

Designing a Schematic and Layout in PCB Artist

Application Note

Max Cooper

March 28th, 2014

ECE 480

Abstract

PCB Artist is a free software package that allows users to design and layout a printed circuit board (PCB). PCB Artist is provided by Advanced Circuits and requires the user to purchase the PCB through the companies website. This application note is set up to describe the process of how to develop a printed circuit board including placing components, creating symbols, completing the schematic, and layout the design.

Keywords

Printed Circuit Board, Schematic, Layout, Net Name, Net Class, Copper, Design Rule Check

Table of Contents

Introduction	3
Objective	4
Body	
1) Getting Started	4
A) Download	4
B) Creating A Project	5
2) Schematic Design	6
A) Adding Component	6
B) Wiring Components	8
C) Net Name and Class	9
3) Layout Design	10
A) Schematic to PCB	10
B) Placing Components	11
C) Routing Traces	12
D) Copper Pour	13
E) Design Rule Check	14
Conclusion	15
References	16

Introduction

Advanced Circuits was founded in 1991 as a business to fabricate PCBs. In 2007, Advanced Circuits created software for users to use as layout software for a PCB called PCB artist. The company describes the software tools to include, “schematic to Printed Circuit Board layout design, free autorouter, gerber format, multi-page schematic & Netlist import as well as a huge library of over 270,000 parts” (Advanced Circuit).

The two main features in PCB artist include the ability to create a schematic and layout. A schematic is a visual detailed representation of all the components the user would include in their layout. Schematics include the wire connections between each component, net names, and classification of the signal for the layout as a power, signal, or ground signal. A convenient feature in PCB artist is Advanced Circuit continuously updates the library in each software update to maintain the most recent components on the market. PCB Artist also includes the footprints of many general components that a user can use to refine to specific ports. One example is an 8-pin op-amp using an SO-8 layout symbol and determining which ports are used for input and output.

After completing the schematic a user must complete a layout to complete the process to fabricate a PCB. A layout is the final step in the process and includes traces, and placement of your component on the PCB. Most general layout software output the design of the layout in a gerber format. One unfortunate part of using PCB artist is that Advanced Circuits requires you to purchase the PCB through the company and therefor does not supply the user with the gerber file. On the other hand the software does maintain the ability to use specific algorithms to conveniently auto route the traces for the user. It is always better to manually route the traces, but the auto route feature can give the user a good starting point if the user has never experienced routing traces on a PCB before.

Objective

Throughout this application note, the user will learn the fundamental tools necessary to design a printed circuit board. This application demonstrates one example of how to create a low-noise inverting op-amp. The PCB will include bypass and decoupling capacitors, test points for ground and power, and the necessary components for the op-amp.

1) Getting Started

A) Download

To start, click on the following link and select the download now button.

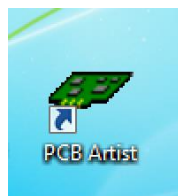
<http://www.4pcb.com/free-pcb-layout-software/>.



PCB Artist is currently only compatible with Windows operating systems. PCB Artist recommends your computer have at minimum the following specifications:

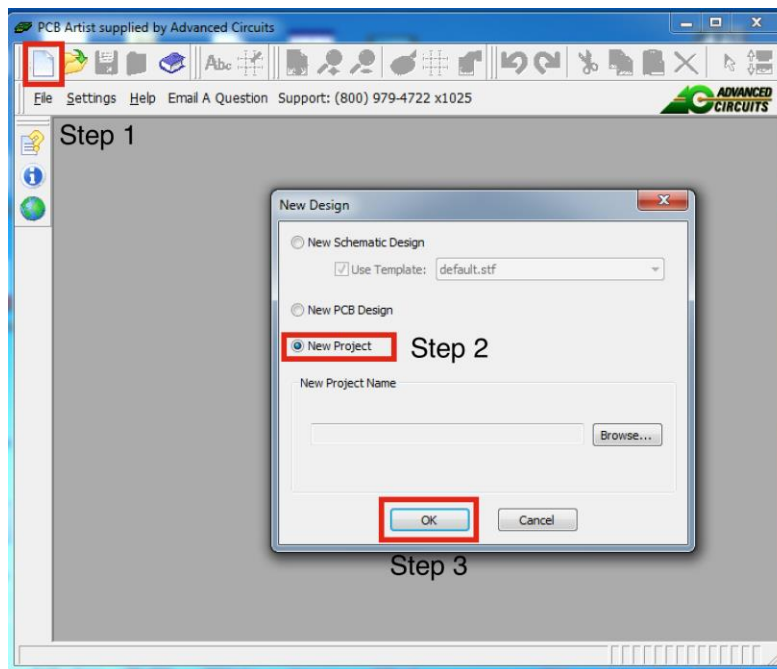
- Pentium Processor faster than 1 GHz
- At least 256MB of RAM
- Installation requires 100MB of Hard Drive Space
- Wheel Mouse is recommended

Save the file to a folder of your choice and open the program.

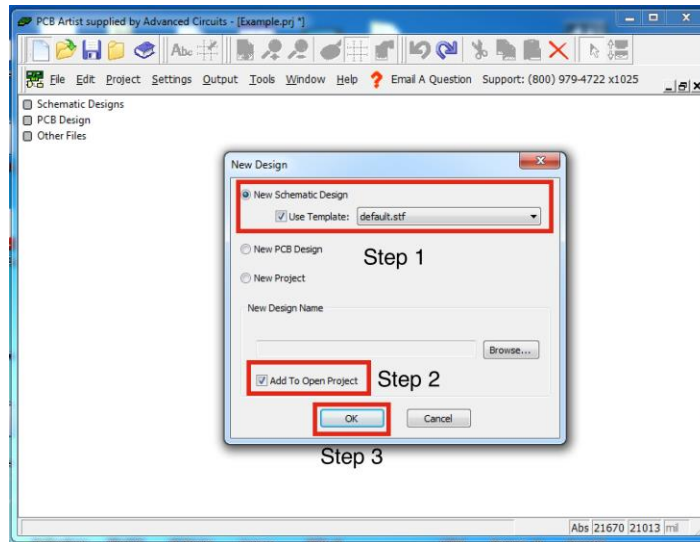


B) Creating a Project

Once you open the program you will see a blank screen with the following toolbar in the image below. The user can select file -> new, or press the blank page to create a project for the PCB Design. Once the new design prompt appears, select 'New Project' and place the project into a directory of choice. This directory will be the same directory that includes the schematic and layout files.



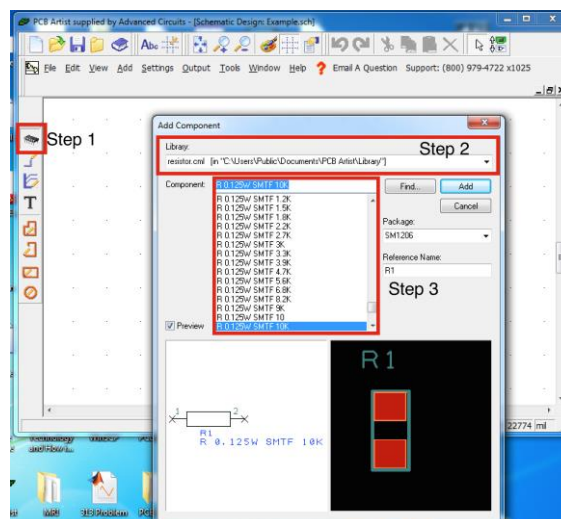
After the user creates a project the user will follow the same process except instead of selecting 'New Project', the user will choose 'New Schematic Design.' The user must also remember to check the box to 'Add To Open Project' in order to keep the project and schematic files linked. Once you press 'OK', enter the name of the schematic and click save. The user is now ready to begin creating the schematic.



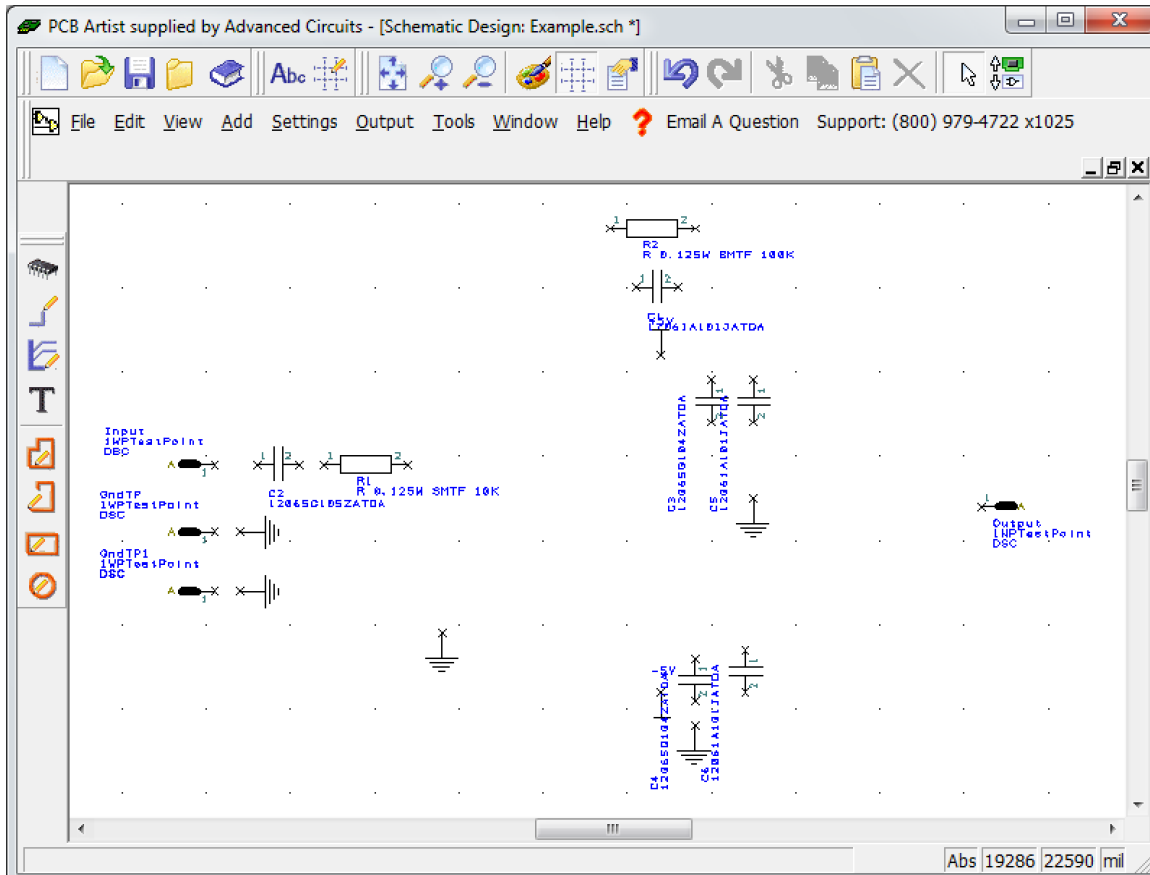
2) Schematic Design

A) Adding Components

The first step to begin the schematic design is to place all of the available components from the PCB Artist library into the Schematic. This can be completed by first clicking the add component button (picture looks like an integrated circuit) as shown below. Then after select the necessary library for the component the user is looking to place. For example as illustrated below, when trying to place a 10kOhm resistor, first select the resistor.cml library and then select the value of the desired resistor. Also the package lists SMTF for a surface mount package, where you can select a different component for a through hole or different size package.



Once the components are selected, the user must place the components in an order that correlates to the desired view of the design. The schematic should be placed for the user to easily be able to debug functionality of the circuit. Below is an example of the components inside the PCB libraries.

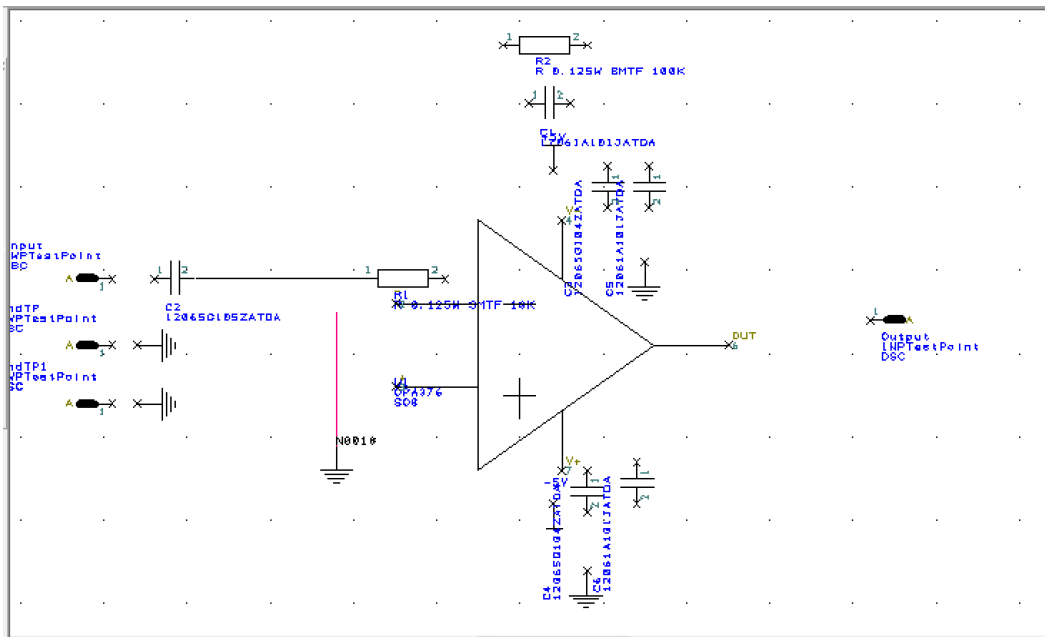


Disclaimer: This application note will not include how to create a new schematic symbol. Please refer to "PCB Artist Tutorial Video – Part 2" at <http://www.4pcb.com/free-pcb-layout-software/>.

B) Wiring Components

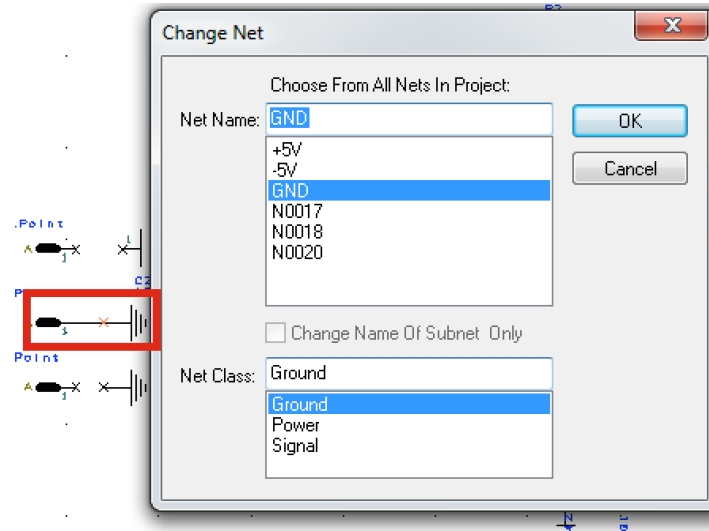
The last essential step of completing a schematic is wiring the components together. This enables the program to understand how to later route the traces in the layout process. A wire also enables the user to set the type of wire to determine whether it is a power, ground, or signal.

To set up a wire the user can double click on the edge of a component where the wire should be attached labeled with an “x” and drag the wire to another component. Another option is to select ‘Add Schematic Connection’ (directly below the add component button) and then you can single click anywhere on the screen to create the connection. In the image below you can identify a red wire. This wire represents one that is not attached and is therefore presenting an error. On the other hand, if the wire is attached the connection will be in the common black color as is represented in the component.

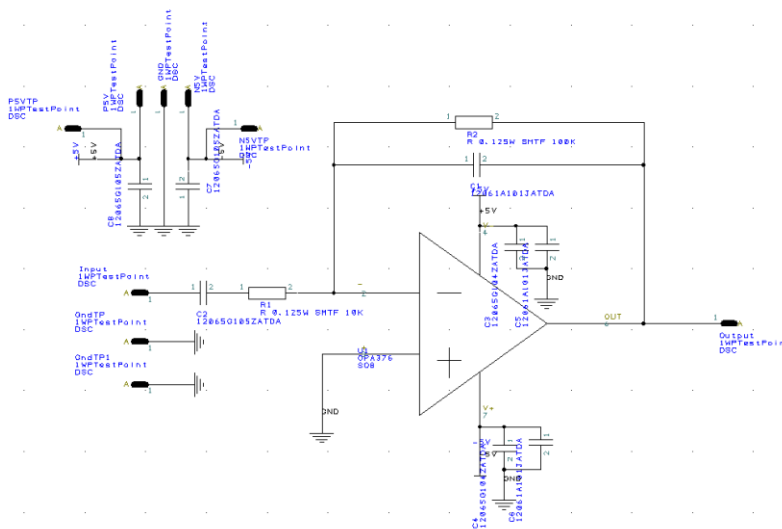


C) Net Name and Class

After ensuring there are no unconnected components, the user can modify the net names of each wire to define the signal type. The user can right click a wire and select 'Change Net...' and then pick the correct Net class. For example, the image below connects a ground test point to ground and by right clicking the wire; the user can modify the Net of the wire to ensure the layout uses a ground signal.



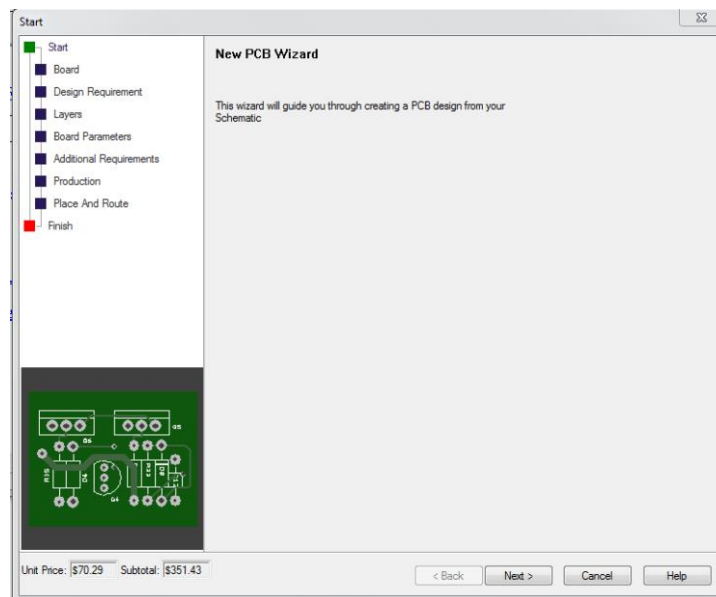
Once the user sets up all the nets and completes the wiring the user has completed the schematic design of the PCB. The image below demonstrates a complete schematic design.



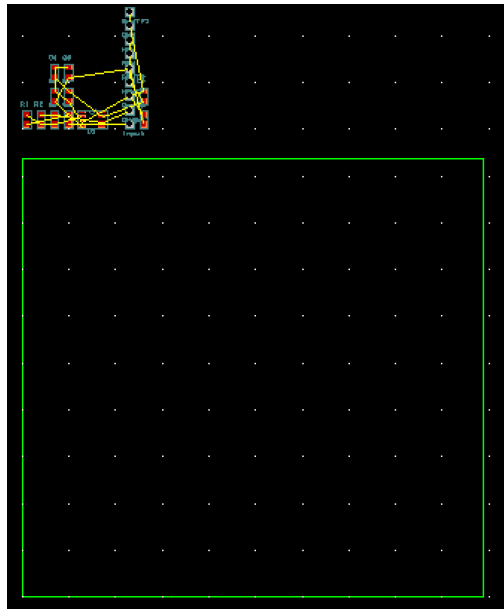
3) Layout Design

A) Schematic to PCB

Once the schematic is complete, the user must create a PCB layout for Advanced Circuits to create a PCB for the user to place the components on a board and include the traces that represent the wires in the schematic. In order to begin the layout, the user can use the program to forward the components from the schematic to the layout. While the user has the schematic window as the selected window, select 'Tools' -> 'Schematic < -- > PCB' -> 'Translate to PCB...' The image below will be the first image to appear.

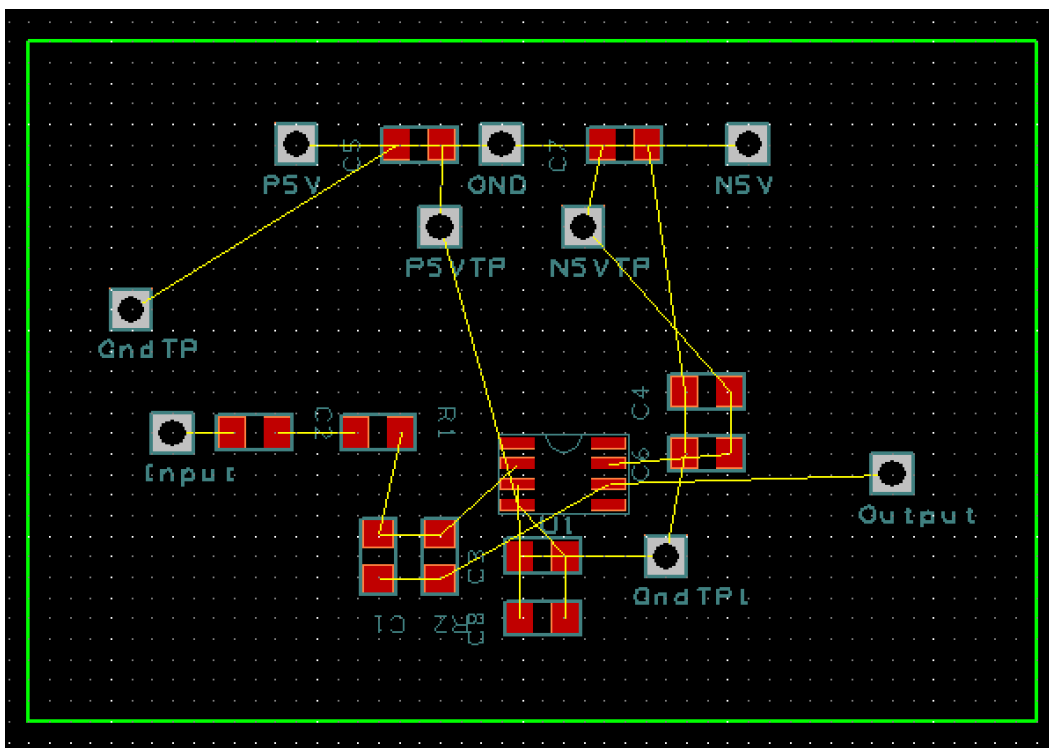


The user then can select the required parameters of the PCB to create a board of the users desired specifications. The parameters PCB Artist allows you to configure include the board size, a standard layer board according to a default description, or custom, modify layers, additional notes for Advanced Circuits, how many boards, and where to initially place the components. If the components are placed outside of the board, the following image will appear with standard settings from PCB Artist.



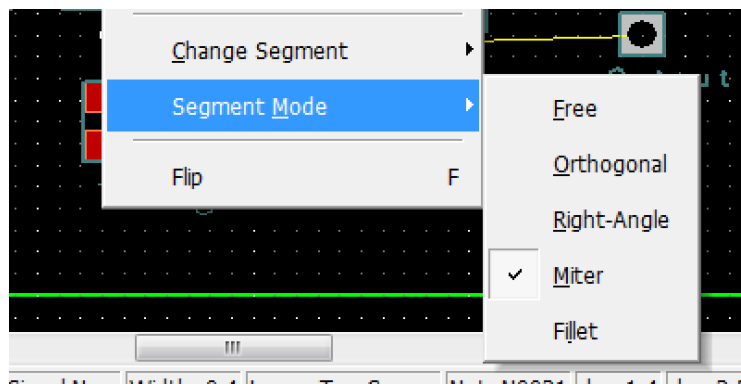
B) Placing Components

Once PCB Artist has imported all the components from the schematic it is now the user's job to properly place the components for the layout of the fabricated board. It is important to avoid having any text underneath the pads as the fabricated board will not allow the user to know which component is supposed to go in that location.

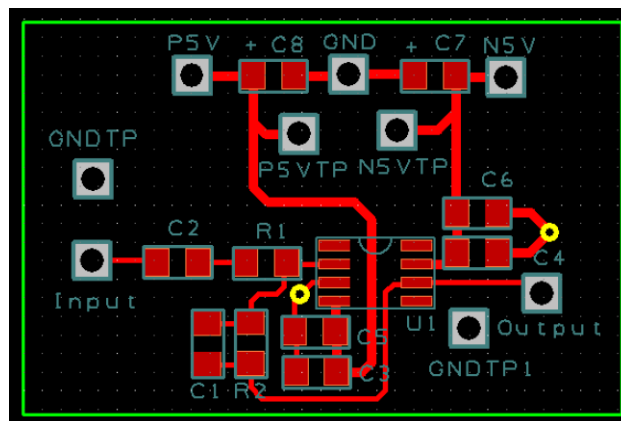


C) Routing Traces

The yellow lines in the layout represent the traces or wires from the schematic. A great tool used in PCB Artist is it represents the closest trace from the nearest component to allow the user to properly orient the board. When initially starting to layout the board the user should double click on a pad and drag the trace slightly, right click -> 'Segment Mode' -> Miter. It is essential to go into Miter mode, as it will allow the user to create traces at 45-degree angles instead of 90-degree angles. This is crucial for high frequency traces to prevent unwanted characteristics of the PCB.

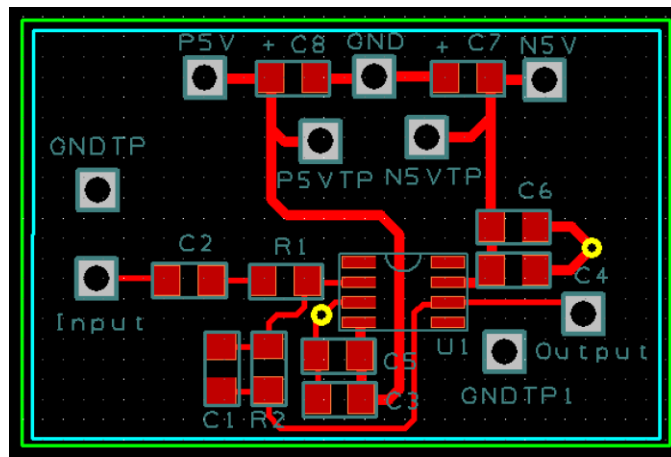


The image below represents a fully routed PCB. It is important to notice there are wider and skinnier traces. This occurs because of the different net class's selected in the schematic. The Power and Ground traces are going to be wider than a signal trace as power traces generally occur at higher currents.

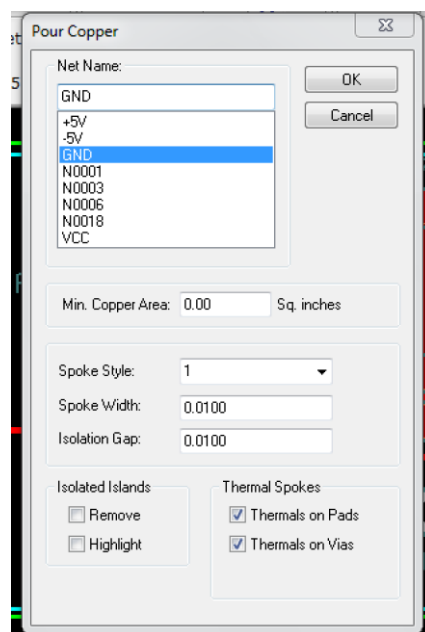


D) Copper Pour

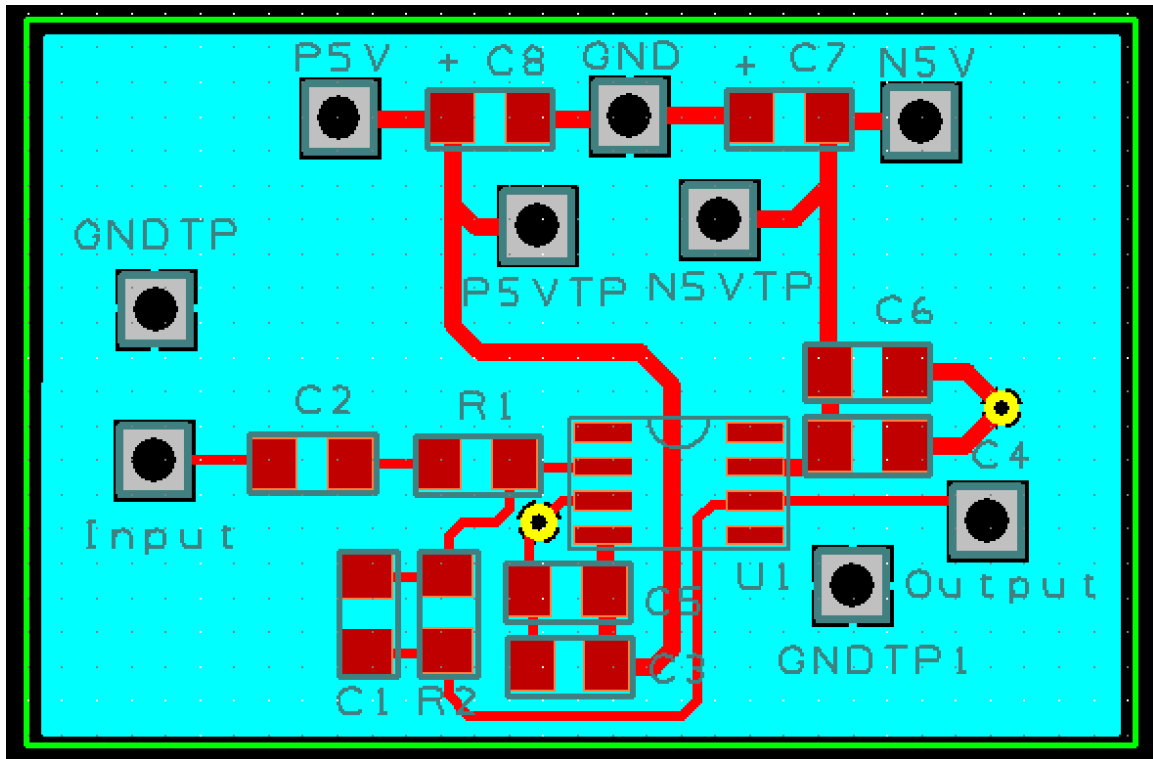
A great feature Advanced Circuits provides with their PCB's includes the ability to pour copper over an entire layer of a PCB. When a ground copper layer is poured it will help prevent noise and present an opportunity to have easy access to ground with the test points and prevent the necessity of longer traces. To add a copper pour the user must first outline the outside of the board with the area of the copper pour. Selecting 'Add Area for Copper Pour' then outlining the desired section completes this step. This will create the blue outline seen below on the PCB Artist window.



The image below represents that you can pour copper of different Net Names, but ground is the most common.

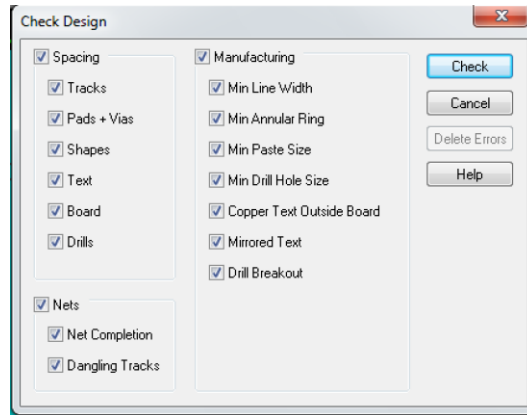


Once the copper is poured the light blue in the image below represents the copper pour. As you can see the light blue does not cover any component placed on the top layer. This is a very convenient feature of the program to prevent having to pour copper multiple times on the same layer.

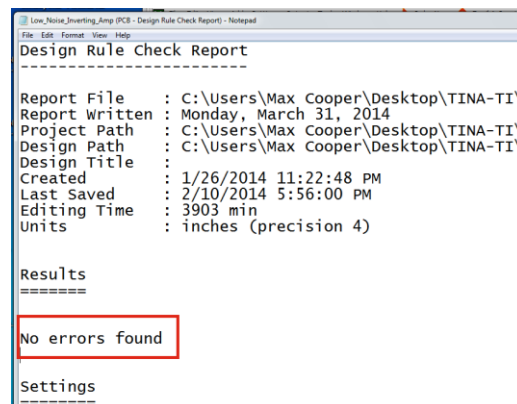


E) Design Rule Check

The final step before submitting the PCB for fabrication is to ensure there are no Design errors. This can inform the user if they have components overlapping, traces too small compared to the desired constraints, overlapping traces, and many other errors described in the Check Design image below. In order to run a Design Rule Check (DRC) the user must go to 'Tools' -> 'Design Rule Check...' This process will open the image below.



Once the user selects 'Check' in the above window, a report will appear. If there are no errors the DRC will state "No errors found," and the user is now ready to submit their PCB for fabrication.



Conclusion

PCB Artist is a great software tool for a user to build a printed circuit board. There is a very large plethora of components to select from that creates ease for users to create PCB's with different components. The software includes several more features not detailed in this application note. Some additional features are creating ones own component, multi-page schematics, and easy to measure distances on the layout in any specific unit with any amount of precision from 0 to a desired value.

References

“Company History – Circuit Board Manufacturing.” *Advanced Circuits*. Web. 27 Mar 2014. < <http://www.4pcb.com/advanced-circuits-history/>>.

“Free PCB Layout Software to Download.” *Advanced Circuits*. Web. 27 Mar 2014. < <http://www.4pcb.com/free-pcb-layout-software/>>.