PCB Artist

Introduction and Tutorial

Copyright Notice

Copyright © WestDev Ltd. 1997-2008
PCB Artist is a Trademark of Advanced Circuits. All rights reserved. E&OE

Copyright in the whole and every part of this software and manual belongs to WestDev Ltd. and may not be used, sold, transferred, copied or reproduced in whole or in part in any manner or in any media to any person, without the prior written consent of WestDev Ltd. If you use this manual you do so at your own risk and on the understanding that neither WestDev Ltd. nor associated companies shall be liable for any loss or damage of any kind.

WestDev Ltd. does not warrant that the software package will function properly in every hardware software environment.

Although WestDev Ltd. has tested the software and reviewed the documentation, WestDev Ltd. makes no warranty or representation, either express or implied, with respect to this software or documentation, their quality, performance, merchantability, or fitness for a particular purpose. This software and documentation are licensed 'as is', and you the licensee, by making use thereof, are assuming the entire risk as to their quality and performance.

In no event will WestDev Ltd. be liable for direct, indirect, special, incidental, or consequential damage arising out of the use or inability to use the software or documentation, even if advised of the possibility of such damages.

WestDev Ltd. reserves the right to alter, modify, correct and upgrade our software programs and publications without notice and without incurring liability.

Microsoft, Windows, Windows NT and Intellimouse are either registered trademarks or trademarks of Microsoft Corporation.

All other trademarks are acknowledged to their respective owners.

Number One Systems & PCB Artist are trademarks of Westdev Ltd and Advanced Circuits respectively.

Issue date: 18/09/08 iss 3

Advanced Circuits Inc.
21101 East 32nd Parkway
Aurora, CO 80011
Toll free: 800-979-4PCB (4722)
1-303-576-6610
Fax: 1-888-224-3291

Contents

CONTENTS 3
CHAPTER 1. GETTING STARTED 5
Introduction to PCB Artist 5
Introduction to the Tutorial 5
Additional Help 5
System Requirements 5
Installation 5
Removing the Software 6
Starting PCB Artist 6
The PCB Artist Desktop 6
Toolbars 7
Cancelling Commands 8
Shortcut Keys 8
Units 8
Grids 9
Shortcut Menus 10
Undo/Redo 10
Properties of items 11
Status Bar 11
Dockable Windows and Browsers 12
CHAPTER 2. CREATING THE SCHEMATIC 13
Schematic Design Tutorial 13
Starting a New Schematic 13
Adding Components 14
Placing Components 25
Adding Connections 26
Adding Power & Ground Symbols 27
Adding Connectors 28
Naming Connections 30
Your Schematic is Finished 32
CHAPTER 3. CONVERTING YOUR SCHEMATIC TO PCB 33
Converting the Schematic to PCB 33
Starting a New PCB Design 33
The New PCB Wizard 34
Schematic Design Completed 38
CHAPTER 4. PCB DESIGN EDITOR 39
Getting Started with the Design 39
Converting the Schematic to PCB 39
Changing the Number of Layers 40
Creating a Board Outline Interactively 40
Placing Components 40
Routing the Design 41
Automatic Routing 42
Unrouting the design 43
Manual Routing 43
Picking in tight areas 46
Pouring Copper into Areas 46
Checking the Design Integrity 50
Design Rules Checking 51

Viewing Power Plane Layers 52
Reports, Part Lists and Net Lists 53
Manufacturing the PCB 53

Chapter 1. Getting Started

Introduction to PCB Artist

Welcome to the PCB Artist electronic design system. The system is built on an integrated design environment providing all the tools required to capture a schematic through to the design and layout of the printed circuit board (PCB).

Introduction to the Tutorial

You can quickly familiarize yourself with the PCB Artist features available and its design methodology by working your way through this tutorial and using the Online Help for more detail.

This tutorial is broken down into logical sections following a typical Schematic and PCB design cycle.

The Schematic design editor is used to capture your logical design and 'drive' the PCB design. However, the PCB design editor can be used with or without the schematic design as you wish.

An important aspect of the PCB design is the production of professional manufacturing data, and hence the final PCB. PCB Artist has the facility to send the manufacturing data directly to your PCB supplier. From this data they can produce the final physical PCBs.

Additional Help

Help is available in a number of formats; on-line help available by pressing the **F1** key on your keyboard at any time when using PCB Artist or by emailing/calling Advanced Circuits.

System Requirements

PCB Artist runs under the Windows operating systems but it is recommended that Windows XP is used. It cannot run under Windows 3.1x or Linux. A Pentium processor faster than 1Ghz and with at least 256Mb of RAM is recommended. It should be emphasized that PCB Artist does not require particularly 'high-powered' hardware to achieve good performance, a regular off-the-shelf PC should be sufficient. A complete product installation requires at least 100Mb of hard drive space. A mouse with a wheel is recommended and an internet connection for downloading the product from the PCB Artist web site. A CD-ROM drive is an option to load the product CD if it has been supplied this way.

Installation

It is assumed that you have successfully installed the PCB Artist product to your computer from the web download. The installation is straight forward using the wizard provided. Follow the instructions and use the default settings provided. The tutorial designs are also installed to allow you to pick up at key stages of the design process.

Removing the Software

If for any reason you need to remove PCB Artist from your computer, you should do this by selecting the **Add/Remove Programs** icon from the **Control Panel** on the **Start** menu. Select **PCB Artist** from the list of installed programs and click the **Change/Remove** button.

Starting PCB Artist

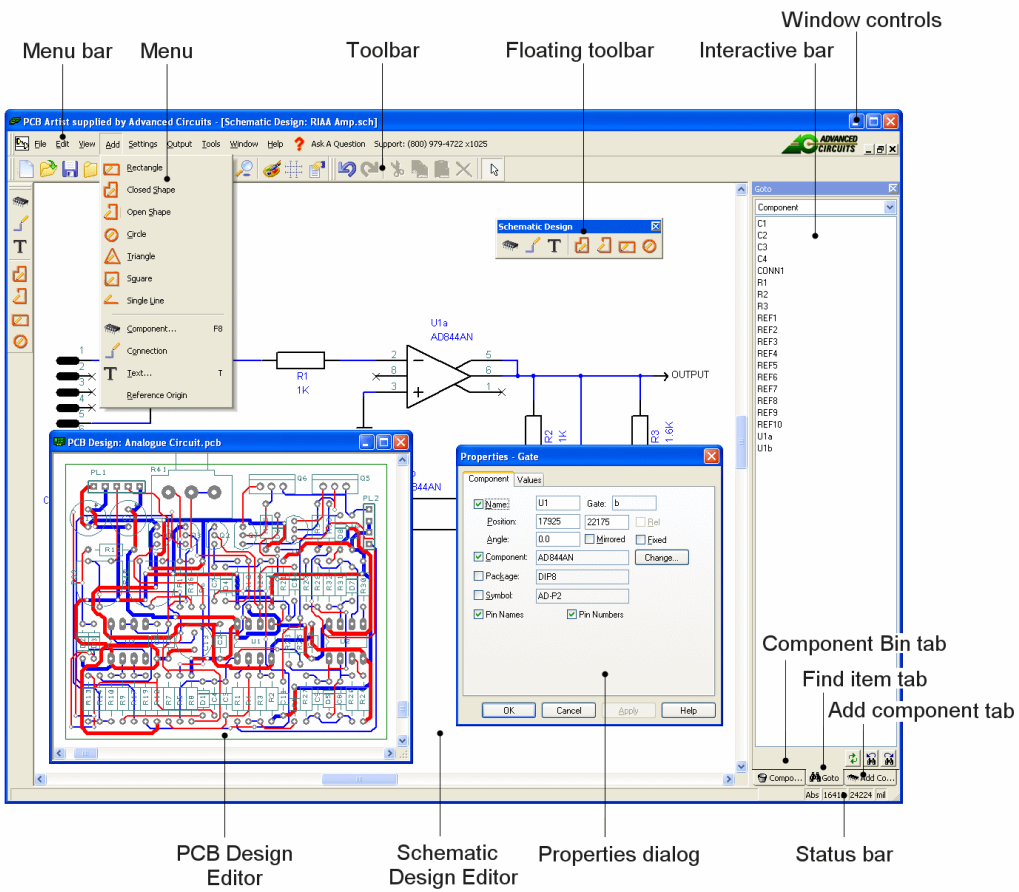
During installation, a PCB Artist program icon is automatically added to your desktop and a Advanced Circuits program group added to the **Start** menu and **Programs**.

To start PCB Artist, simply double-click on the **PCB Artist** icon on your desktop or select the PCB Artist menu item.

The PCB Artist Desktop

When you run PCB Artist, the main application window appears. You can open any number and combination of different designs and library items together in this.

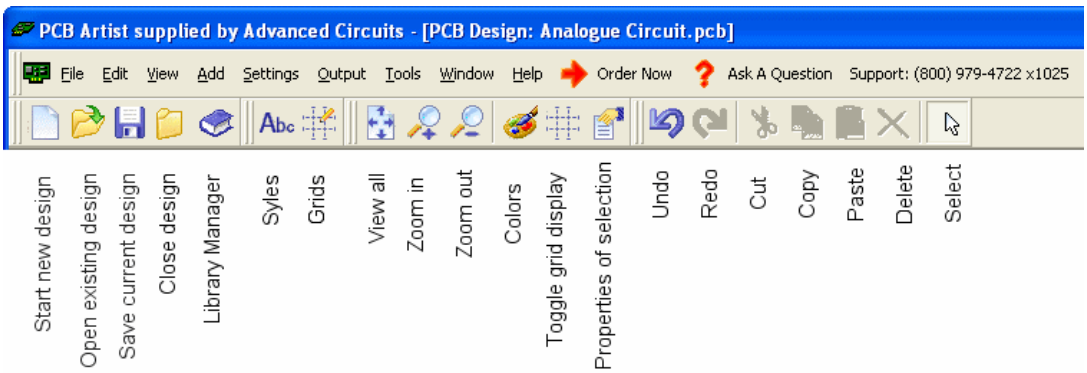
The picture below shows you the major framework facets by name so that as they are used in this tutorial you will understand what is being talked about.



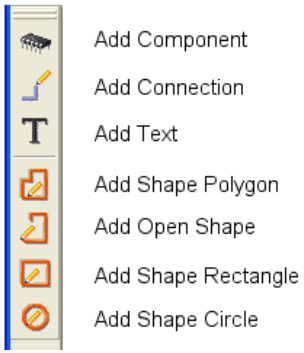
Toolbars

PCB Artist is installed with a set of commonly used tools on the toolbars for you to use.

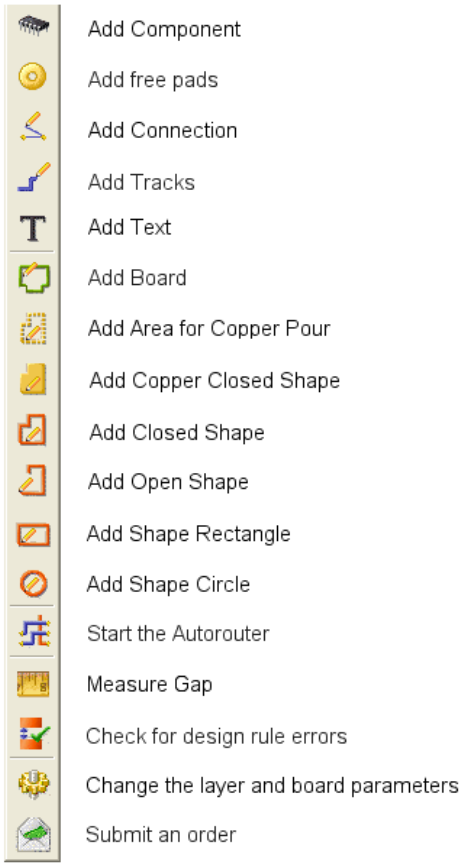
General Toolbar

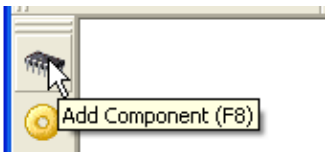


Schematic Design Toolbar



PCB Design Toolbar





If you hover the mouse over the toolbar buttons a small tooltip is displayed showing the button function and its keyboard shortcut if one has been defined.

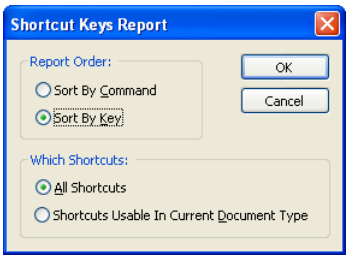
Cancelling Commands

All dialogs and commands can be cancelled at any time by clicking the **ESC** key on the keyboard. Sometimes this will need to be clicked a couple of times if you are 'down' in a sub-menu or lower-level option.

Shortcut Keys

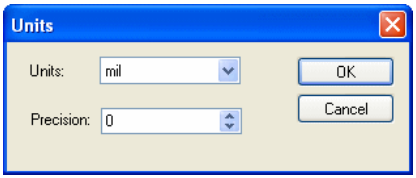
As well as the toolbars, PCB Artist has the ability for you to program your keyboard for commonly used options or commands. This means that once programmed, you can press a key or key combination to enter a function or option. For experienced users, this makes the program's operation far more efficient.

When PCB Artist is installed, you will find a set of shortcuts defined for you. A full report of all the assigned keys can be obtained from the **Help** menu and **Shortcut Keys...** option. A dialog is displayed to allow you to choose how the shortcut keys are reported.



Units

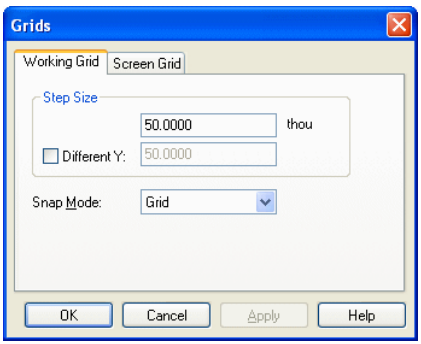
The **Units** option on the **Settings** menu is used to define the units and the precision at which the co-ordinates and lengths displayed in the dialogs and edited by you are displayed. The basic working units can be changed using the **Units:** drop down list.



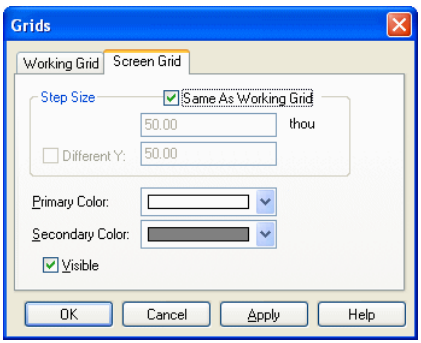
For each type of units, you can specify a **Precision:**. This is the number of decimal places displayed; the value is simply rounded to the nearest last digit.

Grids

Grids are used in PCB Artist for defining the points or steps that items will snap to when they are placed and for visible references in the design. For this, the **Grids** dialog on the **Settings** menu contains two sets of grid types: **Working Grid** and **Screen Grid**.



The **Working Grid** page will allow you to define the grid used for placement of design items during manual interaction. The working grid is not visible but the **Screen Grid** (which is visible) can be the same value, hence making it appear to be visible.

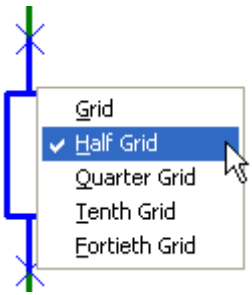


The **Screen Grid** page is used to define the display of 'dots' to guide you while designing. **Size Steps:** is used to define the 'pitch' of the grid and uses the **Secondary Color** to display them. The **Primary Color** is used to display another grid based on a 10 times larger size than the Secondary Grid.

The **Visible** check box enables the grids to be displayed on or off, this can be set at any time in the design using the shortcut key <**G**>.

Snapping to Grid

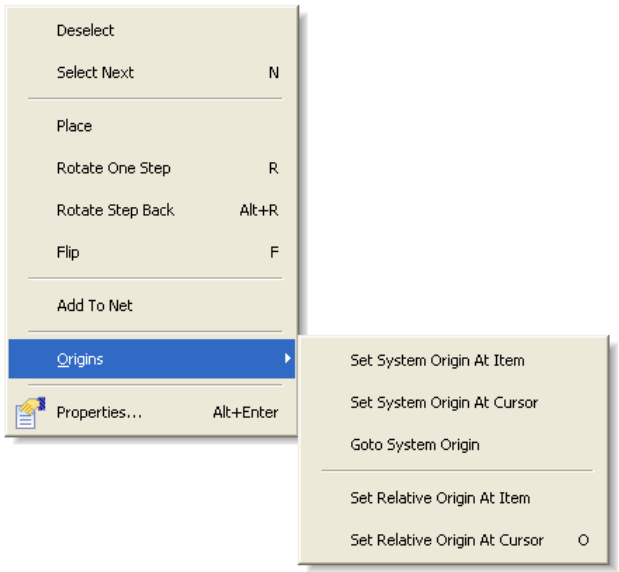
The **Snap Mode** shortcut can be displayed during editing by selecting the direct keyboard command <**Ctrl-G**>. By selecting the snap mode setting you can use predefined multiples of the Working Grid to move items for example.



Shortcut Menus

The right hand mouse button can be used extensively throughout PCB Artist to display **shortcut menus**, these are also known as **context menus**. These menus can be accessed during an operation and contain a set of relevant 'context' commands.

Once familiar with the system, you can access the shortcut menus by right clicking directly on the item and picking the option from the menu.



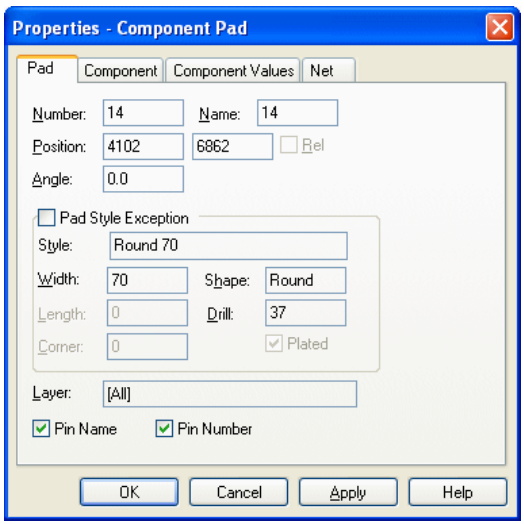
Undo/Redo

PCB Artist contains unlimited multi-level Undo and Redo capabilities for use through the product. Undo can be used to reverse the last operations that have been completed. Undo can be used sequentially to undo more than one operation. It can also be a useful and flexible tool to interactively try different iterations of an operation, manual component placement for example.

Undo and Redo are invoked using the standard Windows shortcut keys **Ctrl-Z** (and Alt-Backspace) and **Ctrl-Y** respectively. These are also available on the Edit toolbar as buttons. Redo is only available once an Undo has been performed, until then, it remains on the menu but is grayed out as unavailable. The number of undo levels can be set in the **Preferences** dialog.

Properties of items

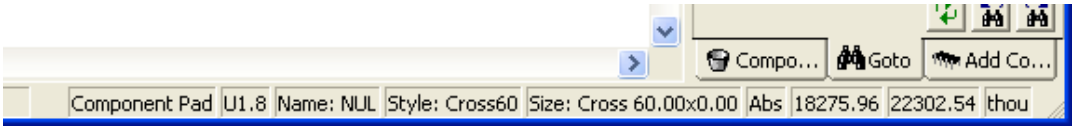
Each **item** in the design contains a set of characteristics which can be viewed or edited using the **Properties** option. Some items within the design are made up of several entities, each of which is individually selectable and has its own properties that can be viewed.



Item properties can be viewed by selecting the item and clicking on the **Properties** option from the shortcut menu. It is quicker though to select the item and click the shortcut key **Alt-Enter**. The example left shows the Component page properties of a PCB Component pad. You'll notice that other properties of the component are available, such as Component and Component Values.

Status Bar

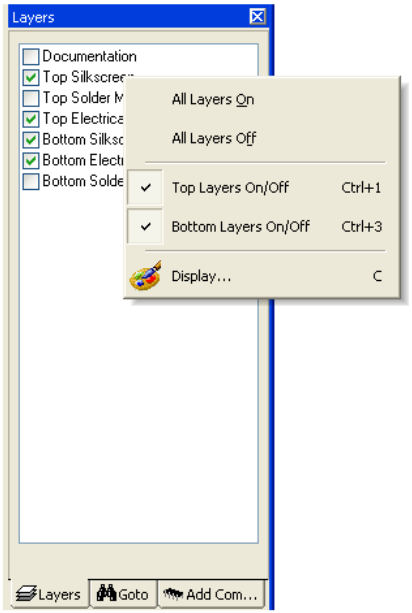
The status bar at the bottom of the design window displays useful information about the design item selected. More detailed information is also available by selecting the item and using the **Properties** dialog from the shortcut menu (key shortcut **Alt-Enter**).



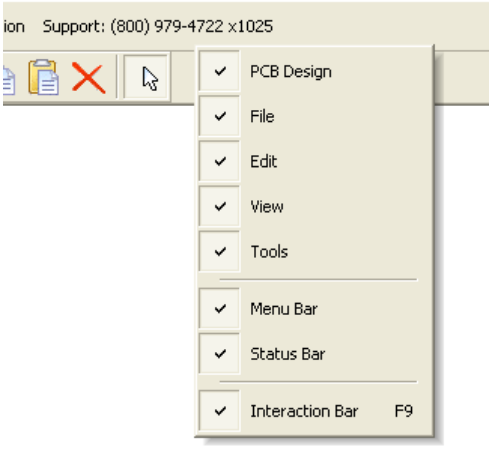
Dockable Windows and Browsers

As part of the design toolset, you have a number of browsers available through the **Interaction Bar**.

Each individual browser can be opened by selecting one of the three tabs at the bottom edge of the browser. You can access **Layers** (in PCB only), **Component Bin** (in Schematics only), **Goto** and **Add Component** options.



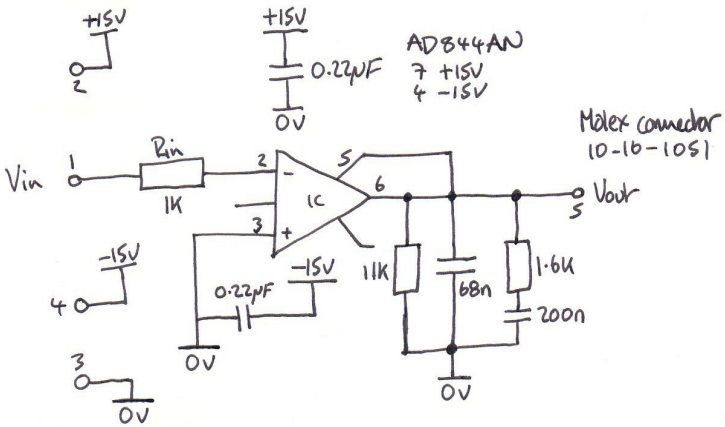
The Interaction Bar is available through the **View** menu and **Interaction Bar <F9>** or by right clicking the mouse over one of the toolbars or menu bar and selecting it from the shortcut menu.



Chapter 2. Creating The Schematic

Schematic Design Tutorial

During the tutorial we will create a Schematic design; this is how the engineer 'conceived' it:

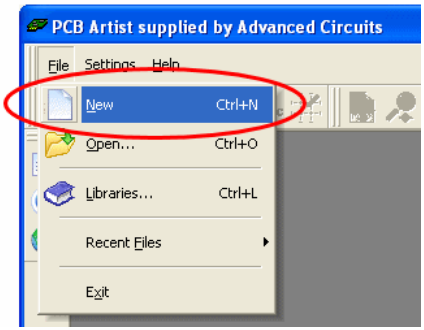


Obviously, once the sketch has been drawn properly as a PCB Artist Schematic it will look quite a bit different.

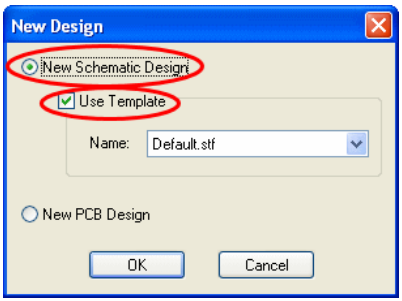
Starting a New Schematic

▶ To start a new Schematic design or a PCB design

From the **File** menu, click **New** (shortcut key **Ctrl-N**)



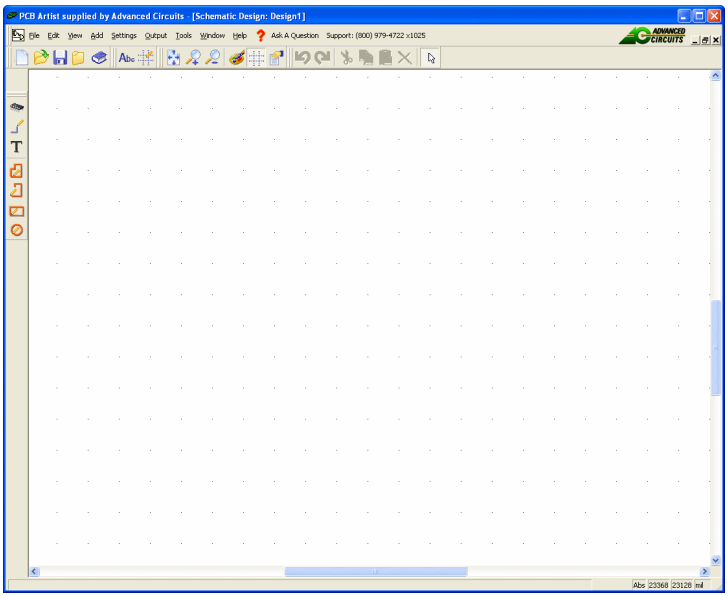
Check the **New Schematic Design** button and the **Use Template** button. The Default.stf template will help you get started quicker. More detail about Template files is available in the Online Help.



Click the **OK** button to start a new Schematic design.

Note: If you intend creating a PCB without a Schematic design (which you can do), just click the **New PCB Design** button and **OK**. Then skip to the chapter headed, ***Starting a New PCB Design***.

Your PCB Artist window will look like this ready to start:



For now, we will progress on to adding components to the design.

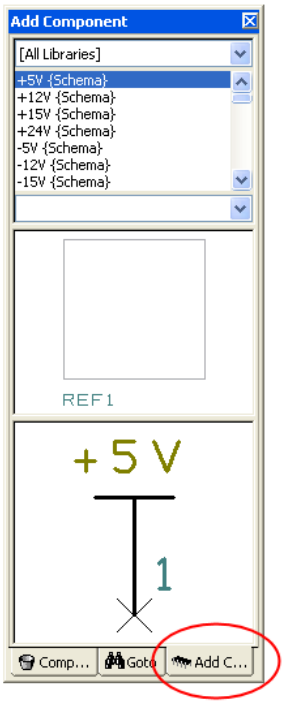
Adding Components

▶ To add a Component

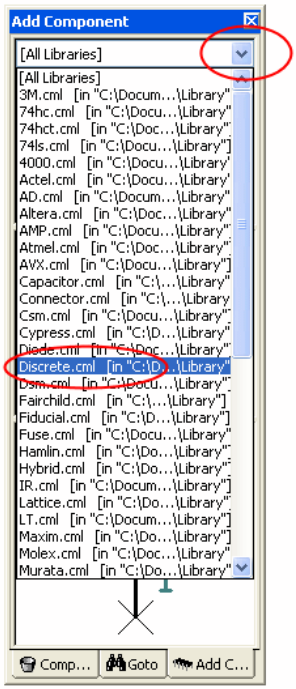
There are two methods for adding components; we will use the **Interaction bar** to start with, you can also use the Add Component option as well.

The Interaction bar has automatically been started when creating a new Schematic design. If it hasn't, click the <**F9**> shortcut key to display it.

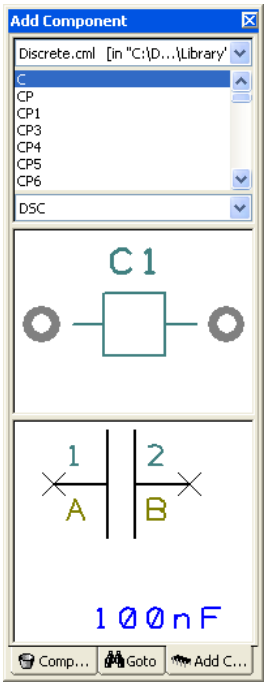
Select the **Add Component** browser tab at the bottom of the Interaction bar.



From the list of available libraries, select the **Discrete** library. Click on the small down arrow to reveal the drop down list and then make the selection.

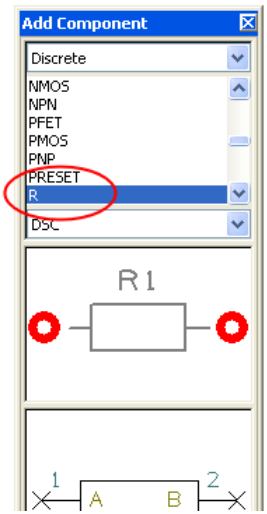


The **Component** list will be populated with components. Selecting this library will automatically select the first component in the list as well.



Scroll down the list of components using the vertical scroll bar. The first in the list is the generic '**C**' component. You can also click in this list and type the first and subsequent letters and numbers of the item required.

Scroll down and select the component **R**. This is a generic resistor that we will change the Value for later on. You can also work with specific named parts if you prefer, both methods are acceptable in PCB Artist.



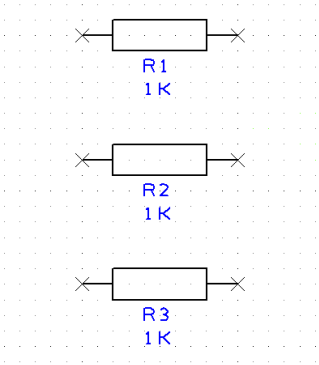
A preview of both the Schematic and PCB symbols will be displayed (the PCB symbol is top most in the preview).

Once you have the correct component, you can add it by selecting the symbol preview and dragging it into the design. You can also add the component by selecting and dragging its name.

This is a one-shot process. If you wish to add more of the same type, drag another from the bin.

The component is currently on the end of your cursor. To place it in the design, click the left mouse button once.

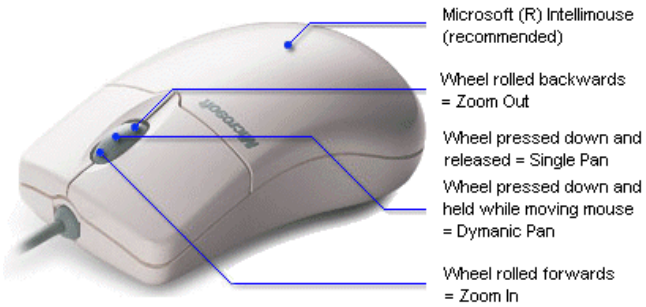
You need 3 resistors in total for the final circuit. Once the first one has been placed, pick and drag 2 more from the Component Browser window. Position then in a stack like the picture below:



▶ To pan & zoom in the design

To view the resistors closer, roll the mouse wheel button forward to zoom in. If you zoom in too fast (and too far), roll the mouse wheel button backwards (slowly). If you zoom too far either way, press <**A>** on the keyboard to **View All** and then zoom in or out again. Shortcut keys are also available for Zoom in <**Z**> and Zoom out <**U**>.

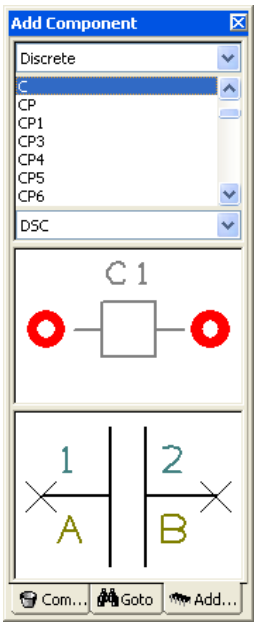
You can zoom while the component is active on the end of your cursor.



Zoom out to give yourself space around the resistors in which to place more components.

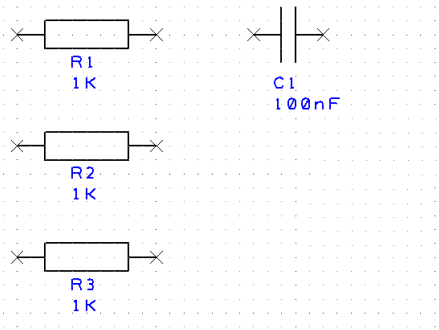
▶ To add the capacitors

We need 4 capacitors for this circuit. From the **Add Component browser** option again type **C** in the component list. The component name will jump back to the **C** component in the list.



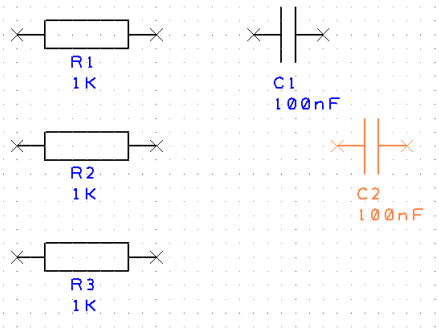
The preview windows confirm this is the correct component. As with the resistors, this is a generic component and we will change its value later on.

As before, dragging a component from the browser will add another to the design. This time only drag out one capacitor to add **C1** and then pause. The design will look like this:



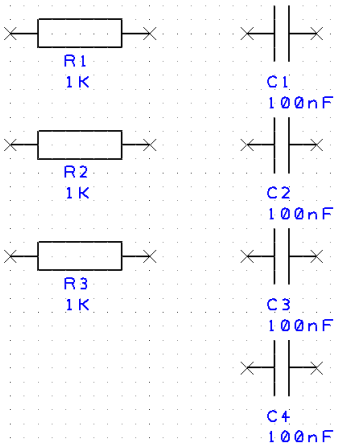
We will use a different technique to add more capacitors.

Select the capacitor **C1** in the design. Press the **Ctrl-C** keys on your keyboard at the same time to *Copy* the capacitor. Now press **Ctrl-V** to *Paste* a copy of the capacitor into the design. You will see that an identical copy of C1 has been made and is named **C2**. These are standard Windows commands.



Move C2 into position and just underneath **C1** by moving the mouse. Single click to release. Now use **Ctrl-V** again, to add another capacitor **C3** and then again to add **C4**.

Your design now looks like this:



Obviously in a design as simple as this, it would have been quicker to drag out 4 capacitors but the Copy/Paste principle is worth demonstrating.

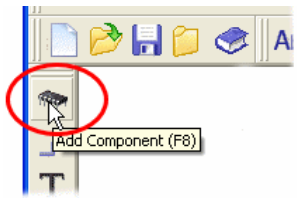
▶ To switch off the displayed grid

At this stage we don't really need to see the displayed grid dots so let's switch them off. Click <**G**> to toggle the grid off. Press this key again will switch it back on at any time.

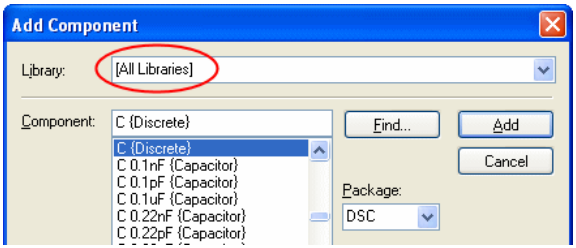
▶ To add the Op-Amp

Our design requires an **Op-Amp**. We will use another technique to add this component.

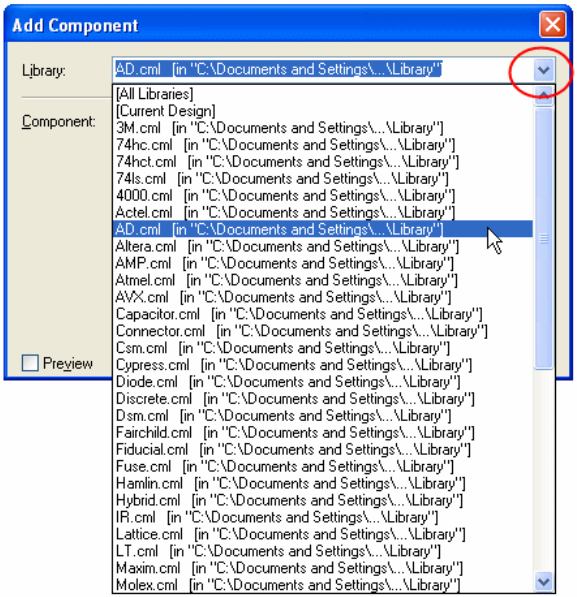
Click the **Add Component** button on the toolbar, shortcut key <**F8**>.



The Add Component dialog will display [All Libraries] in the library path.

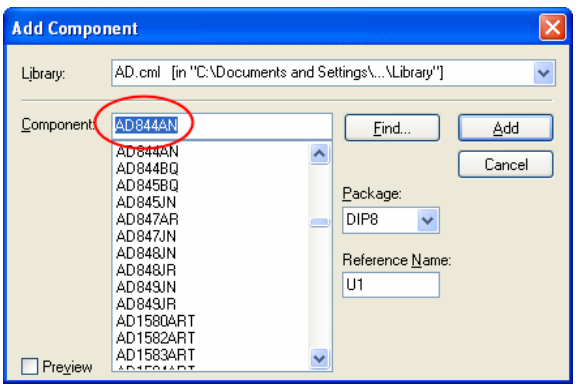


We need to swap libraries to use a different one. Click the small 'down' arrow on the right side of Libraries: to reveal the fully libraries listing.

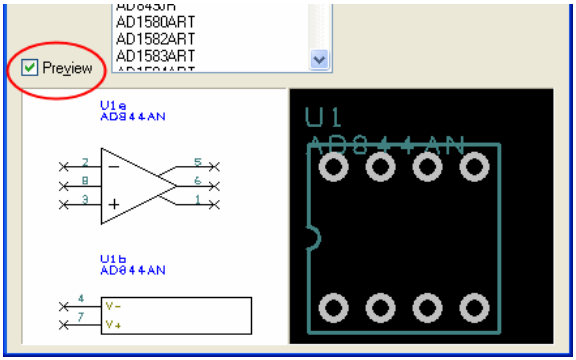


From the list, select the **AD.cml** library.

In the **Component:** list, type in **AD844**, this will select the name **AD844AN** on the list.



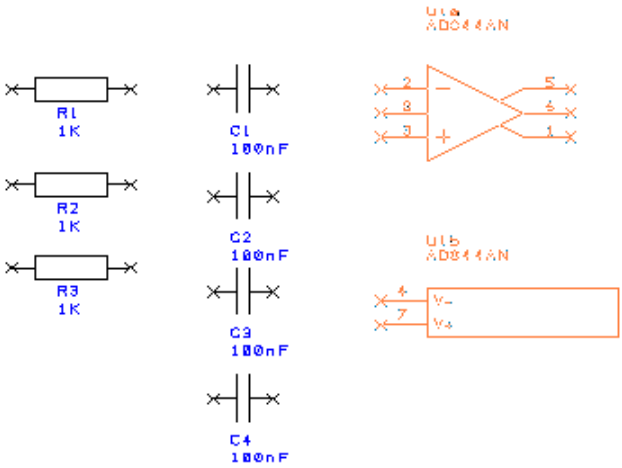
Click the **Preview** button to display previews of the Schematic symbol and PCB footprint.



Click the **Add** button to add an opamp to the design.

Depending on your current zoom level, you may need to roll the mouse wheel to zoom out a little. You will need to give yourself some space on the design in which to place this new component.

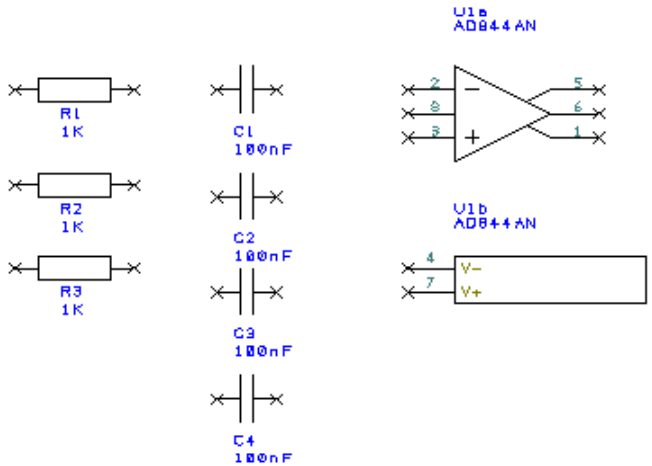
Our Op-Amp (U1) is shown in the design on the end of your cursor as two gates (a & b). Gate a is defined as the logic gate and gate b is the power gate for this component.



Click the mouse once to place the first gate. The second gate (power) will now be dynamic on the end of your cursor ready for placing. Place it near gate a, we will move it later on.

Click the **Esc** key to cancel this option.

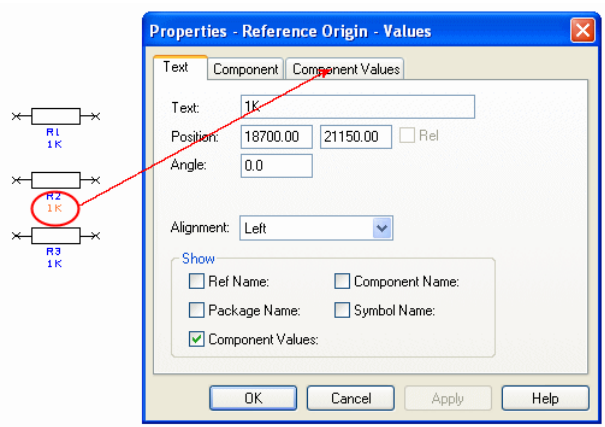
Your design should look something like this:



Before we make the connections we should change the component values to be the ones required. This is easy to do because we've used 'generic' components. If you prefer to use specific Part named components, you can also work this way with PCB Artist.

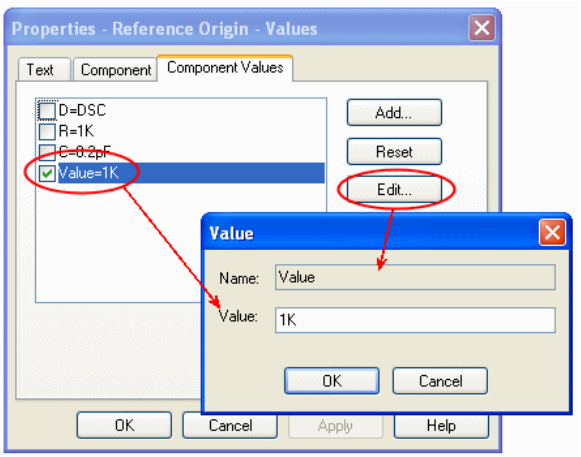
▶ To edit component Values

Double-click on the Value field (**1K**), the **Text** tab of the **Properties** dialog is displayed.



Click on the **Component Values** tab.

The **Value** field shows the current 1K value and is already highlighted. Simply click the **Edit** button or double-click the blue highlighted field to edit the value required.



Type in **11K**.

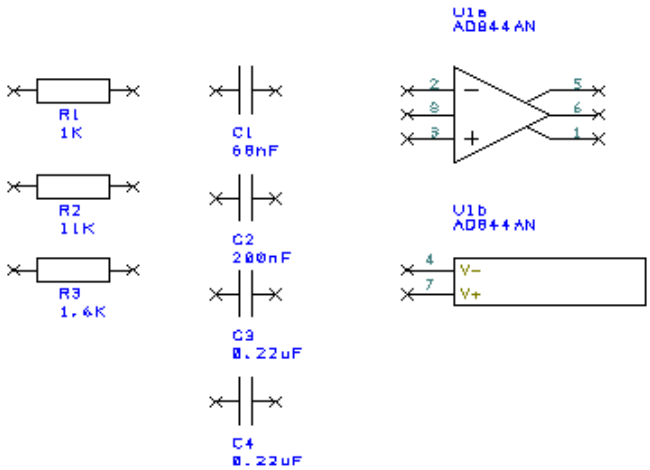
When done, click the **OK** button to exit this dialog and **OK** again to exit **Properties**. When the value of R2 is changed to 11K the design is updated.

Using the same process (double-clicking on the Value name), edit the Value fields for all R's and C's in the design as shown below:

R1 1K **R2** 11K **R3** 1.6K

C1 68nF **C2** 200nF **C3** 0.22uF **C4** 0.22uF

Our design now looks like this:



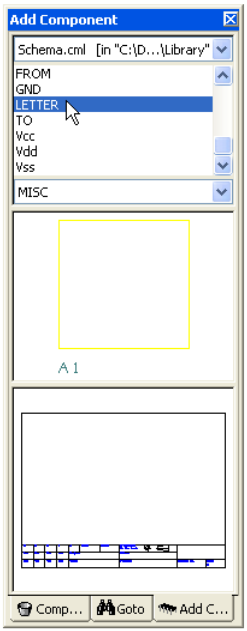
If you select the whole component by mistake, you can still edit the Value field by clicking **Component Values** tab on the **Properties** dialog.

▶ To add a drawing border to the design

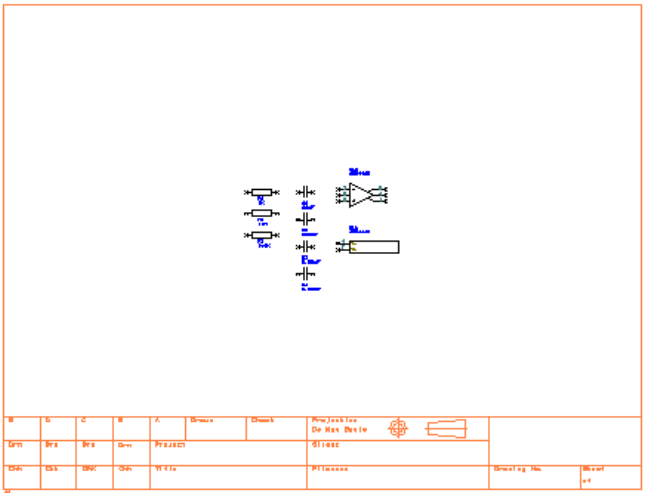
At any time you can add a drawing border to the design. We will add one now.

Using the **Add Component Browser** again (<**F**9>), select the **Schema** library from the drop down list. For reference, this is a schematic only component library and will not have any PCB symbols associated with them.

Scroll down the list and select the **Letter** drawing border. This has been previously created to fit a Letter page size.



Like other components, pick **Letter** from the list or from the preview. Drag it into the design and position it on top of the existing components. You may need to zoom out a bit to see it. Use the <**Z**> key to zoom out or <**A**> to view all of the design. For speed you can also roll the mouse wheel.



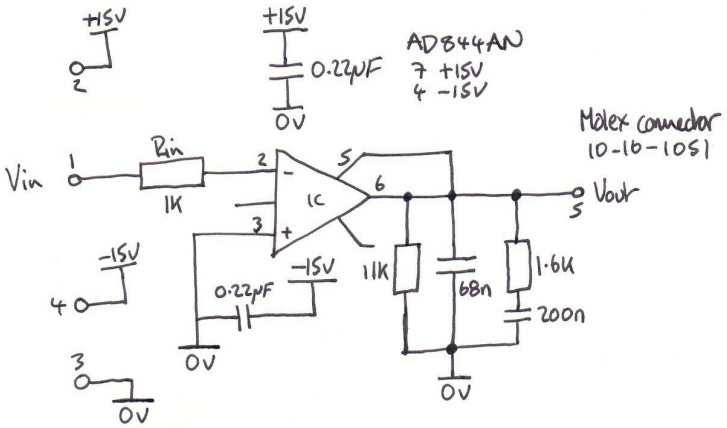
As with other components, this can be moved later if you need to.

▶ To save the design

At this stage it might be a good idea to save your design. Go to the **File** menu and select **Save As**. Save it to the Examples folder and call it something sensible, like **Tutorial.sch.**

Placing Components

The position of the components currently is as they were placed after adding them to the design. We need to place them around the Op-Amp so they can be used in our circuit. On a real design, this would be how you would work in practice. To remind you, the original 'drawn' circuit looks like this:

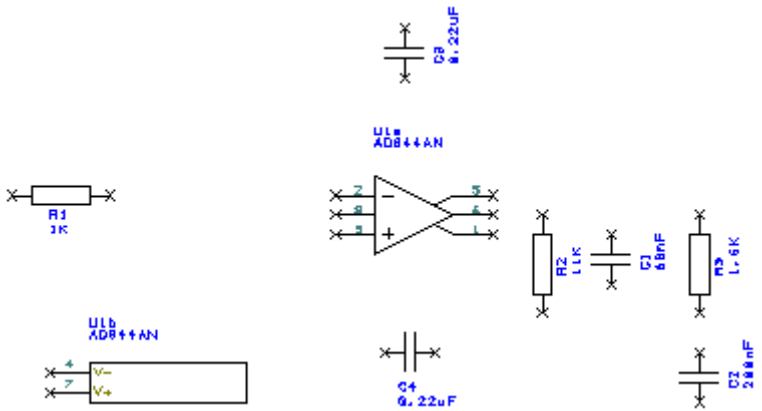


As before, move the components into position by **picking** and **dragging** them in one move using the mouse.

Components can be rotated during move by clicking the <**R>** key. A single click of the R key will rotate the component by 90 degrees. Click R more than once to rotate it by another 90 degrees and so on.

Note: You can also change the placement to fine tune it after adding the connections as well.

The design will now look like this:



You are now ready to add connections.

Adding Connections

Connections are added to the Schematic design to make connectivity between electrical pins. When translated to the PCB design, these connections give you your net list.

You can add connections in a number of ways:

- Using the **Add Connection** option from the **Add** menu or from the **Schematic** Toolbar, click on the pin to start adding a connection, or;
- Double-clicking on a component pin to start a new connection, or;
- 'Dragging' off an unconnected component pin to start a new connection.

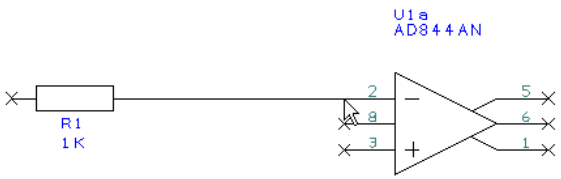
For this tutorial we will use the 'dragging off pins' method as this is the easiest to start with.

▶ To add connections by dragging off pins

To aid the addition of connections to a schematic, you can simply 'drag' off an electrical pin, each click of the mouse will then add a corner. Moving over another electrical pin will allow you to finish the connection.

Using the design that you have created so far, zoom in on the area around R1 like the picture below.

Click and drag (keeping the mouse button pressed down) off pin 2 of R1, this will start a new connection. Move the mouse over U1a pin 2 and single click to finish.



You do not have to be exactly over a pin to finish a connection; as long as the cursor is within range then it will snap onto the terminal.

Click the mouse button once to finish. You'll notice that the component pin terminals X disappear when you connect to it; this indicates that it is connected.

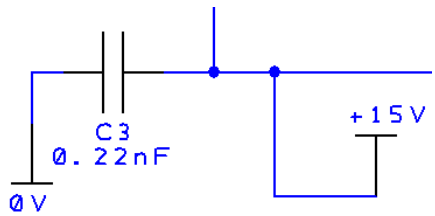
▶ To join connections together

There are a couple of connections which must be connected together. Let's assume that you've made a connection and now you need to connect another to it using a junction dot. When you attempt to finish a connection on an existing connection, when you release the mouse, the connection will automatically be added (as if it were an electrical terminal) and a junction dot displayed to indicate it is connected.

Following our sketched design above, complete the design adding the connections required.

Adding Power & Ground Symbols

Power and ground symbols are used in the Schematic to indicate a connection to a power source. These symbols are required for aesthetic purposes and are not translated to the PCB (because they do not have PCB footprints associated with them). The signals they represent are connected together at the translate stage by 'implied' connectivity. The power & ground symbols themselves can contain inherent net name properties that are automatically used when the symbol is attached to the net.

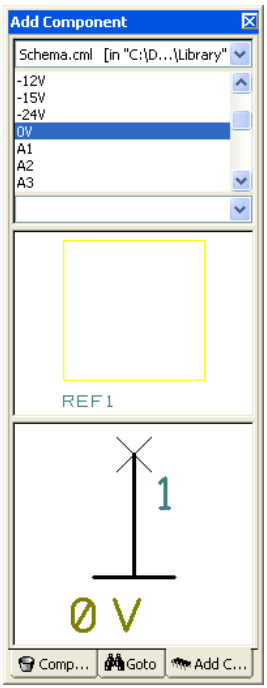


▶ To add Power symbols

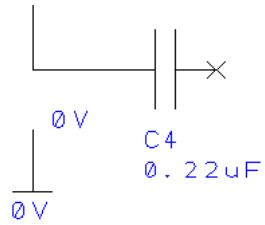
Using the **Add Component** browser on the **Interaction bar** <**F9**>.

The **Schema** library will still be selected after using it previously when adding the Letter component. If for some reason it isn't, select it now from the library list.

In the Component list select **0V**. You will see the symbol selected in the preview box. Remember, all of these symbols have no PCB Symbol associated with them, they are Schematic only symbols.



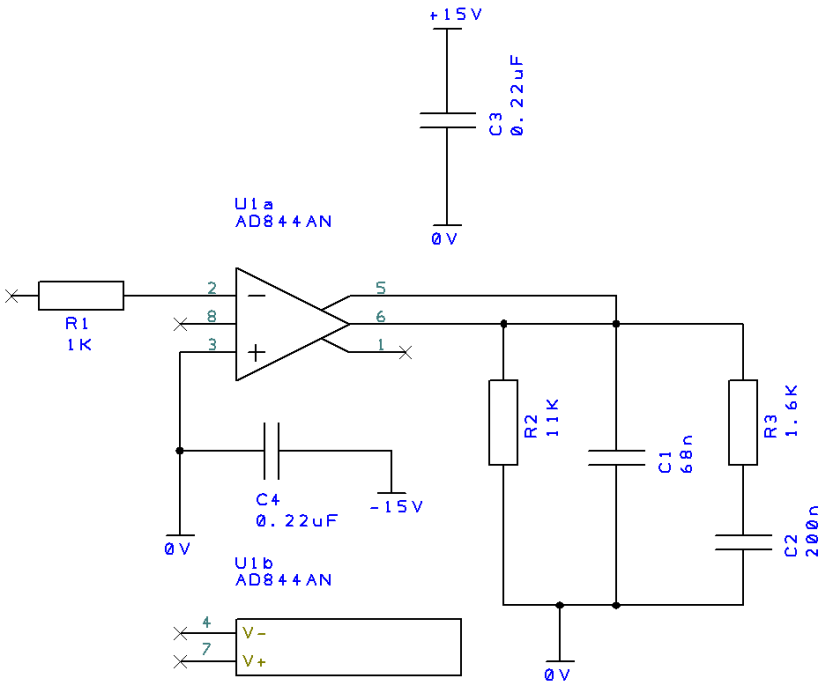
Drag the **0V** symbol into the design from the browser. Move it near to **C4** and click the mouse to release it.



Using our working design, add 2 more **0V** symbols. From the **Add Component** browser, this time add one **+15V** symbol and place near **C3**. Now add one **-15V** symbol, place it near **C4**.

Drag a connection off the end of the symbol terminal and connect as shown below.

Your design will look something like this:



Generally speaking you would now need to add some connector pins to connect the circuit to the outside world.

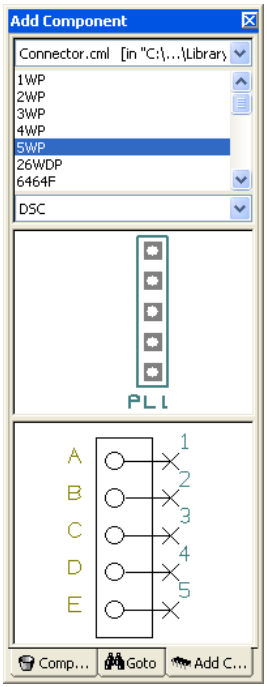
Adding Connectors

For this example, we will use a connector component from the library.

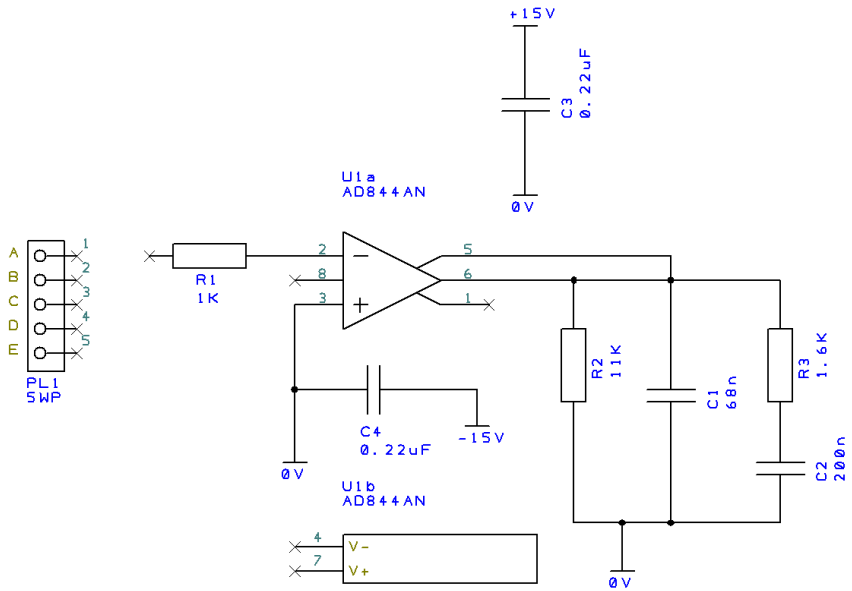
▶ To add Connector pins

Using the **Add Component** browser from the **Interaction bar** <**F9**>. Choose a new library, select the **Conn_std** component library.

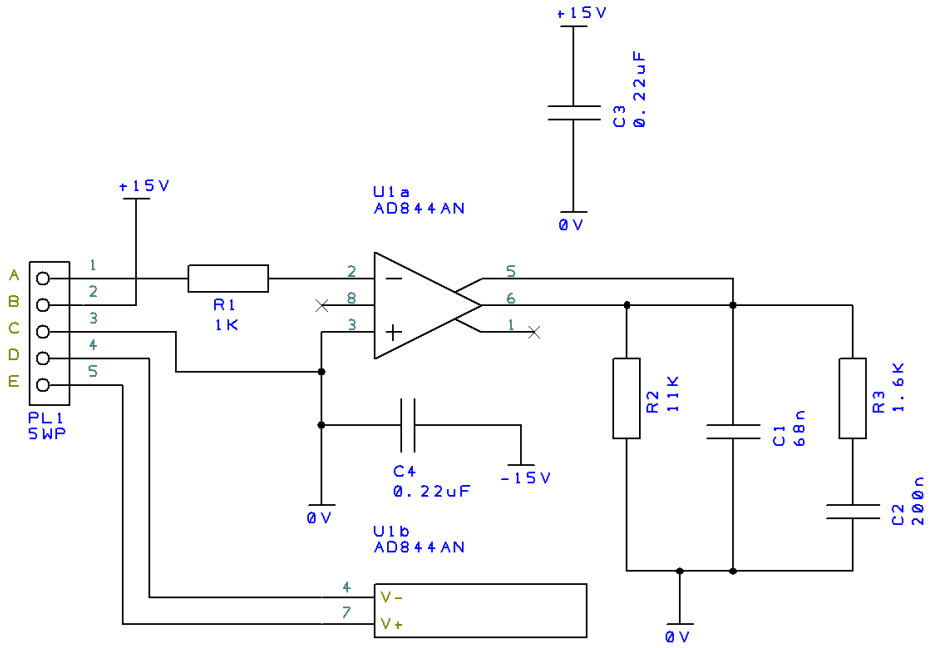
Select the **5WP** connector component from the list. The PCB Symbol and Schematic Symbol previews will be displayed.



Drag it into the design and place it near the resistor **R1** as shown below:



Connect in the connector to the design using **Add Connection** or by **dragging** off a pin. You will also need to add another +15 component for completeness. Your design should look like this below:



Click the **Save** button on the **File** menu to save the design.

Naming Connections

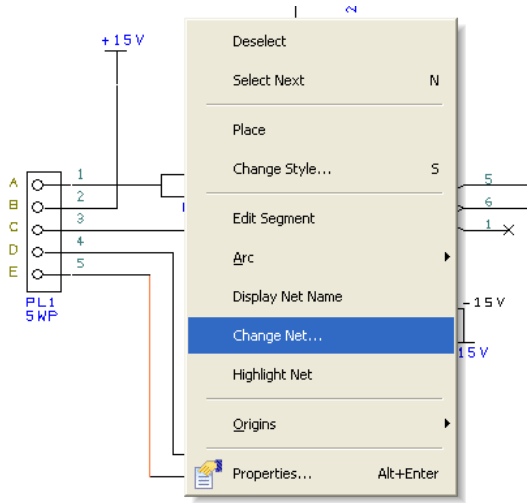
One last task that is often required is that of naming connections in the design. We've added 'normal' connections which have been named using the default naming convention and have been given 'system' names like N0005, N0100, N008 etc. We've also added connections between pins and components that have pre-defined net names (power and ground components, +5V, 0V etc.).

We now need to name a few connections to make them more specific for use in the PCB design.

▶ To edit net names

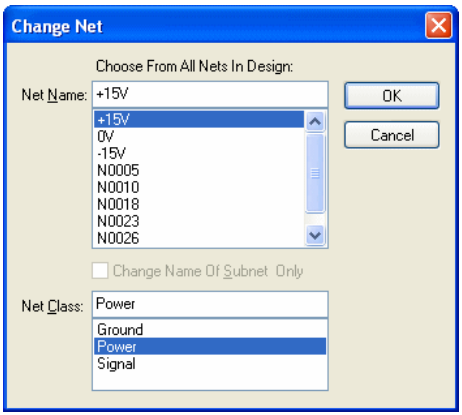
Select the connection just off pin 7 of the Op-Amp's power gate, the gate marked as U1b.

Right click and from the shortcut menu, select **Change Net...**



From the dialog now displayed, select the **Net Name: +15V** from the list.

From the **Net Class:** list, select **Power**. This will pre-assign you a set of connection properties for this net when it gets used in the PCB portion of the design.



Click **OK** and return to the design.

If you wish to see the Net Name in the design, right click again with the connection selected and choose **Display Net Name** from the menu. The net name will now be displayed in the design, you may reposition it if required by picking and dragging it.

If this is difficult to select because you accidentally pick the wrong item, press the <**N**> key to select Next. The next item in the picking tolerance will be picked.

Now select pin 4 on U1b. Using **Change Net** again, this time select the net name **-15V**. Select the Power **Net Class** again. Click **OK** to exit this dialog. The Net Name can be displayed using the method above if required.

Your Schematic is Finished

The schematic design is complete. You may wish to print it to check it, use the **Print** option from the **File** menu.

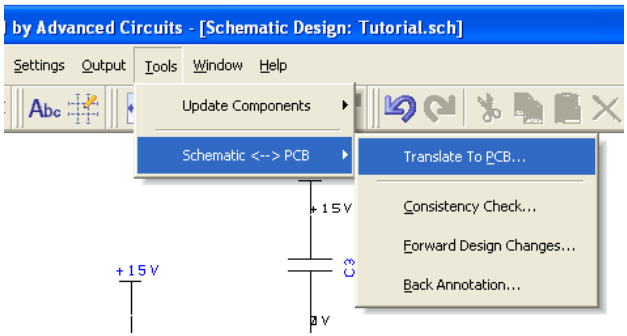
The next stage is to convert the design into the PCB design environment. To do this, move onto the next chapter *Chapter 3. - Converting Your Schematic To PCB*

Chapter 3. Converting Your Schematic To PCB

Converting the Schematic to PCB

If you created a **Schematic** in PCB Artist, the final process before starting on the PCB design itself is to translate the Schematic design into the PCB design editor.

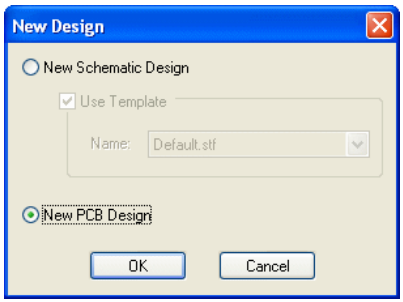
When you are in a position where you wish to go-ahead with this translation you should select **Schematic <- - > PCB >** and use the **Translate To PCB** option from the **Tools** menu.



This will display the **New Pcb** wizard.

Starting a New PCB Design

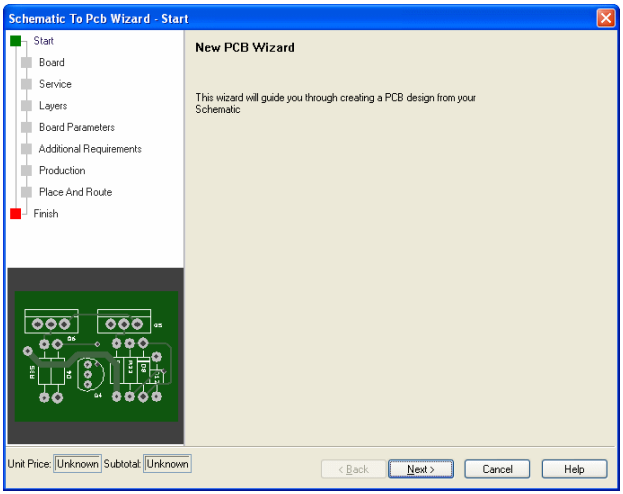
If you intend starting a new PCB design without using a Schematic design first (which you are quite able to do), from the **File** menu select **New** and check the **New PCB Design** radio button. Click the **OK** button and this will start the **New PCB** Wizard.



The New PCB Wizard

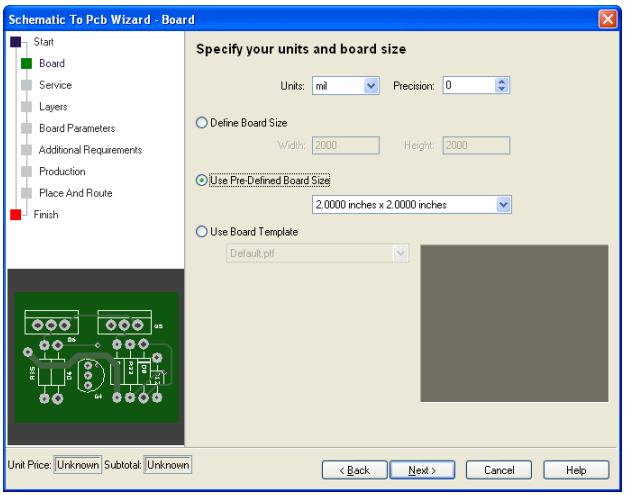
This wizard is a set of dialogs which you step through in sequence to define aspects of your design in the PCB design editor. The online help is available for each page by pressing the <**F1**> key at any time.

Click **Next >** to move onto the next page. You can also click the page name you require in the list at the top left hand side of the page but is recommended to work through the wizard pages in sequence.



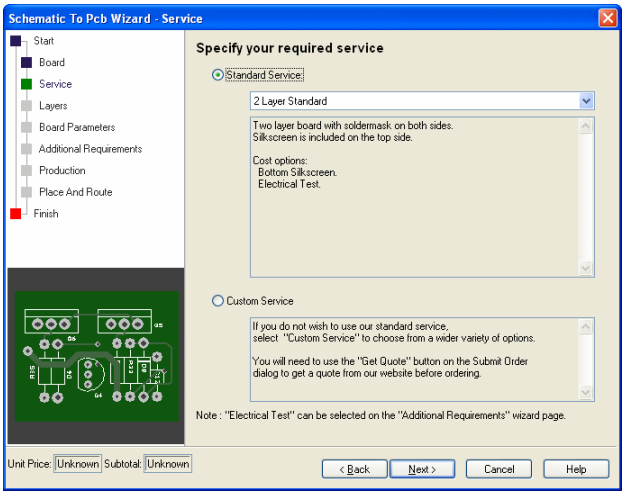
The **Board** page allows you to define the Units used in the design and the board size. If your manufacturer has supplied some template designs with commonly used board outlines, they will be presented in the Use Board Template list for you to choose from.

For the tutorial, select the **Use Pre-Defined Board Size** radio button and select **2.000 inches x 2.000 inches** from the list. Click **Next >**



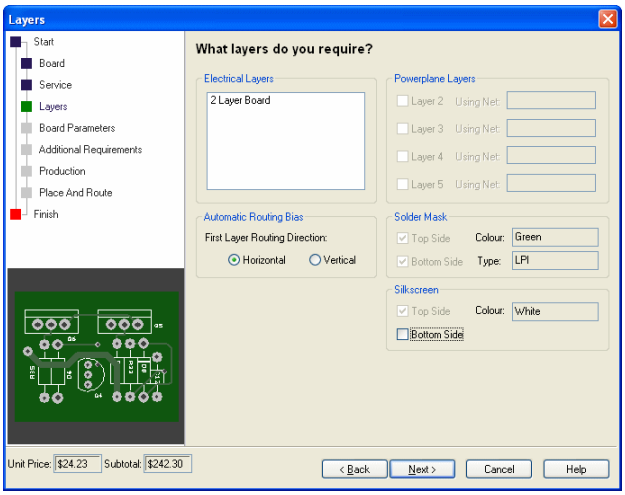
The **Service** page is used to select a named service. This is used to predefine 'standard' services that we provide or to customize the service required.

Using a **Standard Service** will enable you to place an order directly using the **Submit Order** option. If defining a **Custom** service, you will have to get a quote from the **Submit Order** dialog before proceeding with the order.

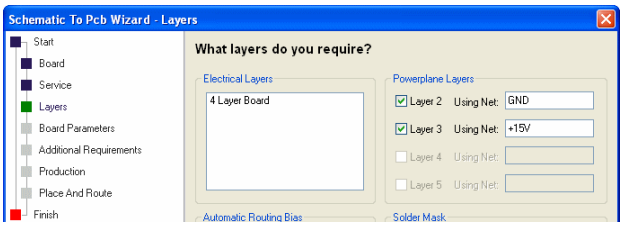


The **Layers** page is used to define how many layers the design contains and how they look when manufactured. Remember that the number of layers used will also be a factor when the cost of the board is calculated.

Leave the setting of **2 Layer Board** for **Electrical Layers** and all other settings for now.



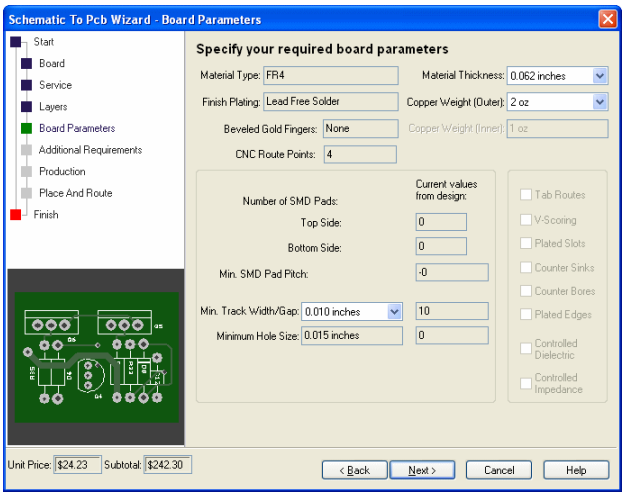
If you had selected a greater number of layers than 2, 4 or 6 for example, you would need to also select how the inner layers would be used. If you wish to use the inner layers as power planes, check the Powerplane Layers check boxes and enter the net name associated with that layer. For the example below, we have selected both Layers 2 and 3 and have assigned them plane net names of GND and +15V respectively.



Click the **Next >** button to continue.

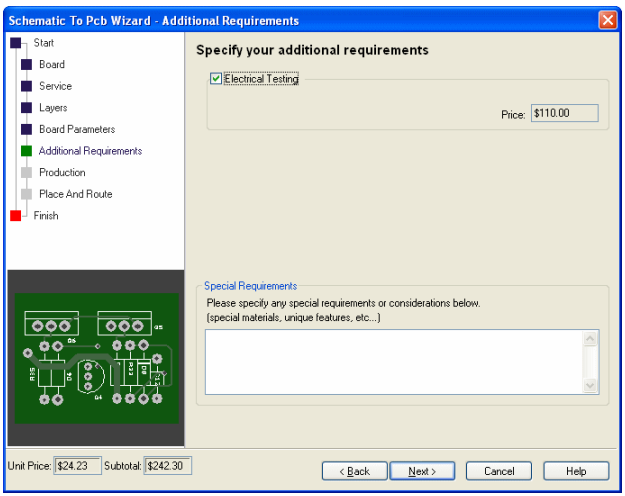
The **Board Parameters** page is used to define the material used to manufacture the final PCB.

Use this as it is for now. Click the **Next >** button.

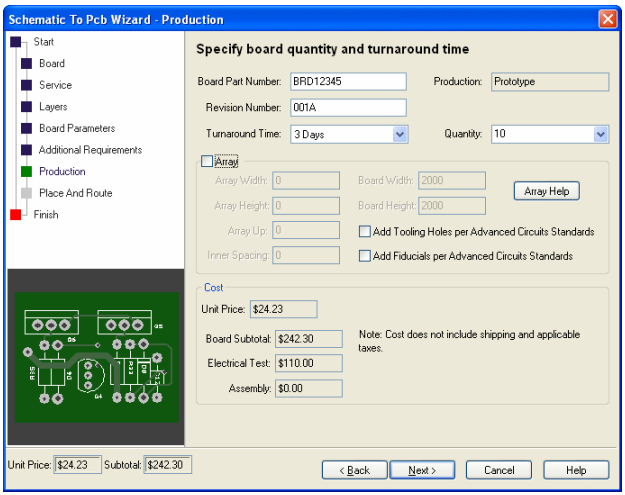


Additional Requirements is used to tell the manufacturer about any special requirements using the notes window into which you can type your message.

You can specify **Electrical Testing** and **Component Assembly** as 'cost' options.

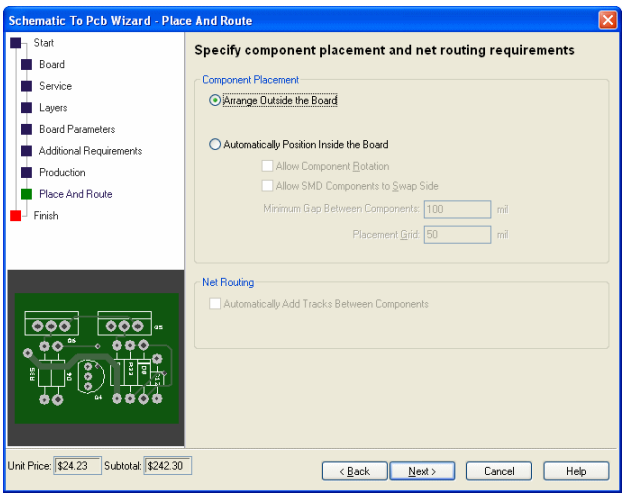


The **Production** page is used to specify how fast you require the board and the production quality and quantity. The cost of the PCBs, if available, is shown in the cost box.



The **Place And Route** page is shown if you are transferring a schematic. It allows you to tell the PCB editor what to do with the parts and nets once in the PCB design. You can opt to place components and route them automatically, or just leave the placement and routing for you to do manually.

For the tutorial we will change the settings to allow you to do the placement and routing manually once in the PCB design editor. Select the **Arrange Around the Board** button.

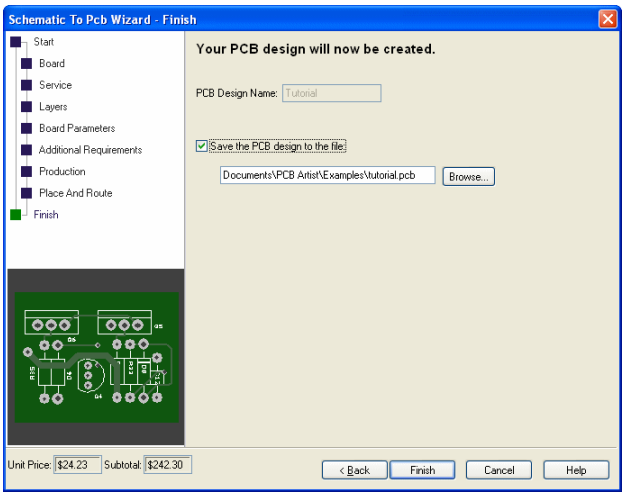


Click the **Next >** button.

The **Finish** page allows you to give your PCB design a name and to save it to a file in a chosen location.

Click the **Save the PCB Design To File** check box and type in a suitable name. You should give it the same name as the schematic, but with the .pcb file extension **Tutorial.pcb**

Click the **Finish** button.



At any time you can go back to one of the previous pages by selecting the page name form the list displayed.

When you click the **Finish** button, the PCB design editor will run up and your PCB design will be ready for use.

Schematic Design Completed

You have now completed the first section of this introductory tutorial so let's move on. Leave everything as it is and read on if you wish to continue with the PCB design tutorial.

Chapter 4. PCB Design Editor

Getting Started with the Design

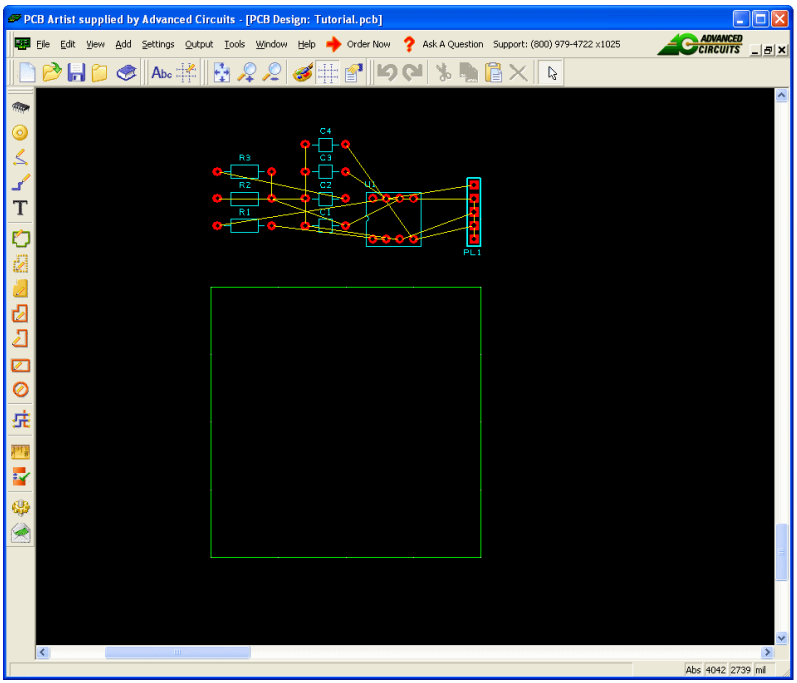
PCB Artist PCB is flexible in that you can start your layout in two ways:

- Use PCB Artist Schematic Capture as the 'front-end' to the PCB system. The nets and components are converted to PCB in the integrated environment.
- Use the PCB Artist PCB design editor interactively creating the design on-the-fly without the need for an initial netlist. You just add components and connections as required to create the PCB design.

Converting the Schematic to PCB

If you are following the tutorial, you would have already processed your 'logical' Schematic design into the 'physical' PCB design layout editor using the **Schematic To PCB** wizard. If you didn't do this, go back to the previous chapter (*Chapter 3 - Converting your Schematic to PCB*) and convert your schematic into a PCB design.

At this point you are ready to start laying out the PCB design. You will see the board outline ready to place the components in. We previously chose to leave the components unplaced and the nets unrouted. These two features can both be selected during the final conversion process.



The board outline is displayed with the components stacked neatly at the top edge ready for placing.

Changing the Number of Layers

If you decide at any time to change the number of physical layers or the type of manufacturing, you can do this using the **Settings** menu and the **PCB Configuration** option.

This opens a reduced page set of the **New PCB** wizard where you can edit the parameters of the finished PCB. Be aware that reducing the number of layers could lead to items on the inner electrical layers being lost.

Creating a Board Outline Interactively

If you require a custom board size and shape, and decide not you use one of our pre-defined templates, this can also be easily achieved.

Deleting an existing board outline

PCB Artist only requires one board outline so you would need to delete any previous one first. You can edit an existing board outline as well. If you do need to delete it, simply select the outline and click the **Delete** button on your keyboard.

Adding a new board outline

Board outlines can be created easily in PCB Artist; by adding board shapes (including circles). The **Add Board** option on the **Add** menu allows you to interactively create a board outline by drawing it into the design. Three shape styles are provided: **Rectangle**, **Polygon**, and **Circle**. There is also a shortcut on the **PCB Design** toolbar to add a **Board Polygon**.

Board outlines are created as a Closed shape so during addition, you'll see the 'trailing' board segment trail back to the start point. Click to add corners. After adding the last corner, use **double-click** to finish. If during the insert, you wish to cancel, click the **Esc** key on the keyboard. This will create you a board outline as a continuous shape starting and finishing at the start point.

Once the board has been added, it can be modified afterwards, extra segments can be added and the corners can be mitred or made curved.

For this tutorial, we would have specified the board outline from one of our templates available during the Schematic To PCB conversion process earlier. If you require cutouts in your board, add these using the **Add Board Shape** option.

Placing Components

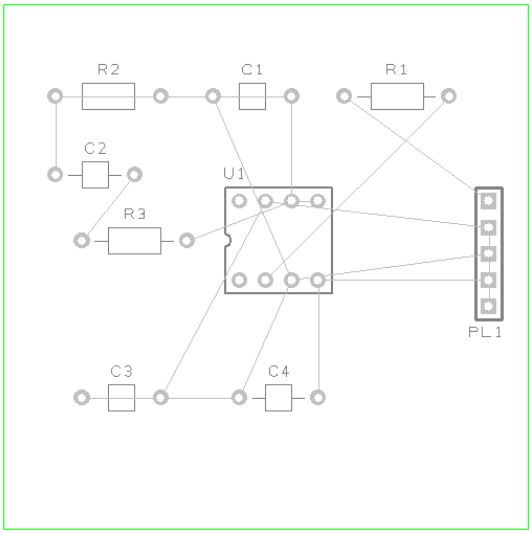
We've started the design with the board outline and are now ready to manually place the components. This can be an automatic process either from the conversion phase or by using the **Auto Place Components** option from the **Tools** menu. For this exercise we will use the manual placement method.

Placing components uses the **Place** option and because PCB Artist uses standard Microsoft Windows methodology throughout, Place is simple case of picking and dragging the selected component.

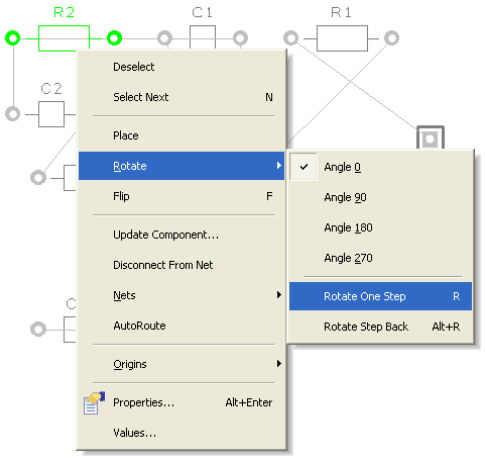
***Note**: during place the drag may be cancelled by pressing the **Esc** key, or once released by using Undo (**Ctrl-Z**).*

Place all the components to look like the picture below (note, we have changed the background color of the design for this document for clarity). It is suggested you place **PL1**

and **U1** first and place the other components around these. Placement on this design isn't critical but improve it if you feel you can or want to.



During Place or after placing components, use the shortcut key <**R>** to Rotate them and <**F>** to Flip (mirror) surface mount components to the other side of the board. These options (and more) are available on the shortcut menu during Place by clicking the right mouse button.



For our design, no mirroring (Flip) will be required. The placement and overall connection lengths could be improved by rotating some of the components. If you feel you would like the practice placing components, select a component and click <**R>** to rotate it. Component names can be rotated back to be a readable direction separately using the same pick and rotate method.

Once fully placed, the design should be routed.

Routing the Design

The design can be routed manually or automatically. This is the process of converting the electrical point-to-point connections into physical copper 'tracks'.

Initially we will route the design automatically to see what the result looks like and to try this option. We will then Unroute is back to the 'rats nest' and use the manual routing tools to route.

Automatic Routing

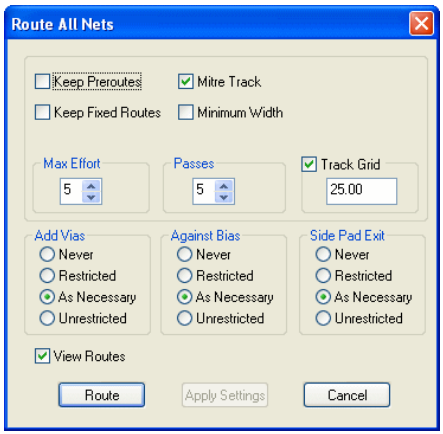
The Autoroute option is used to convert your nets into electrical tracks as you can with manual routing. However, on dense designs it can save you many hours or days of work. The autorouter can take an unrouted design and route it within minutes quite often.

You can use the autorouter in many modes; on **All Nets** in the design, on **Browsed Nets**, **Browsed Net Classes**, on **selected nets** and on **components**.

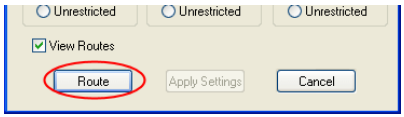
▶ To use the autorouter

From the **Tools** menu under **Auto Route Nets>**, select **All Nets**.

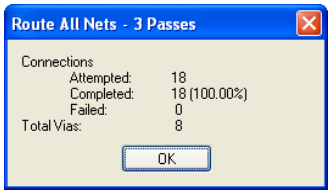
Leave the default settings as they are:



Simply click the **Route** button at the bottom of the dialog to start the router. This small design will easily route 100%.

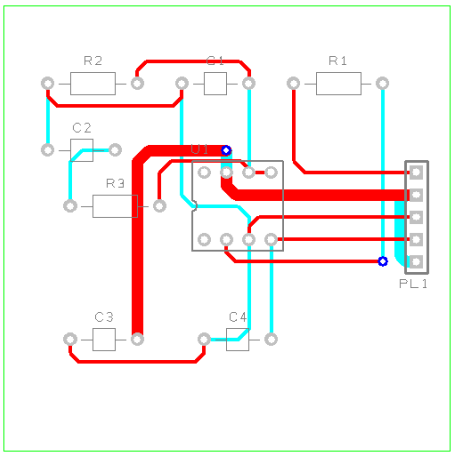


The results are quickly displayed:



Click **OK** to close the report dialog and view the routes.

The routed design might look something like this below. This will vary depending on how you placed the design.



▶ Save the design

The PCB design has already been named when you translated from Schematic to PCB, we named it Tutorial.pcb. We named it this so it matches the name of the Schematic. This will become more important later on when you run the integrity checks before plotting.

For now, click the **Save** icon on the toolbar or **Save** from the **File** menu.

Unrouting the design

You may sometimes have a requirement to unroute all or part of the design. Unrouting is the process of removing tracks leaving only the connections remaining.

The **Unroute Nets>** option is available on the **Tools** menu, with three options available to control the amount of unrouting to perform.

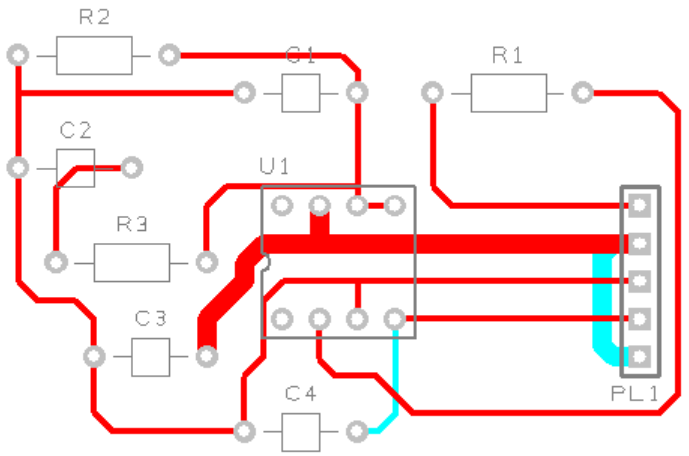
▶ Unroute the design

Use the **Unroute Nets> All Nets** option to unroute the whole design removing all the tracks already done. Don't forget, once unrouted, you can always restore the routes using Undo (**Ctrl-Z**).

Manual Routing

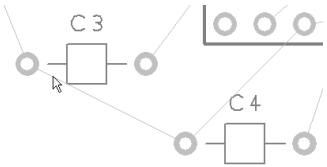
PCB Artist has a set of powerful manual routing tools to aid this process. These tools allow you to swap layers, automatically add vias on layer swaps and edit track thickness.

The routed board might look something like this (the colors are still changed to that in your design for clarity in this tutorial):

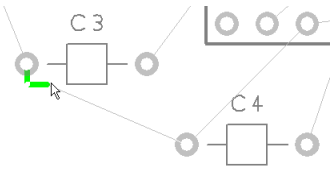


▶ To start routing tracks manually

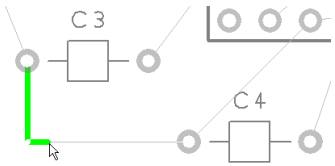
To start adding tracks, **double-click** on a connection, this is the quickest method to start routing, other methods are available.



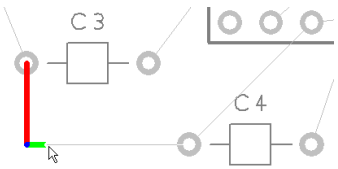
If you followed our design placement loosely, double-click the connection between C3 pin 1 and C4 pin 1.



Move the cursor to the down and right slightly. You'll see the track starting to take shape and a connection between the end of the track and the 'target' pad (C4.1). In this mode, the track is placed orthogonally at 90 degrees with two floating segments.

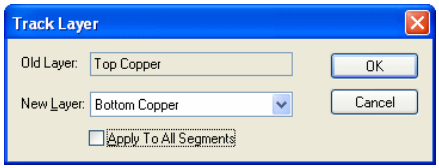


Move the cursor down to draw the vertical track segment. The track is still being placed with 2 segments and a 90 degree corner.

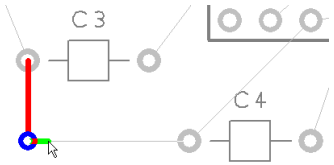


Click the mouse once, this places the first track segment (the straight track segment coming out of C3.1). This is temporarily 'locked' in place for this routing session. It can be moved or edited later on.

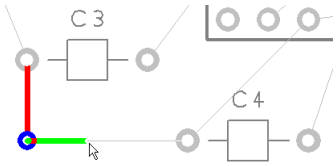
Now we will add a track layer change.



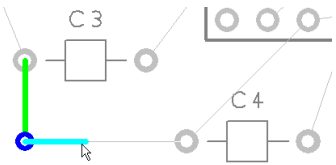
Press the '**L**' key on the keyboard and the **Track Layer** dialog is displayed. During normal routing where a layer change from one side to the other is required, a simple '**L**' followed by <**Enter**> is all that is required. The **Enter** button is the same as clicking **OK** on the dialog. In time you will be able to change layers without looking at the dialog.



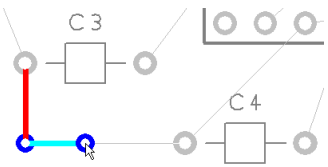
A via has been added automatically and the track layer is now on the Bottom side. This will be displayed on the Status bar at the bottom of the PCB Artist window.



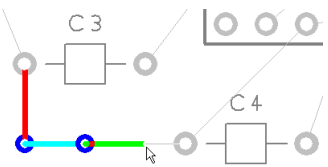
Move the cursor right and click again.



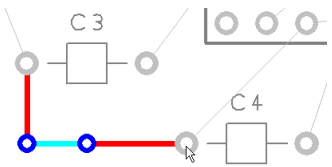
This will either be the next corner or the next location for a via for a layer change. Notice the track has changed color.



Press '**L**' and <**Enter**> again to change layers back to the Top layer. Another via is added automatically.



Continue moving the cursor right towards the target pad C4.1. Notice how it is on the Top layer shown in Red.



Once over the target pad, click once to finish the track editing for this connection.

After the track has been added, it can always be modified afterwards by picking and dragging the track or by double-clicking on the track to edit it.

Move onto the next connection for editing.

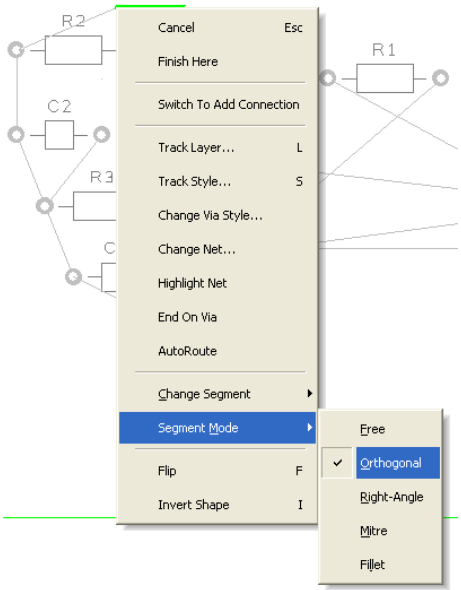
Summary of the basic modes of routing

During track editing, the basic commands for use are:

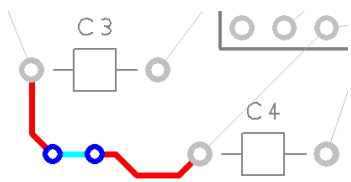
- **Single-Click**, once editing, this will insert a corner to change routing direction.
- **L** followed by <**Enter>** will enable a layer change to the opposite side of the design. This can be used during editing, or afterwards on selection of a track.

▶ To change manual routing modes

You would have been using the standard routing mode of **Orthogonal** routing (two 'floating' segments at 90 degrees to each other).



This mode can be changed by accessing the routing mode during routing. Select the **Segment Mode>** from the shortcut menu and choose the mode from the choice of five available.



On our example design, the routing may now look like this using the **Mitre** mode.

Picking in tight areas

When attempting to select items within a dense area of the design or where multiple connections cross each other for example, it is not always easy to pick the item required at the first attempt. Under these circumstances PCB Artist has a system of picking that 'cycles' through items adjacent to it until you select the item required. Once selected, an action may be made on that item, **Properties** for example.

The shortcut key used for the command **Select Next** is defined as **N** for **Next**.

To try this, click on a pad that has a track attached to it, the pad (or track) is selected. Now click <**N>** and the track (or pad) will be selected. Depending on how close it is to other design items, the component outline and the whole component may also be selected at each click of <N> is made.

Pouring Copper into Areas

Copper Pour is used to flood areas of a PCB layout with copper, usually connected to a nominated signal through 'thermal' connections, typically Ground (GND) or VCC. This facility will then create voids around tracks, component pads, vias and other electrical obstacles.

For this we have the **Add Copper Pour Area** and **Pour Copper** options.

If a copper area is to be inserted without the need for voids around unconnected tracks, components or pads, the **Add Copper** option can be used.

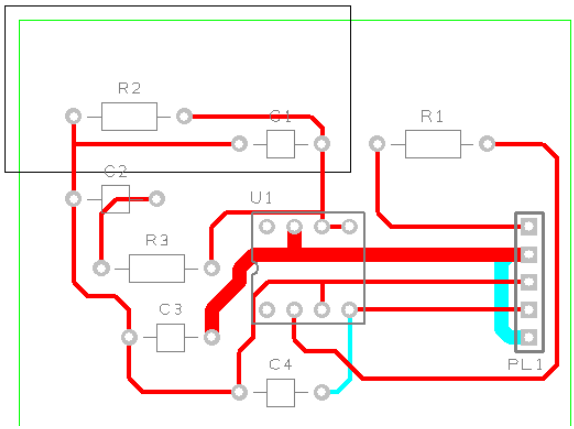
The principle of this facility is to insert a Copper Pour Area and 'pour' the copper into this area.

▶ To create a Copper Pour Area

From the **Add** menu, select **Copper Pour Area> Rectangle** Other Pour Area options are available for adding different shapes. We will add the pour area to the **Top Copper** layer. pour areas and poured copper can be added to any electrical layer. To change layers, select the pour area and click **L** followed by <**Enter**> as we did when editing tracks.

The shape can be created on the design and the pour area can be drawn protruding outside the board outline. PCB Artist will pour the copper using the Spacing rules and rules defined for the Pour area.

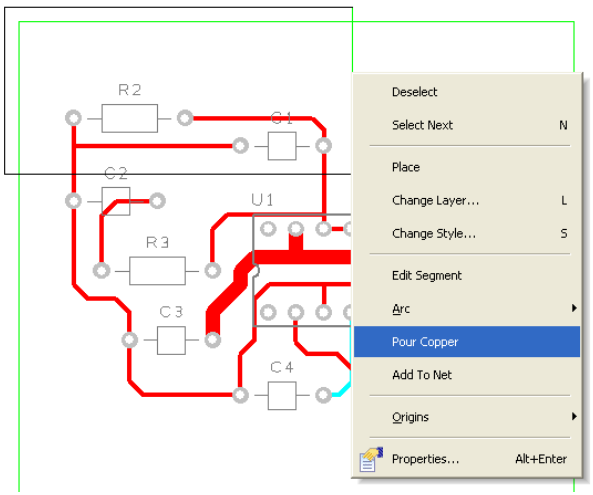
Following the picture below of our example, draw in a basic rectangle pour area shape, this is shown in black in the top left hand side of the design:



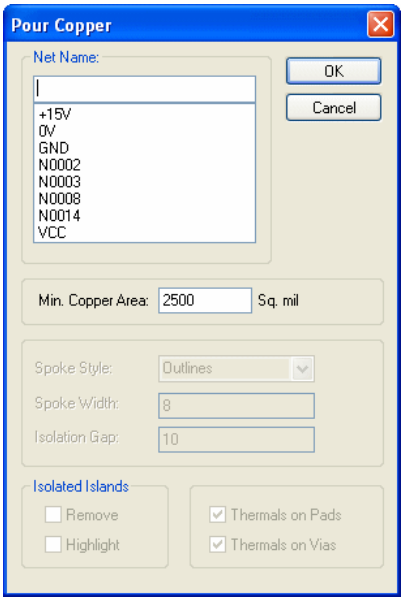
For this example you will see how easy it is to add a poured copper area.

Right click on the pour area.

From the shortcut menu select **Pour Copper**.

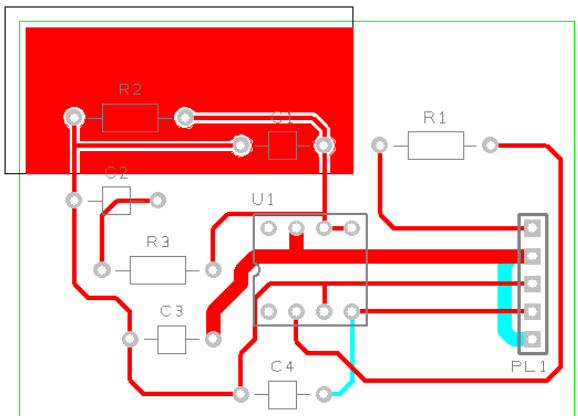


A dialog is displayed from where you can select parameters for the poured area, such as the net name association and thermal connections etc.



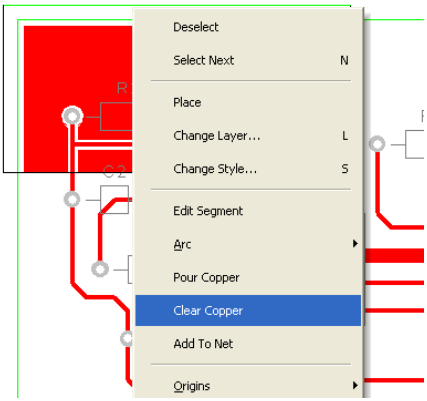
For this exercise, leave the **Net Name** blank and simply click **OK**.

Notice that the poured copper has obeyed the **Spacing** rules of all defined items to copper including Board to Shape rules.



▶ To remove poured copper from a copper pour area

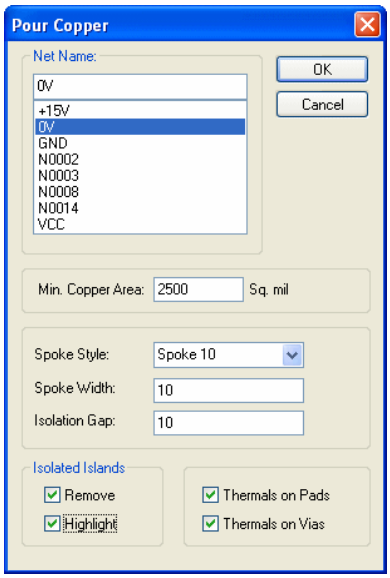
To remove a poured copper area (including any thermal spokes), simply select the area outline and select **Clear Copper** from the shortcut menu. This removes the copper but not the Copper Pour area.

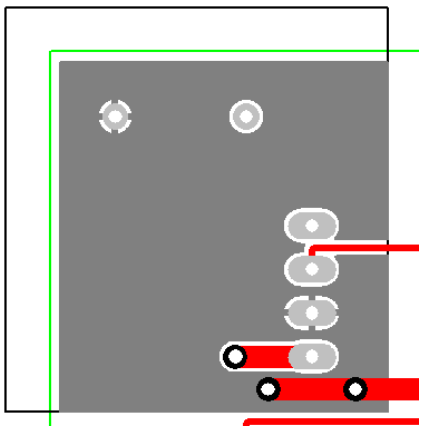


When using poured copper remember you shouldn't use Delete but should always use Clear Copper. Copper Pour Areas can be modified at any time and the area re-poured using the option from the shortcut menu. If you wish to completely remove the copper and area, you must use Clear Copper first and then delete the area afterwards using the Delete key.

▶ To make the copper Pour Area intelligent

There are more facets to the poured copper than we have discussed above. The poured copper can also have a net name associated with it so that when poured, pads are connected via thermal spokes to the copper itself. To do this, select the pour area and from the shortcut menu select **Add To Net**. Select the net name required from the list and then re-pour the copper. Try it again using 0V as the net name and click **OK**.





Using the **Thermal Pads** check box and the **Spokes** settings, the style of how the copper is connected to pads can be defined.

Copper Pour Areas will not appear on your manufactured boards, they provide the system with an intelligent area into which the copper can be poured.

Checking the Design Integrity

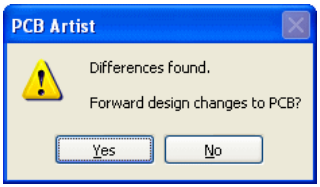
Various checks are available in both the Schematic and PCB editors on the **Tools** menu. This is used to compare and verify designs and make changes in the PCB so that they are both in-sync with each other at all times.

▶ To check the designs

You can use the **Consistency Check** option from the **Schematic < -- > PCB >** menu on the **Tools** menu at any time. By default, the file used for the comparison will be the same name as that used for the Schematic (or PCB design if running from the Schematic editor).

When run, if there are no differences, the following dialog is displayed:

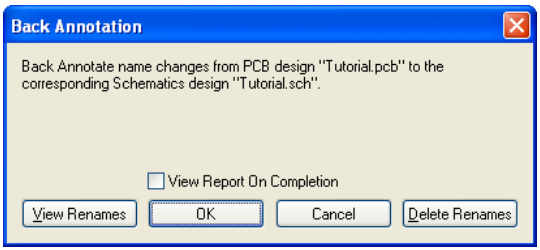
If differences are found, Notepad will be run to view a detailed report. The report will give you a list of changes that would have to be made to the PCB in order to make it match the schematic (the schematic always being the 'master'). You can then perform these changes automatically by selecting the **Yes** button on the dialog.



Run this option on our tutorial design, there should be no differences. If there are, click the **Yes** button and make the changes.

▶ To back annotate name changes to the Schematic

Back annotation changes can be made in the PCB, and these can be passed back to the schematic also using the **Back Annotation** option from the **Tools** menu. When run, it will report and allow you to perform any back annotation changes to the Schematic that are 'pending'.



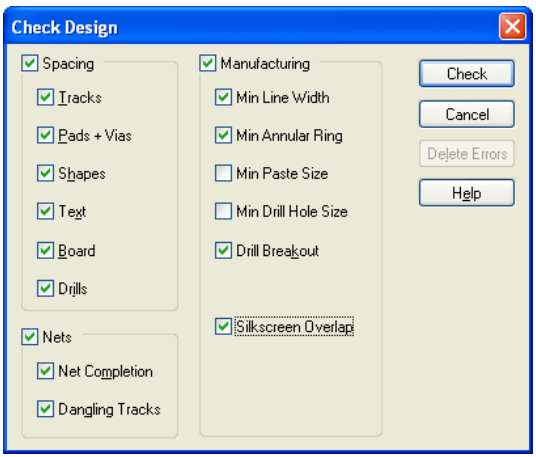
Design Rules Checking

Before sending your design for manufacture, you must run the **Design Rules Check** option. This dialog contains check boxes that enable various checks to be made. The checks are made against the **Spacing** rules defined. This is also used to check various manufacturing rules have been obeyed.

When errors are found Error Marker letters are added to the design on the appropriate layer to help you find and correct the actual error.

▶ To run a Design Rules Check

Select the **Design Rule Check** option from the **toolbar**.

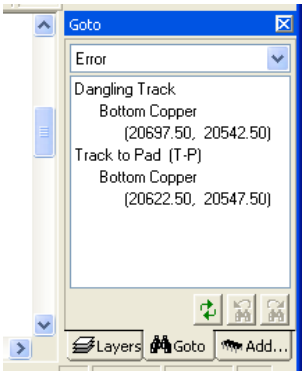


A dialog is displayed that allows you to control which items are included in the check. Click **Check** to run the DRC.

▶ To locate DRC error markers

After running a design rules check, you can find the error markers in the design using the **Goto** browser. This is available on the **Interaction bar** by pressing the **F9** key.

When the **Goto** browser is displayed, from the list of available Goto options, drop the list down and Select **Error**.

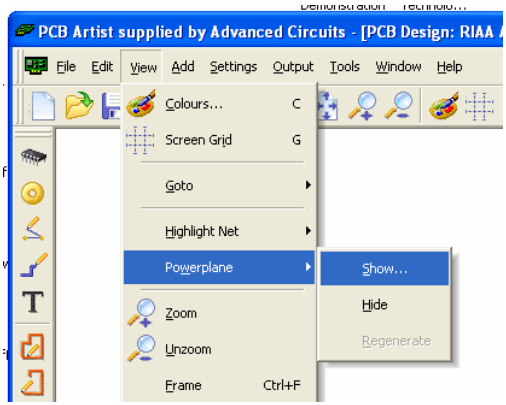


This list is used to view all the design error makers as a sorted list. It is displayed in an 'active' browser, click to select an error, you can locate the error maker in the design. If the error marker is off the screen, it will be brought into the centre of the display so it can be seen.

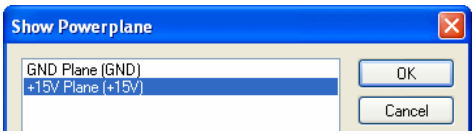
Viewing Power Plane Layers

If you have included powerplane layers in your design on the inner layers, before you send off the PCB design for manufacture, it is worth running a visual check of the thermal connections. This visual check will validate any potentially missing thermal spokes where a pad to copper connection has not been made. Where a pad to copper cannot be made because of design restrictions, the thermal spoke will not be added. Your check will identify these missing connections.

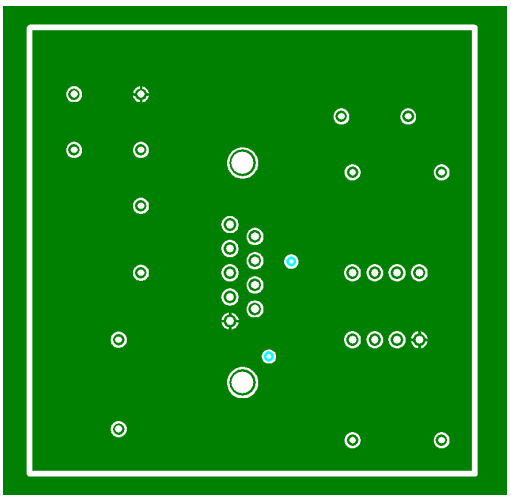
To do this, select the **Powerplane >** option from the **View** menu. Select **Show**…



From the list of powerplane layers, select the layer required.



If we select the +15V layer for our example, the plane will look like this:



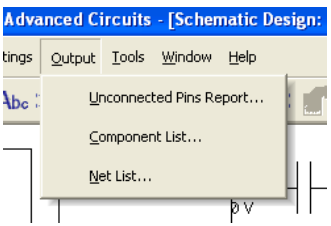
Once finished, select the **View** menu and use **Powerplane > Hide** to switch it off.

Reports, Part Lists and Net Lists

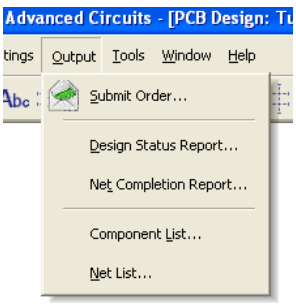
PCB Artist can generate reports where all vital aspects of the design can be output in detail. The reports also include a set of 'standard' reports like Component lists and net lists.

These reports are all available on the **Output** menu.

In Schematic editor:



In PCB design editor:

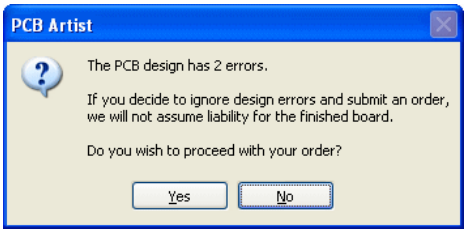


During your testing of the options, run the reports to see what they can produce for you. Each report will be displayed using Notepad, close the report once viewed.

Manufacturing the PCB

PCB Artist provides you with an output mechanism for sending your order and design to Advanced Circuits to be plotted and manufactured.

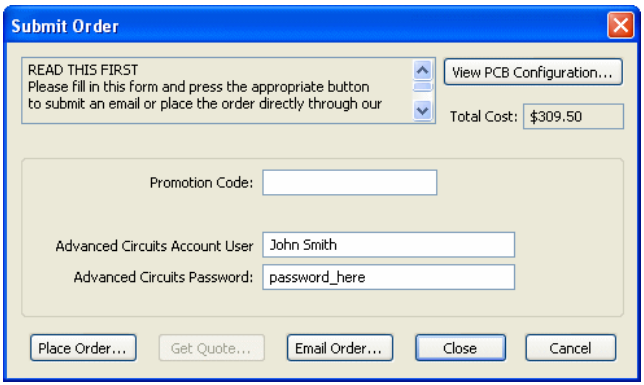
Using the **Submit Order** from the **Output** option, the design is checked. If DRC errors exist in the design, a dialog is displayed:



You should check the design and ensure that all known errors are corrected.

The **Submit Order** dialog is displayed. Fill in the order information. The asterisk at the end of a field means the information must be provided.

Use View PCB Configuration to get a report of the manufacture options you have selected.



Click **Place Order** to send your order direct to our web site. Click Get Quote if the price is shown as unknown. This will take you to our web site to get a quote for a custom manufacture.

Click **Email Order** to send your order to our sales desk using email.

For all options, a copy of the design will also be sent with the order.