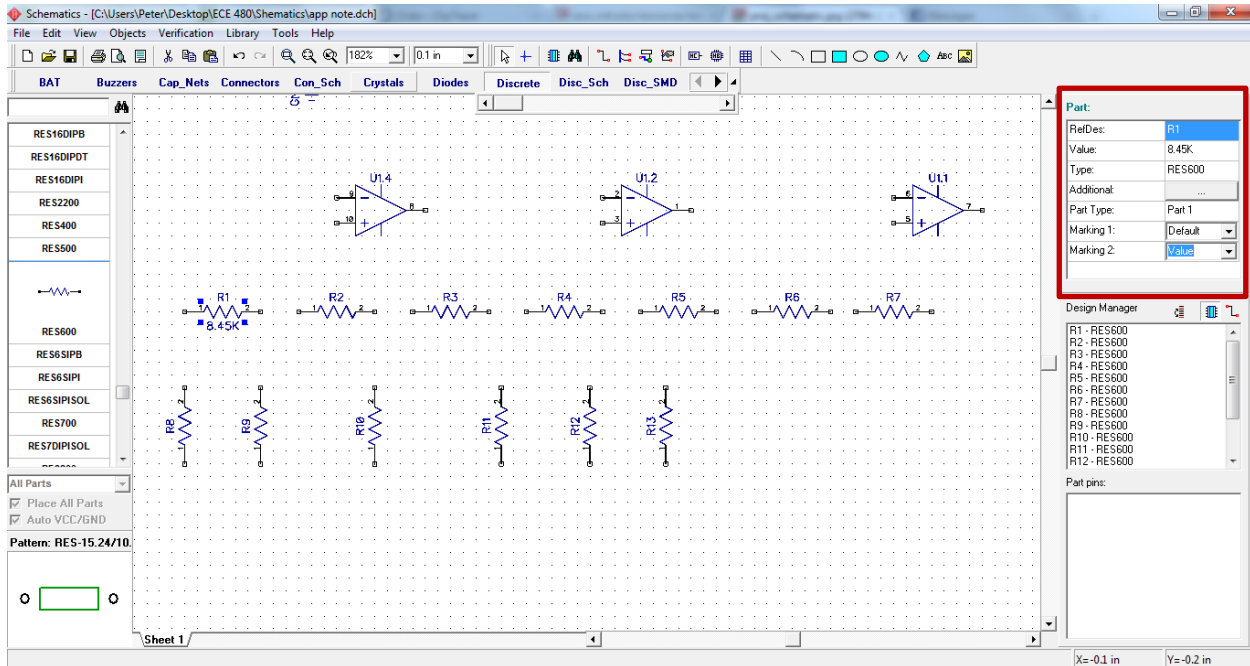


Figure 7

Next select the component “CAP250RP” from the "Discrete" library. Insert one horizontally oriented capacitor by left clicking below the resistors in the schematic pane. Then press space bar 3 times to rotate the capacitor and place three vertically oriented capacitors. Left click to exit placement mode.

Using the same method as used to edit R1, change the labels of all of the resistors and capacitors to match the values listed in Table 1.

Component	Value
R1	8.45K
R2	7.15K
R3	17.4K
R4	4.12K
R5	47K
R6	1K
R7	1K
R8	10K
R9	102K
R10	28K
R11	220
R12	12.1K
R13	1.62K
C1	1uF
C2	1nF
C3	1nF
C4	1nF

Table 1

Components can be moved by left clicking them and holding down the mouse button. When the mouse button is released the component will be dropped onto the schematic. Orient the components in table one as shown in Figure 8. Then select the connection tool and connect them as shown in the same figure. If an error is made, connections can be moved similar to components by clicking and holding then dropping them in the desired location. **Be sure to left click to exit connection mode before attempting to move connections or components.**

Notice that if it is convenient, any component can be flipped by right clicking it and selecting flip/vertical or flip/horizontal. Also fine-tuned movements can be made by selecting a component and moving it with the arrow buttons on the keyboard. Figure 8 shows the completed filter designed for this example.

Note: the input of the filter is intended to go to pin 1 of C1 and the output of the filter is pin 2 of R5.

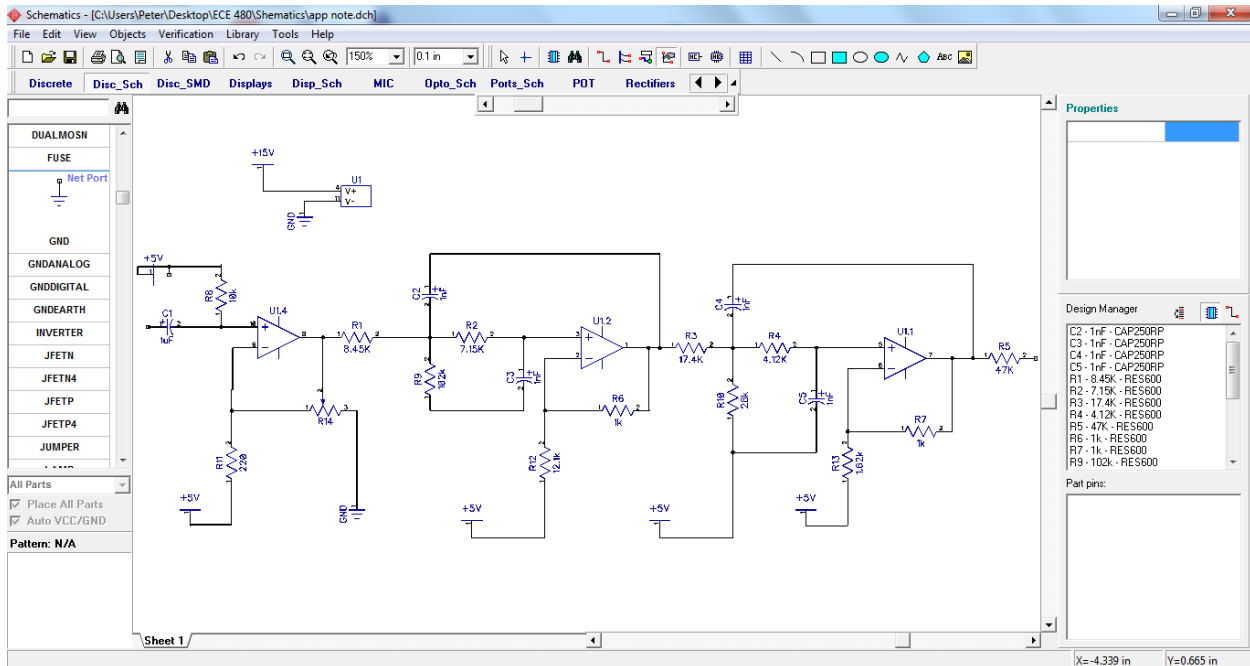


Figure 8

Procedure: PCB Auto-Layout

After a schematic is created Dip Trace allows users to take advantage of automatic placement and routing tools provided in the software package. In the Schematic window, select File/Convert to PCB or simply press Ctrl + B. A “Making PCB from Schematic” dialog box will open up select “Use Schematic Rules” and click OK. This will open up the PCB tool and automatically populate a circuit board with all of the components laid out in the schematic it was created from. The initial PCB will look similar to Figure 9.

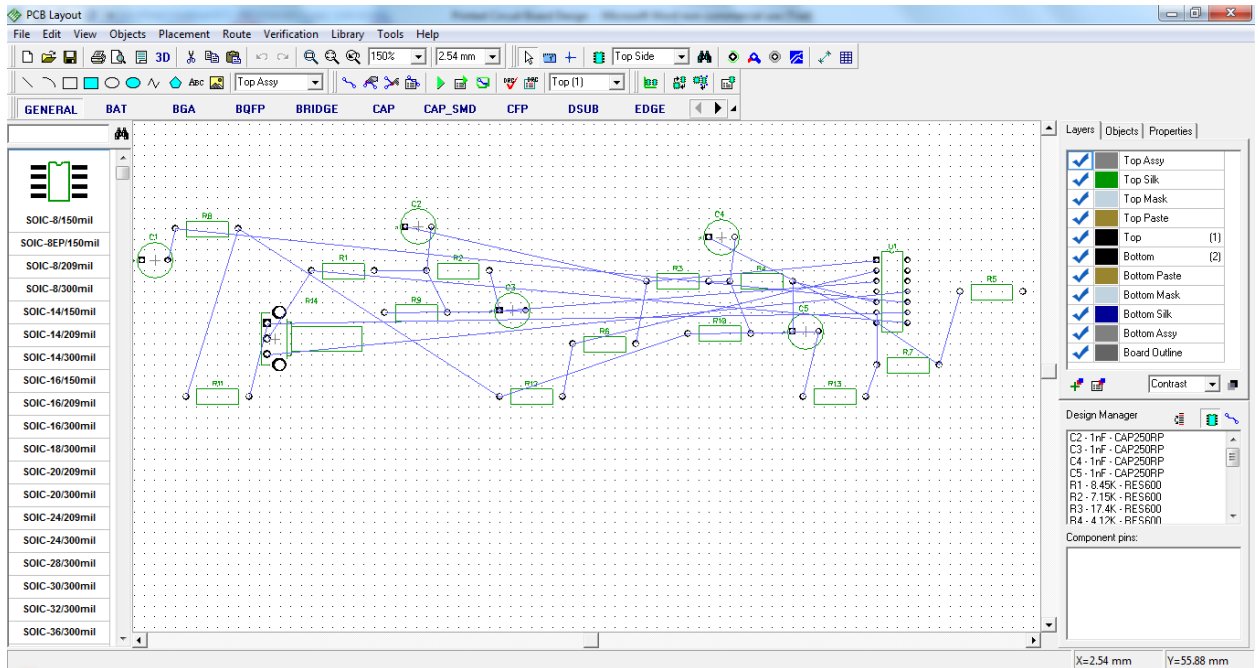


Figure 9

Components can be moved using the click and hold controls similar to how they are moved in the schematic software. However it is convenient, especially with complicated schematics to use the auto-placement tool. Select Placement/Run Auto-placement. If the initial auto-placement is not aesthetically pleasing, moving key components and rerunning the auto-placement tool may be necessary. To undo changes and revert to a previous placement press Ctrl + z. Figure 10 shows a satisfactory auto-placement.

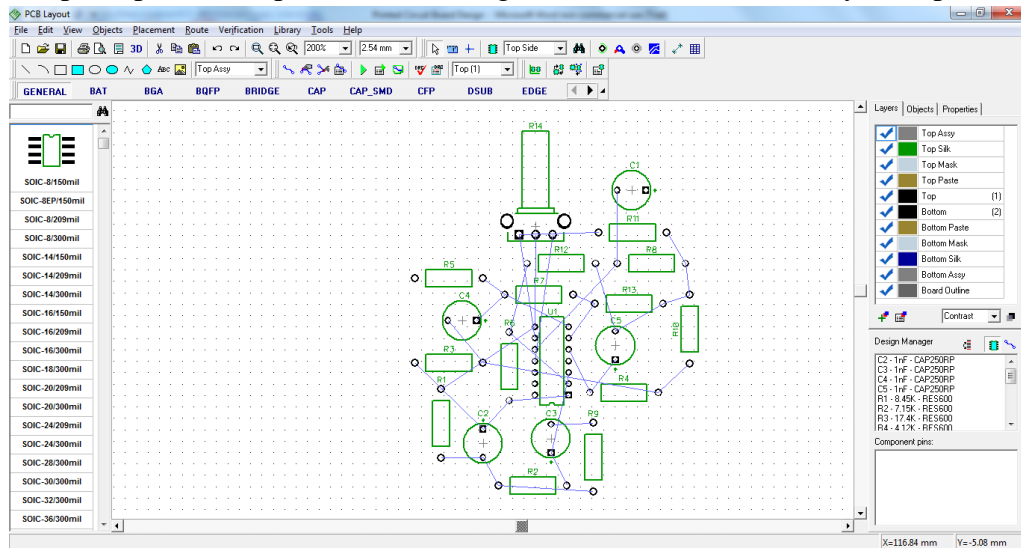


Figure 10

The next step is to create traces to connect the elements in this design. Select Route/Autorouter Setup..., the window in Figure 11 will open up. In the Settings Tab check the “Use Priority Layer Directions” box. Other settings in the Shape Router Setup window can be changed based off of the needs of individual products and the tools available for PCB fabrication. Aside from the Priority Layer Direction setting we will use the default auto-routing feature in this example. Select OK.

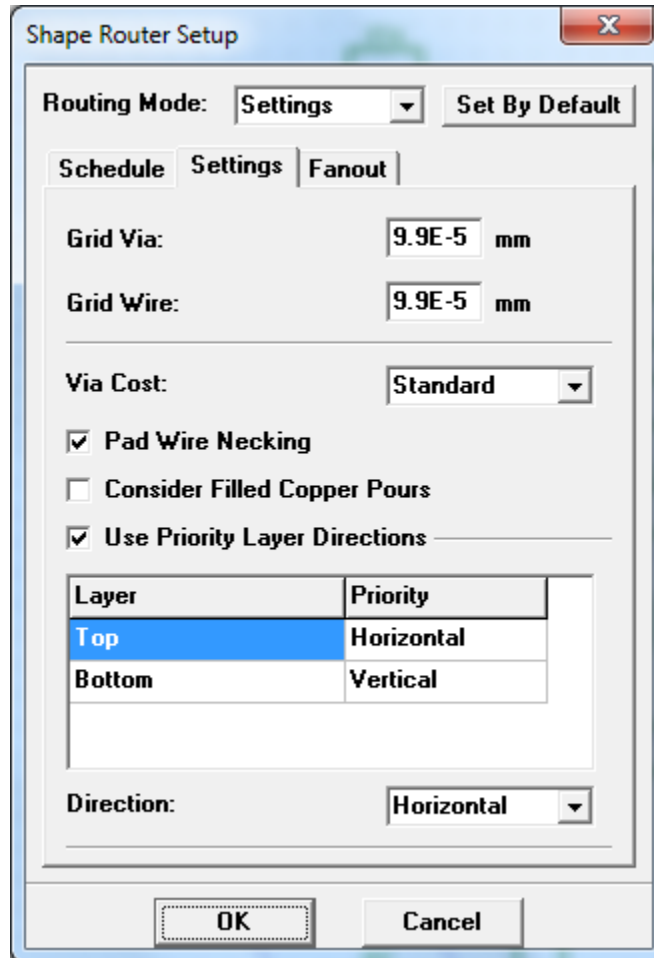


Figure 11

Next select Route/Run Autorouter; this will create traces between the components as laid out on the schematic. Once auto-routing is completed select Verification/Check Design Rules. This verifies that there are no short circuits and unintentional open circuits on the PCB. As long as no errors are found the PCB is ready to be fabricated. If there are errors some traces may have to be moved. Traces can be moved similar to components. When moving traces be sure to avoid creating 90 degree angles. In most industries 45 degree angles are preferred. Figure 12 shows an automatically routed PCB with no routing errors. Once a PCB is laid out with no errors proceed to the exporting files section.

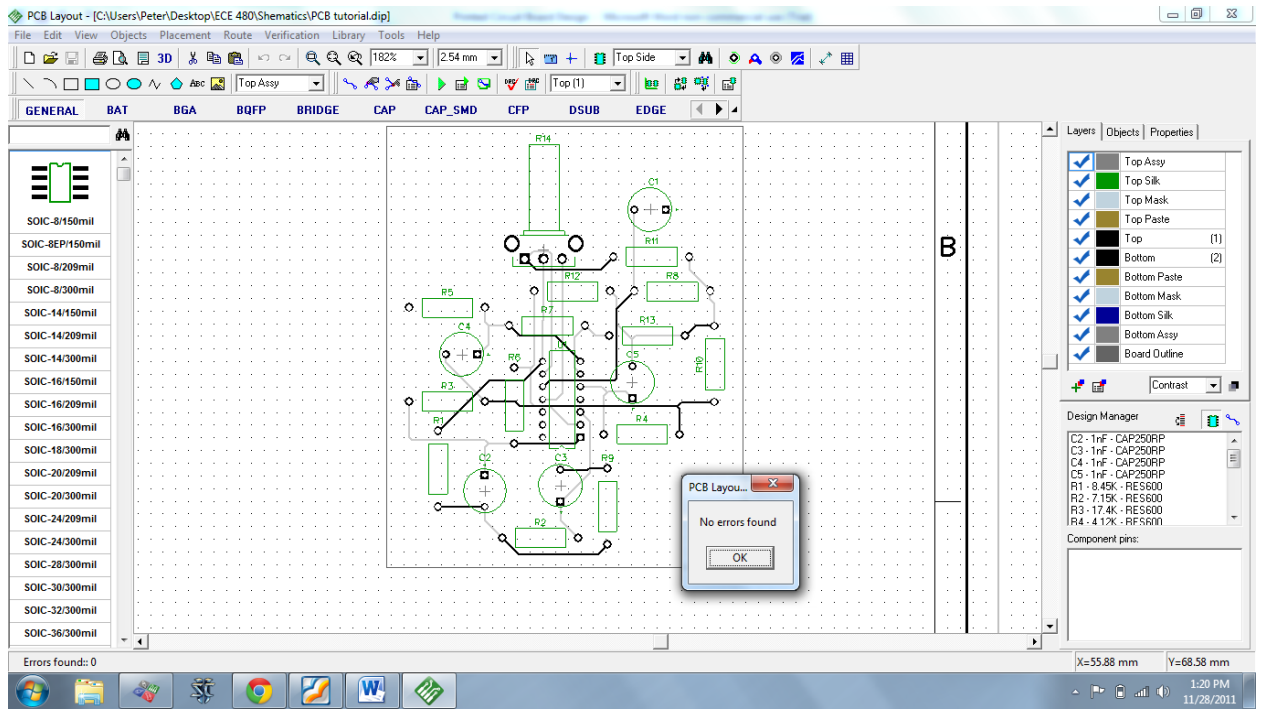


Figure12

Exporting Files

PCB's can be fabricated from several file formats. One of the most universal file formats used in PCB fabrication is Gerber files. To export the files needed to fabricate a PCB select File/Export/Gerber. A dialog box will pop up. Select Export All and save the Gerber files in an appropriate folder. Dip Trace allows PCB files to be exported in several other formats. Be sure to check with the person or company that is fabricating the board and supply them with the correct file format. All formats supported by the Dip Trace software package are found in the File/Export path. Figure 13 shows the file types supported by Dip Trace for exporting.

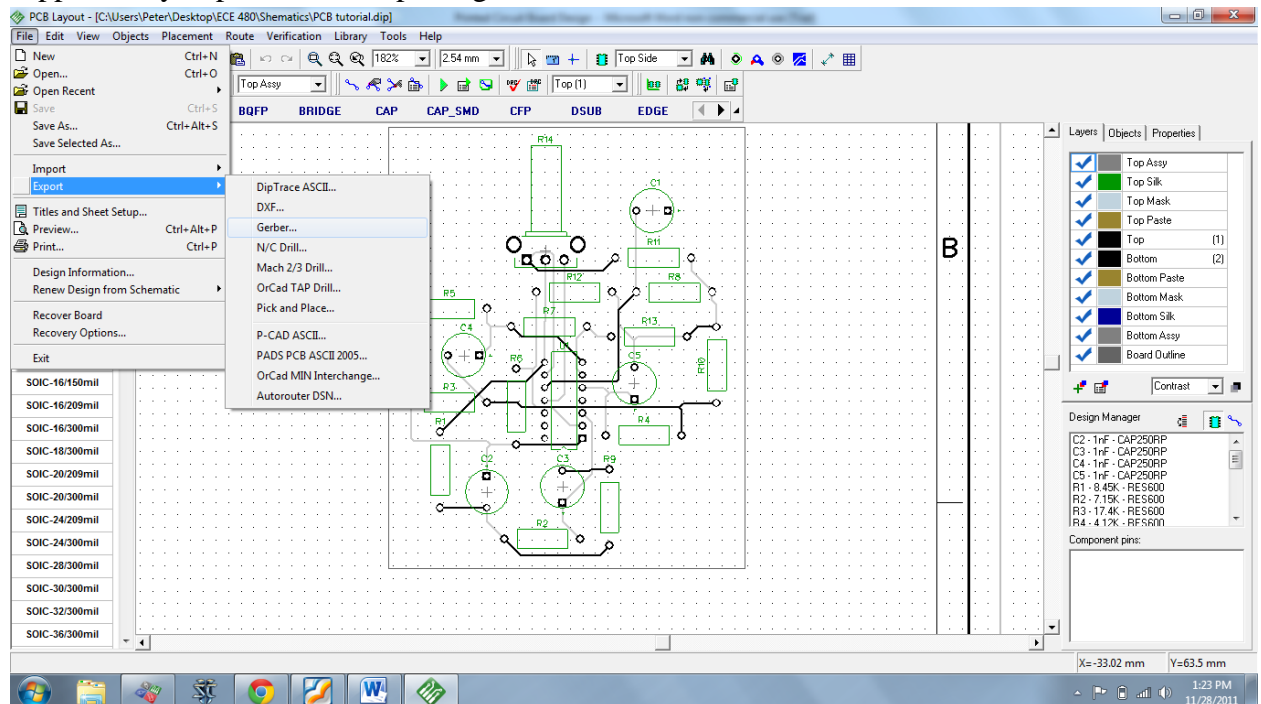


Figure 13

Conclusion

This document is intended to introduce basic features of Dip Trace for users that have little to know experience creating schematics and laying out PCBs. Dip Trace is a user friendly and versatile PCB creation tool. This application note only scratches the surface of the capabilities of Dip Trace software. A major advantage of Dip Trace is that a single software package can allow users to go from schematic design to finished PCB formatting files. PCB design is a meticulous process; therefore auto-placement and auto-routing tools are very useful when putting together complex PCBs.

One important feature Dip Trace offers that was not covered in this application note is the ability to create custom pads and components. This feature is important when the provided libraries do not contain all of the components used in a circuit design. For more

information regarding custom PCB design in Dip Trace complete the tutorial listed in the references section.

References

"Dip Trace Tutorial." *Diptrace.com*. Web. Fall 2011.
<<http://www.diptrace.com/books/tutorial.pdf>>.