Getting Started with PCB Design using Altium Designer

Patrick O'Hara
ECE 480 Design Team 6
Michigan State University
1 April 2011

Abstract

This introduction is intended to go into the basics of Printed Circuit Board design, while using Altium Designer to run through a tutorial. Altium Designer is a powerful development tool used for circuit board create and simulation. It is easy to use and a very powerful tool.

Introduction

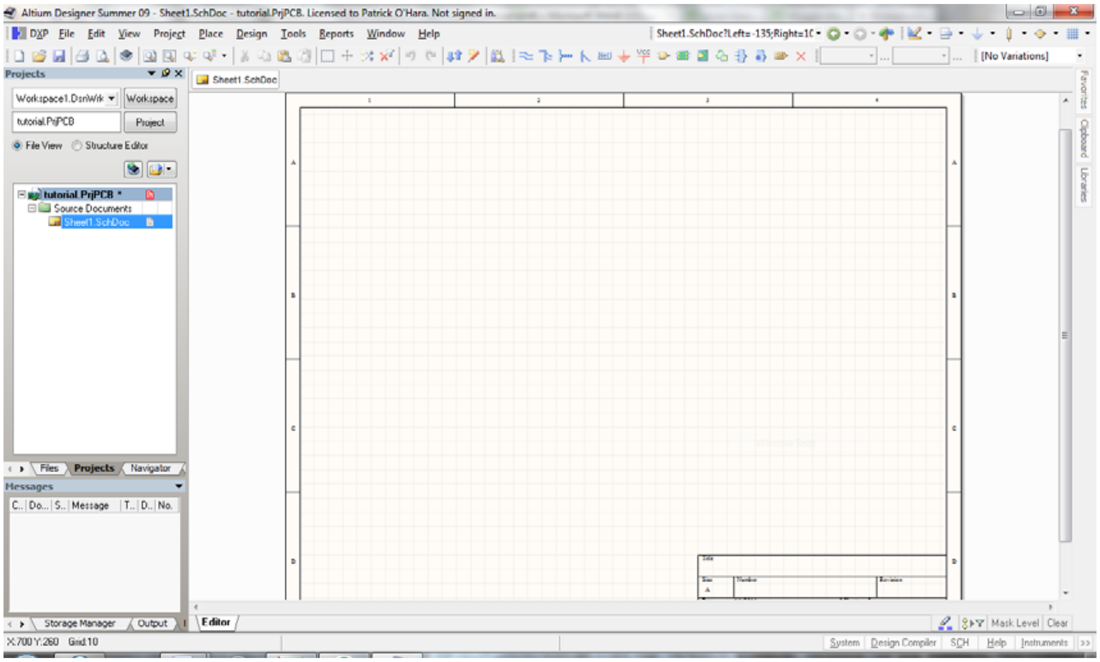
Printed Circuit Boards are used to electrically connect circuit components in a nice neat fashion. PCB's are used in almost every commercially available item that includes electronics. The initial cost of a PCB is high, however as the quantity increases the price per board is greatly reduced. Because the boards are fabricated by machines the human error that would occur in the fabrication and the assembly. The fact that the boards are fabricated by machines also means that the density of elements fit on the board can be much greater. The ability to create and have board fabricated is a useful skill to present a clean final design. Initially the design of a PCB may be difficult to understand, however this document will help to get the ball rolling.

Objective

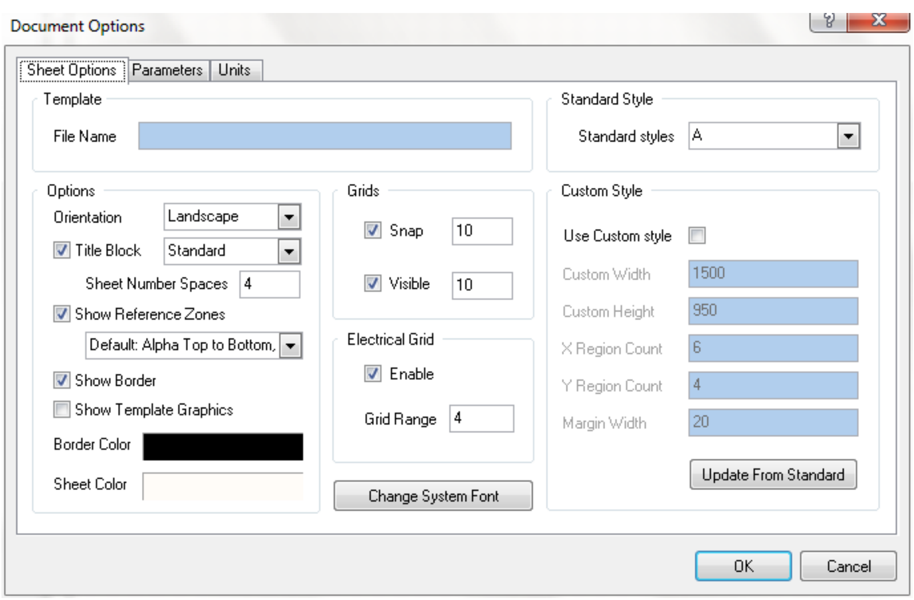
The objective of this document is to get the user started creating a simple printed circuit board. Check the board against design rules to ensure they board is correct. The user of this document should be able to understand the basics of PCB design when it comes to board layers, placing components, routing, and verification. The user should also be able to output their design for fabrication.

Getting Started

To get started we will be using Altium Designer Summer 09. To start a new project go to **File>New>Project>PCB Project**. The Projects panel should now be displayed with the file name PCB_Project1.PrjPCB. To rename the project go to **File>Save Project As**. Currently the project is an empty project. The next thing to add to the project is a schematic file, select **File>New>Schematic**. A blank schematic sheet named Sheet1.SchDoc will display and should be linked to the project in the Project panel.

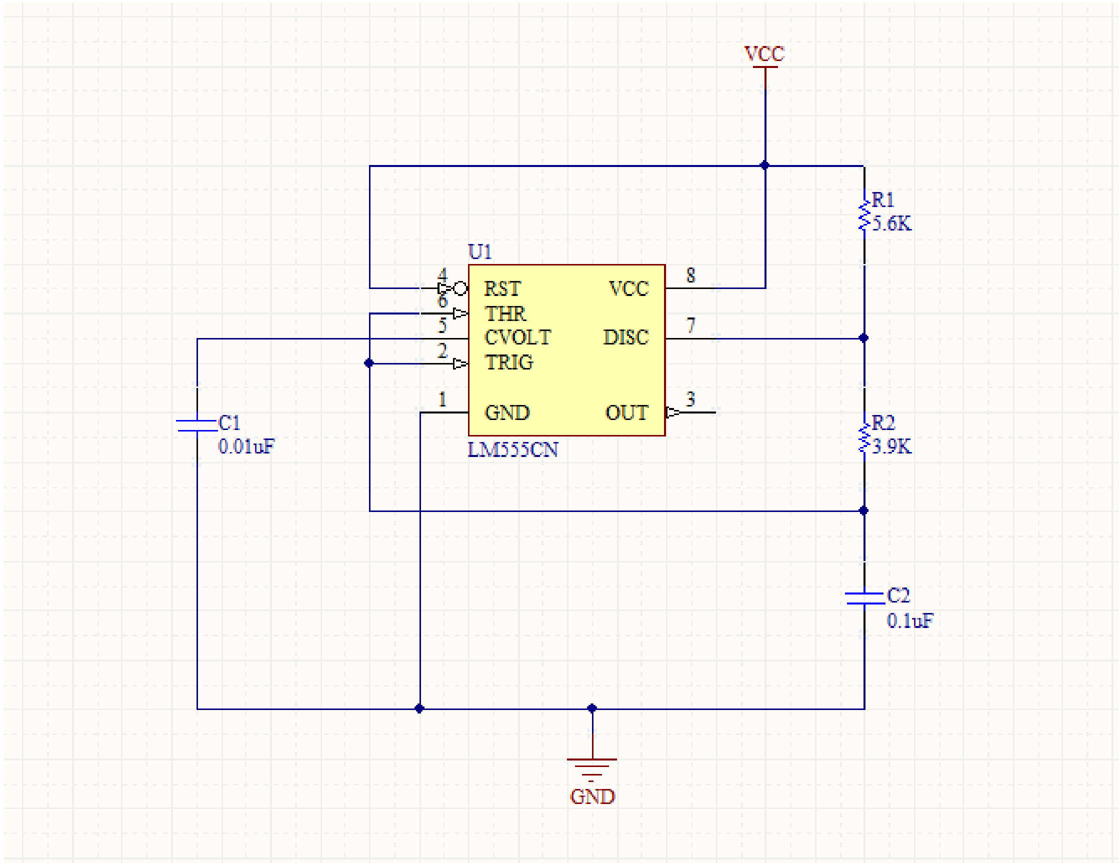


To rename the schematic file go to **File>Save As** and save the file in the same directory as the project. If the schematic sheet is not automatically added to the project you can drag the file in the Project panel from free documents to the project. Before starting creating the circuit several schematic options may be set by going to **Design>Document Options**. Here we will only change the Standard Style to A4.

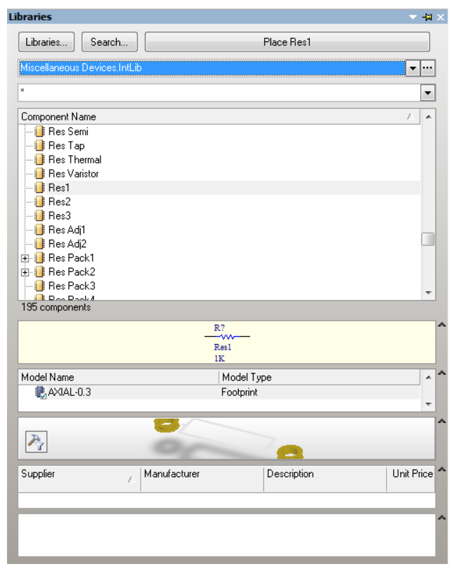


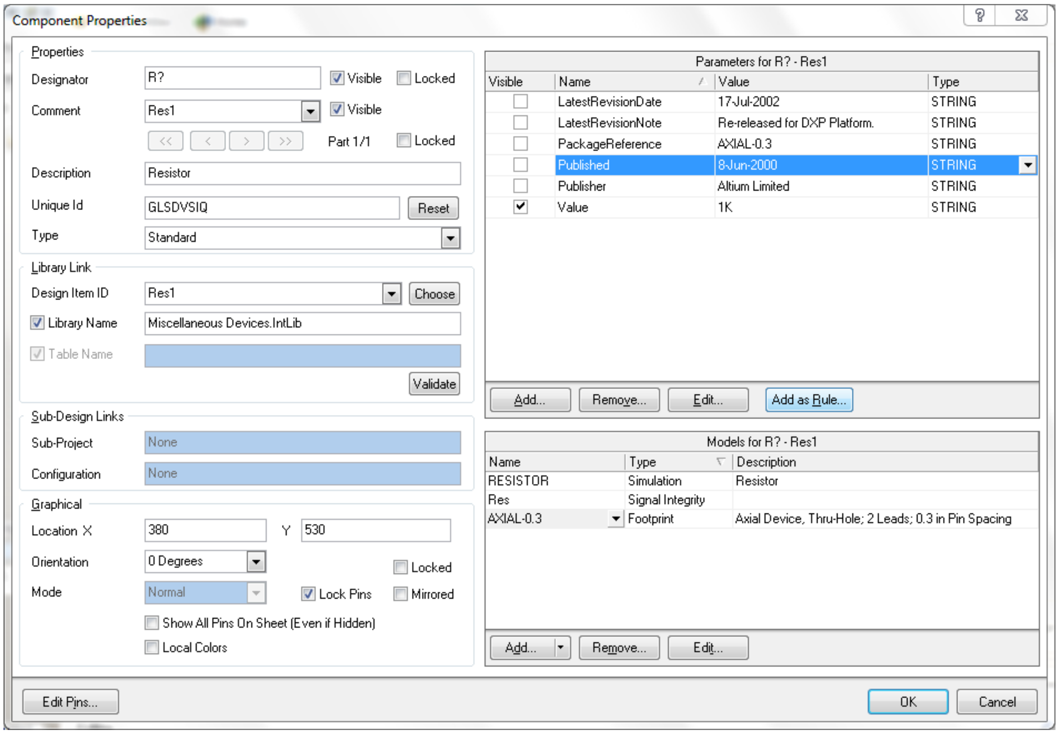
Drawing the Schematic

Now that the project is setup, we are ready to start drawing the actual schematic. For this demonstration we will be using the circuit below.



To start out, let's add all the components needed for the schematic. Altium provides many components in its libraries, and we can search through these libraries to find the components we want in our schematic. Click on the **Libraries** tab on the right to display the Library panel. In the drop down select **Miscellaneous Devices.IntLib** and scroll to **Res1** and select **Place Res1**. To place the resistor on the schematic, just left click. Right clicking will exit the place mode. To edit the properties of the resistor such as the designator, label, and value press **Tab** before you place the resistor.





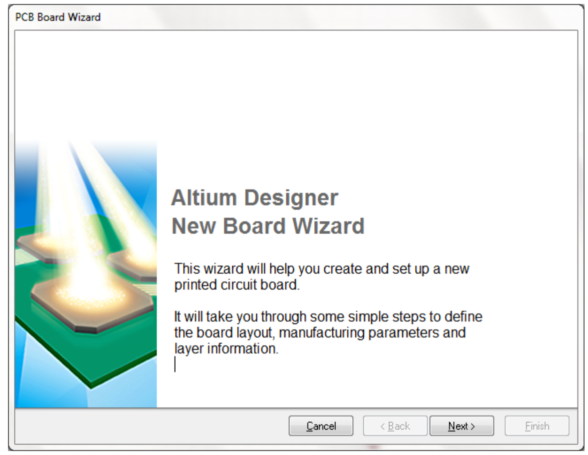
In this window change the Designator to **R1** and its value to the appropriate value and uncheck visible next to the comment. While placing the component the **spacebar** can be used to rotate the component, the **x** key will flip the component horizontally, and the **y** key will flip the component vertically. Place both **R1** and **R2**. Remember to change the value of R2 before you place it. The designator will automatically change. Next select **Cap** in the Library Panel, modify the values, and place the capacitors.

Next we need to place the LM555CN timer. If the library for the timer is not installed go to [http://www2.altium.com/forms/libraries/designer/IntLib_list.asp and download the NSC_420805.zip](http://www2.altium.com/forms/libraries/designer/IntLib_list.asp) file. The desired library is called **NSC Analog Timer Circuit.IntLib**. Once the file is downloaded it needs to be installed. In the libraries panel click **Libraries>Install** and select the desired library. Once the library is installed we can select LM555CN and place the timer on our schematic. To wire the schematic click **Place>Wire** and connect the components. The GND and VDD symbols can be found on the toolbar here. Save the schematic.

Creating the PCB

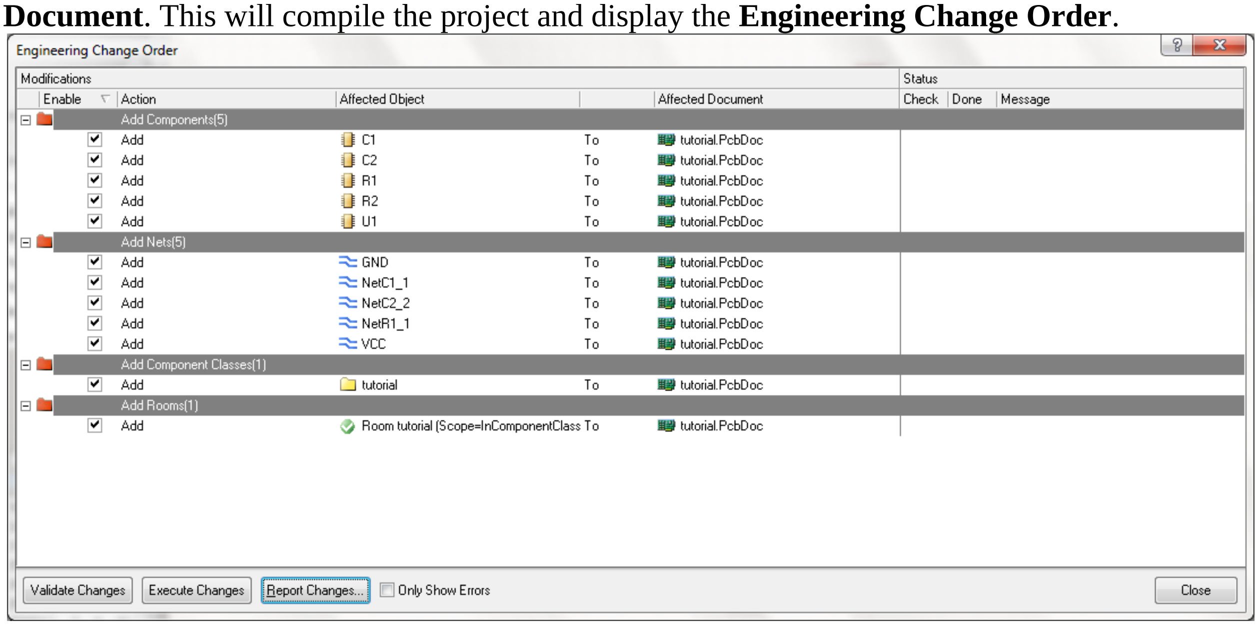
The next step is to create the PCB file. In the Files panel on the right, under **New From Template** select **PCB Board Wizard**.

1. When the wizard opens select **Next**.
2. Set the measurements to **Imperial**, 1000mil = 1 inch.
3. The next view lets you select the board outline to use. Select **Custom** and **Next**.
4. For this tutorial we will use a 2 x 2 inch board. Type 2000 in both the **Height** and **Width** fields. Deselect **Title Block & Scale, Legend String,** and **Dimension Lines**. Click **Next**.
5. This page allows you to modify the number of layers for the board. We will be only using 2 layers. Click **Next**.
6. Select **Thru hole Vias only** and **Next**.
7. This window sets the component/track technology. Select **Thru hole Components** and **One Track**.
8. This window sets the design rules for tracks and vias. Click **Next**.
9. Click **Finish**. The PCB setup is complete. Save the file in the same place as before. If the file is not in the project drag it into the project. The display is showing the board in black and the sheet in white. Select **Design>Board Options**. Deselect **Display Sheet** and click **OK**.

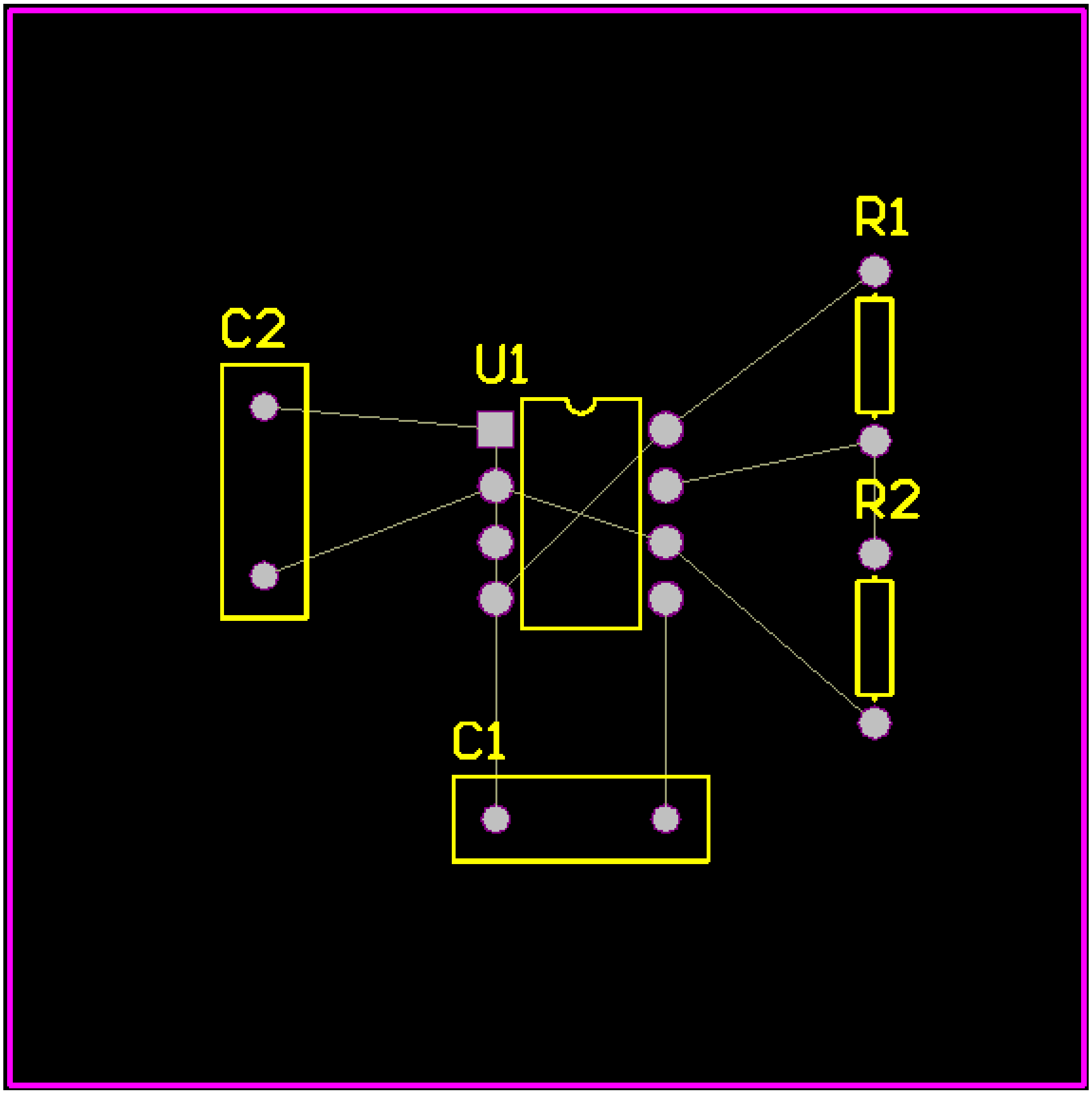


Routing

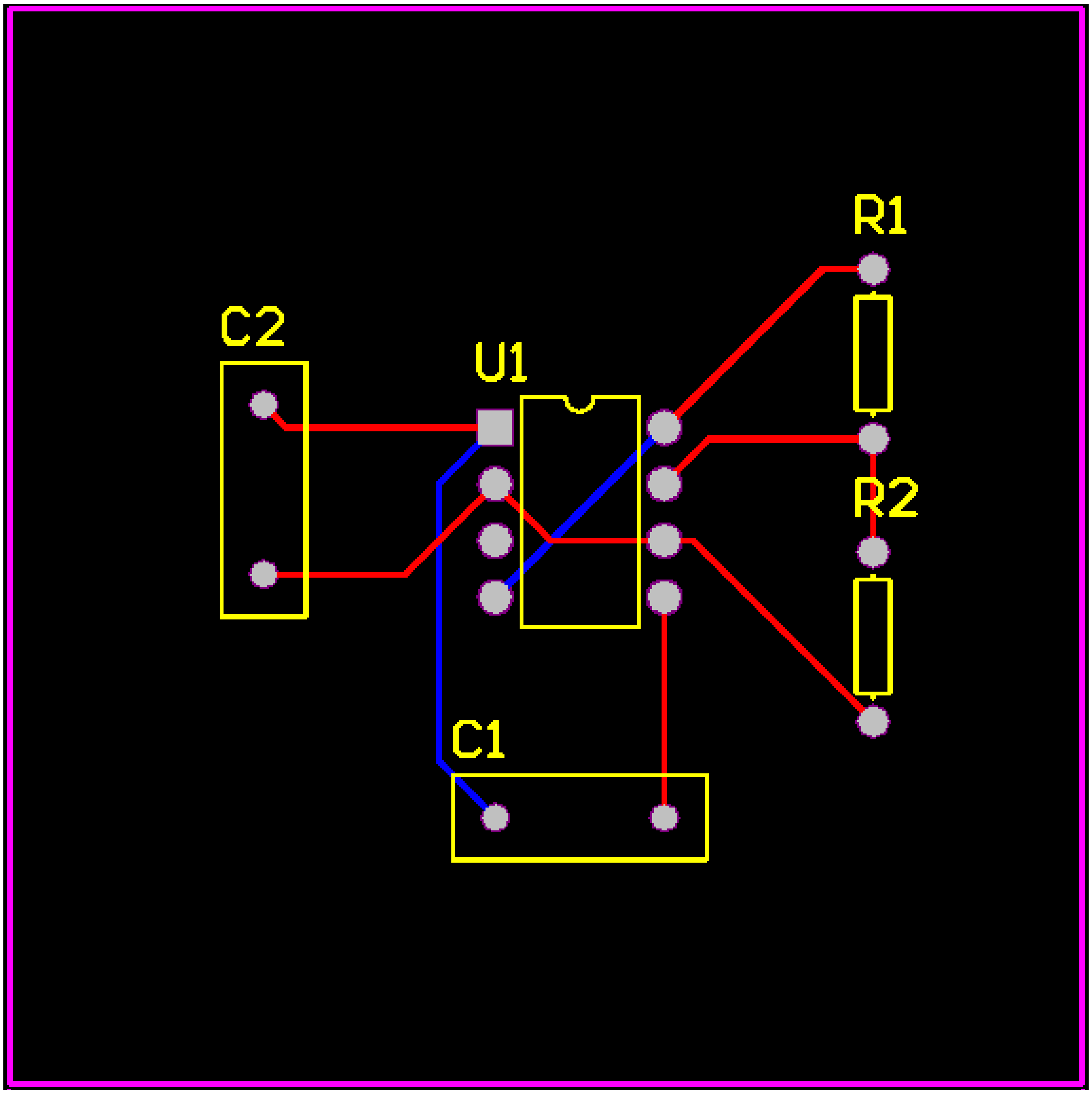
Before routing the components, we must transfer the components to the PCB file and place them on the board. Go back to the schematic file created earlier. Select **Design>Update PCB Document**. This will compile the project and display the **Engineering Change Order**.



Select **Validate Changes**. If the changes are not validated check the **Message** panel. If they are validated, select **Execute Changes** and **Close** the dialog box when it is done. To position a component, click and drag it to the desired location. Rotation can be done in the same fashion as before. Position the components in a fashion such as this. As you move components Altium Designer displays the lines showing the connections still necessary.



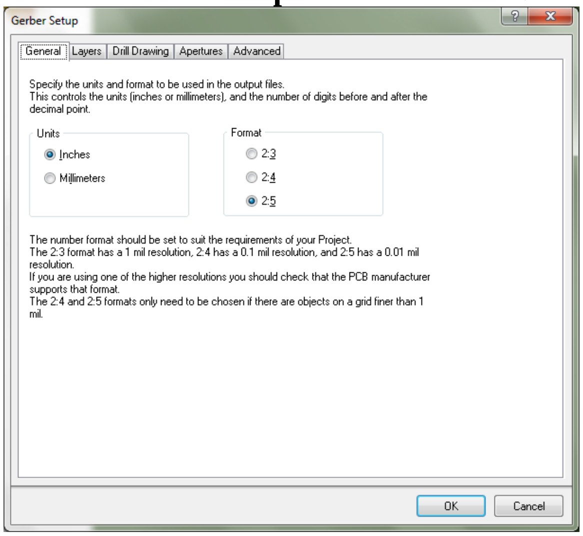
To manually route the circuit, select **Interactive Routing** on the tool bars above. The cursor will change to a cross hair indicating you will be placing tracks on your board. Make sure **Top Layer** is selected at the bottom of the screen. This indicates you will be laying traces on the top copper layer. To lay a trace on the bottom layer select the **Bottom Layer** tab and then route. As you route Altium Designer shows you what traces you have already laid, but also shows a look-ahead path. It will also display a connection line showing you where you must go. Wire the components up using these techniques to get a board similar to this. You may want to route a few tracks on the bottom layer to make the design smaller. The cursor should snap over the connection points of the components creating an easier time wiring.



Altium Designer also provides an option to Auto-route your board for you. To do this, un-route the board, and select **Auto Route>All**. The compute will then automatically route the board for you. This option may work well with small designs, however with much larger designs the computer may have trouble or even fail to route connections. Altium Designer also provides the ability to view the board in a 3-dimensional view. Select **View>Switch to 3D**. This is a nice feature because it gives a realistic view of what your board might look like after fabrication.

Output the Project

Once you are finished with your PCB, you will want to output the design in order to fabricate it. Gerber files are the files that are used by manufacturing companies to create the design. To generate these files from the finished design select **File>Fabrication Outputs>Gerber Files**. In the **Layers** tab select all the layers used in your design. In this case we have Top and Bottom layers, Top overlay, Top and Bottom Solder, Keep-Out, and Multi-Layer. To be safe, you can output all of them and if anything is not used, the file will not be created when you export the Gerber files. Click **OK**. To export the files select **File>Export>Gerber** select **OK** and give it the location of where to save the files.



References

"Getting Started with PCB Design", Altium. http://www.altium.com/files/Altiumdesigner6/LearningGuides/TU0117%20Getting%20Started%20with%20PCB%20Design.PDF