Introduction and Tutorial

Copyright Notice

Copyright in the whole and every part of this software and manual belongs to RS Components 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 RS Components. If you use this manual you do so at your own risk and on the understanding that neither RS Components nor associated companies shall be liable for any loss or damage of any kind.

RS Components does not warrant that the software package will function properly in every hardware software environment.

Although RS Components has tested the software and reviewed the documentation, RS Components 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 RS Components 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.

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

DesignSpark is a Trademark of RS Components, Microsoft, Windows, Windows NT and Intellimouse are either registered trademarks or trademarks of Microsoft Corporation.

Eagle is the copyright of CadSoft

All other trademarks are acknowledged to their respective owners.

Copyright © RS Components. 1997-2011. All rights reserved. E&OE

Issue date: 18/10/11 iss 3

RS Components Ltd
International Management Centre
8050 Oxford Business Park North
Oxford
OX4 2HW
United Kingdom
Tel: +44 (0)1865 204000
Fax: +44 (0)1865 207400

Contents

CONTENTS 3
CHAPTER 1. GETTING STARTED WITH DESIGNSPARK 4
Introduction to DesignSpark 4
Introduction to the Tutorial 4
Additional Help 4
System Requirements 4
Installation 4
Removing the Software 4
Starting DesignSpark 4
The DesignSpark Desktop 5
Toolbars 6
Cancelling Commands 7
Shortcut Keys 7
Units 7
Grids 7
Shortcut Menus 8
Undo/Redo 8
Properties of items 9
Status Bar 9
Dockable Windows and Browsers 9
CHAPTER 2. CREATING THE SCHEMATIC 11
Schematic Design Tutorial 11
Starting a New Schematic 11
Adding Components 12
Placing Components 22
Adding Connections 23
Adding Power & Ground Symbols 24
Adding Connectors 26
Adding Net References 27
Your Schematic is Finished 29
CHAPTER 3. CONVERTING YOUR SCHEMATIC TO PCB 31
Overview 31
Converting the Schematic to PCB 31
The New PCB Wizard 32
Starting a New PCB Design without a Schematic 36
Schematic Design Completed 36
CHAPTER 4. PCB DESIGN EDITOR 37
Converting the Schematic to PCB 37
Unroute the Design 37
Changing the Number of Layers 38
Creating a Board Outline Interactively 38
Placing Components 39
Routing the Design 42
Automatic Routing 42
Unrouting the design 43
Manual Routing 43
Picking in tight areas 45
Pouring Copper into Areas 45
Checking the Design Integrity 48
Design Rules Checking 49
Viewing Power Plane Layers 50
Reports, Part Lists and Net Lists 51
CHAPTER 5. MANUFACTURING THE PCB 53
Plots and NC Drilling - Overview 53
The Plotting Dialog 53
The Plotting Process 55

Chapter 1. Getting Started With DesignSpark

Introduction to DesignSpark

Welcome to the DesignSpark PCB 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 DesignSpark 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. DesignSpark 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 DesignSpark or by emailing/calling RS Components.

System Requirements

DesignSpark 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 DesignSpark 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 DesignSpark 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 DesignSpark 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 DesignSpark from your computer, you should do this by selecting the **Add/Remove Programs** icon from the **Control Panel** on the **Start** menu. Select **DesignSpark** from the list of installed programs and click the **Change/Remove** button.

Starting DesignSpark

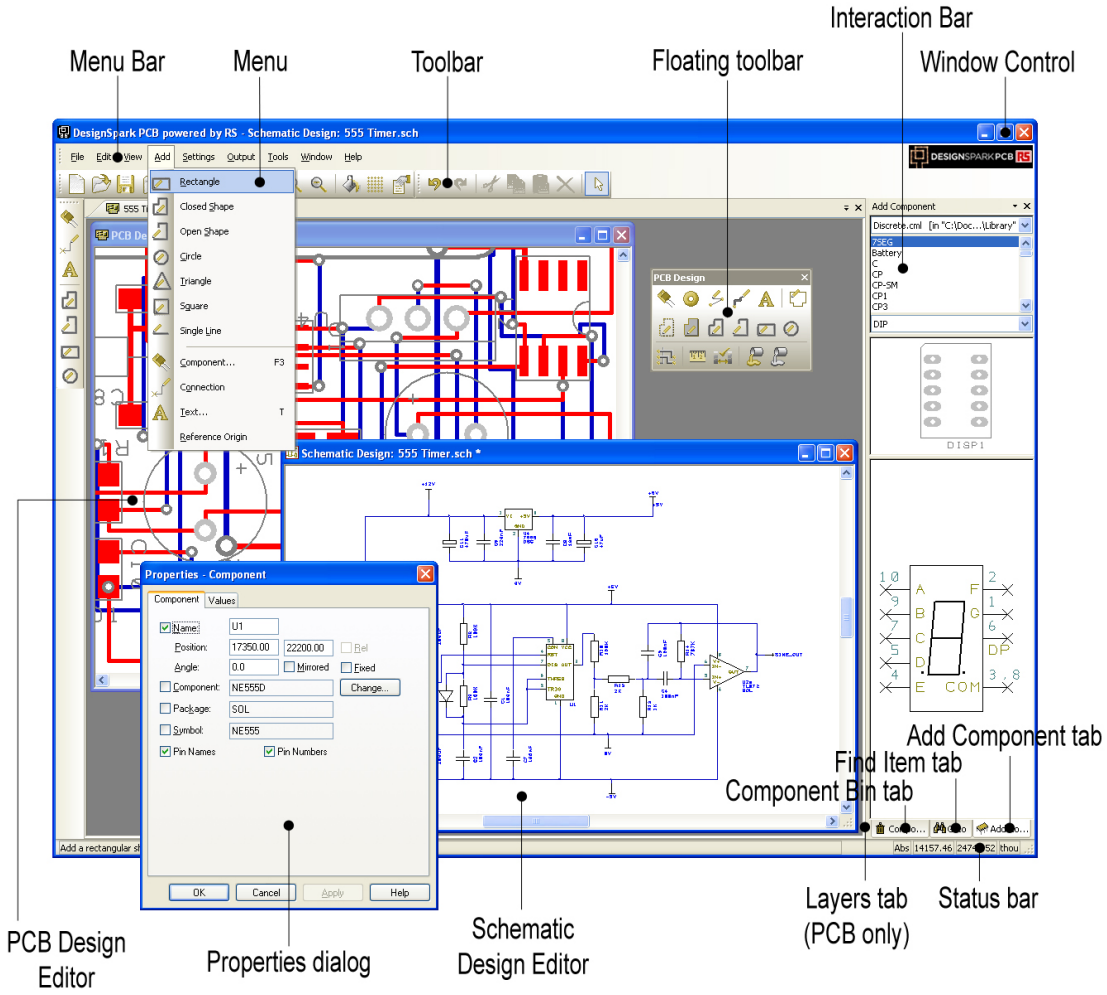
During installation, a DesignSpark program icon is automatically added to your desktop and a RS Components program group added to the **Start** menu and **Programs**.

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

The DesignSpark Desktop

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

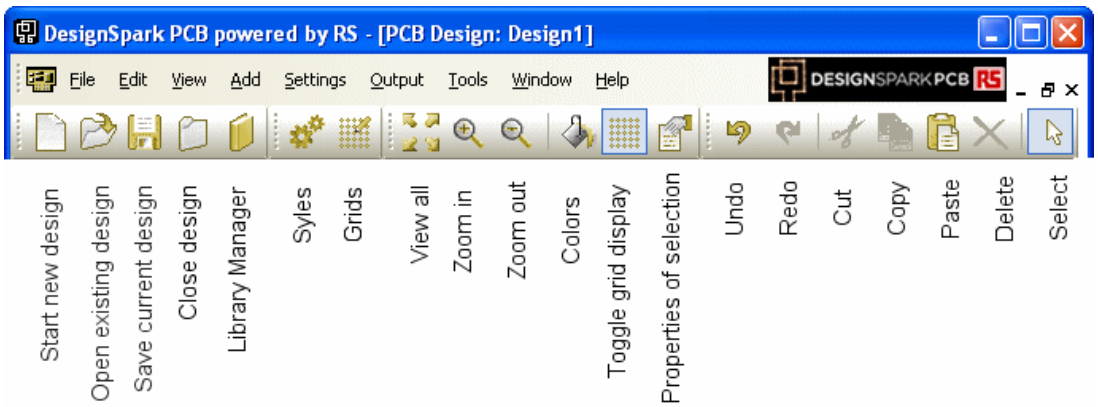
The picture below shows you the major framework facets by name so that as they are used in this tutorial you will understand what is being talked about.



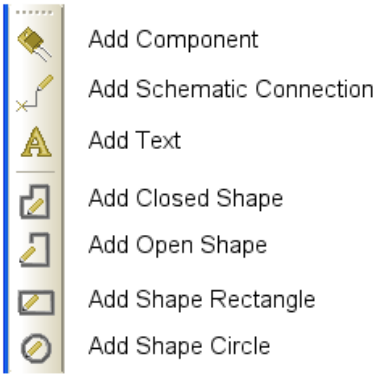
Toolbars

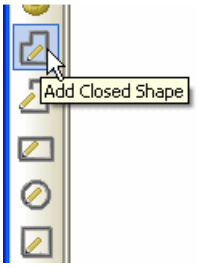
DesignSpark is installed with a set of commonly used tools on the toolbars for you to use.

General Toolbar



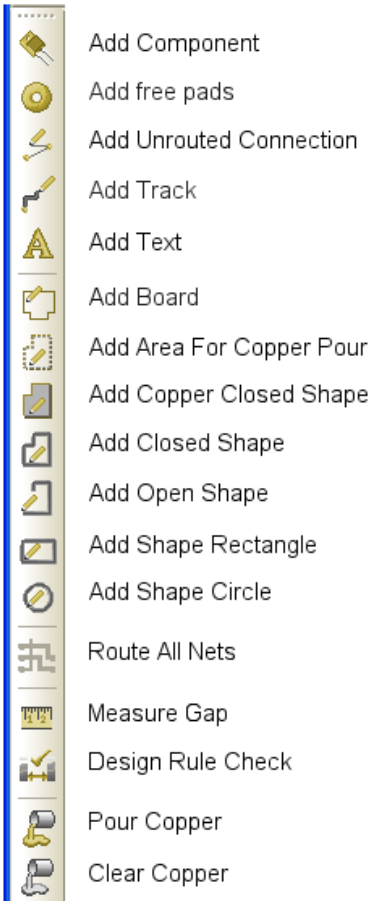
Schematic Design Toolbar





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.

PCB Design Toolbar



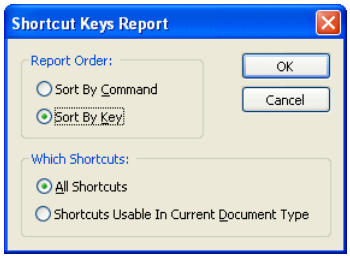
Cancelling Commands

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

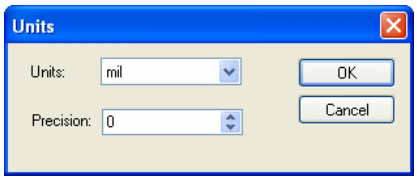
As well as the toolbars, DesignSpark has the ability for you to program your keyboard for commonly used options or commands. This means that once programmed, you can press a key or key combination to enter a function or option. For experienced users, this makes the program's operation far more efficient.

When DesignSpark 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

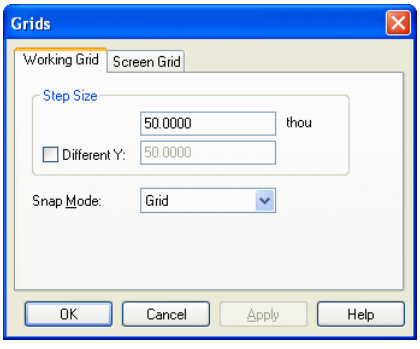
The **Units** option on the **Settings** menu is used to define the units and the precision at which the co-ordinates and lengths displayed in the dialogs and edited by you are displayed. The basic working units can be changed using the **Units:** drop down list.



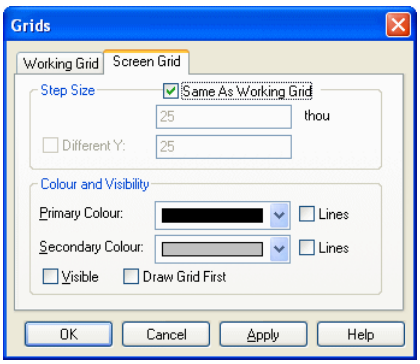
For each type of units, you can specify a **Precision:**. This is the number of decimal places displayed; the value is simply rounded to the nearest last digit.

Grids

Grids are used in DesignSpark 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.

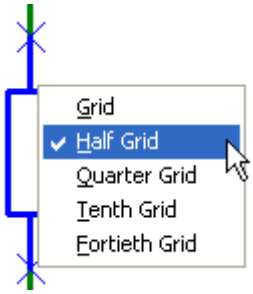


The **Screen Grid** page is used to define the display of 'dots' to guide you while designing. **Size Steps:** is used to define the 'pitch' of the grid and uses the **Secondary Color** to display them. The **Primary Color** is used to display another grid based on a 10 times larger size than the Secondary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

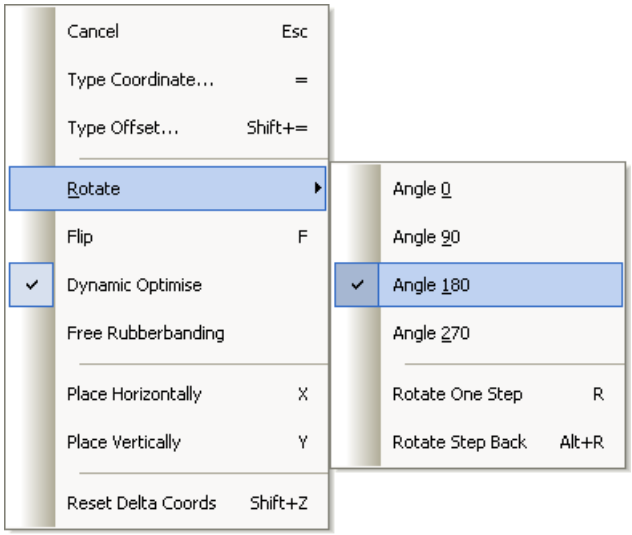
The **Snap Mode** shortcut can be displayed during editing by selecting the direct keyboard command <**Ctrl-G**>. By selecting the snap mode setting you can use predefined multiples of the Working Grid to move items for example.



Shortcut Menus

The right hand mouse button can be used extensively throughout DesignSpark 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.



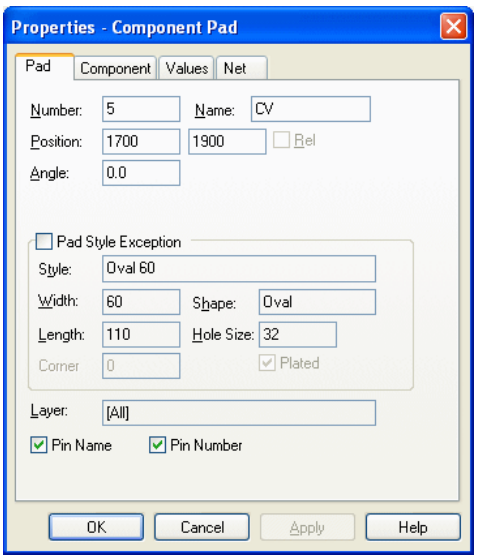
Undo/Redo

DesignSpark 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

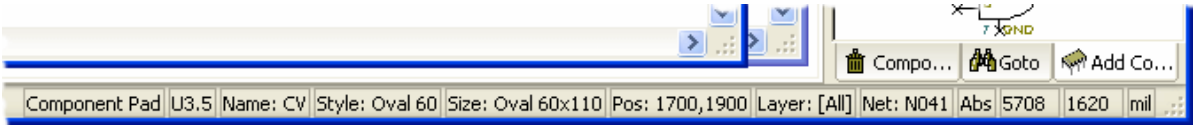
Each **item** in the design contains a set of characteristics which can be viewed or edited using the **Properties** option. Some items within the design are made up of several entities, each of which is individually selectable and has its own properties that can be viewed.



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.

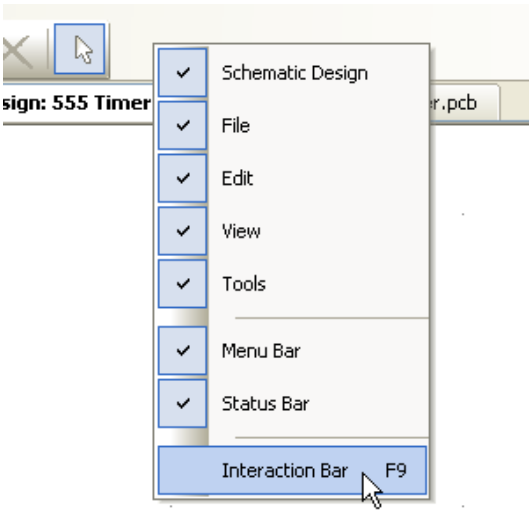
Status Bar

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.

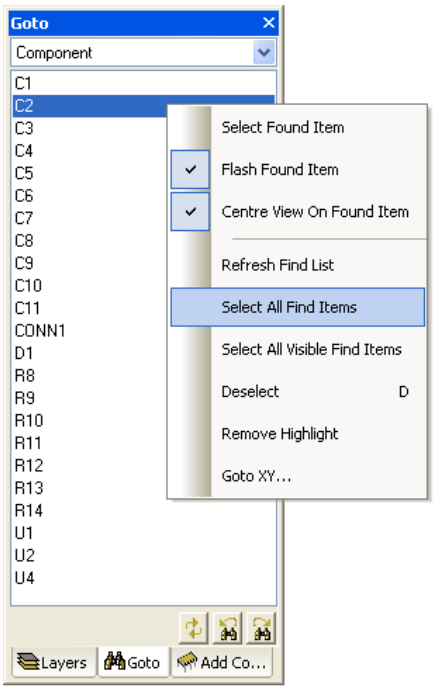


Dockable Windows and Browsers

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



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.

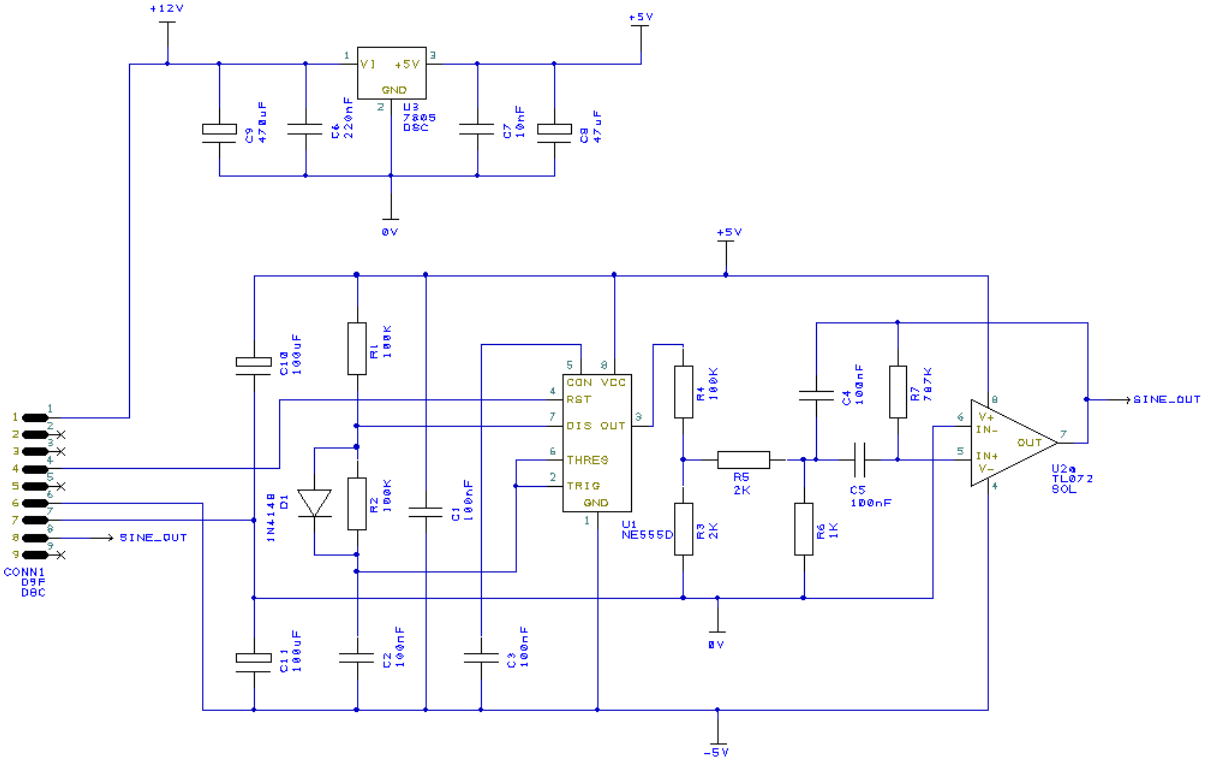


Each individual browser can be opened by selecting one of the three tabs at the bottom edge of the browser. You can access **Layers** (in PCB only), **Component Bin** (in Schematics only), **Goto** and **Add Component** options.

Chapter 2. Creating The Schematic

Schematic Design Tutorial

During the tutorial we will create a Schematic design; this is how it will look when finished.



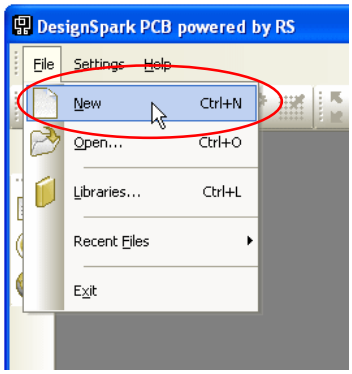
During the design process, we will walk you through all the aspects of drawing this and adding the detail to give it intelligence.

The design consists of two parts; the first is the power regulator circuit at the top which converts a 12V supply to 5V. The bottom half is the main 555 timer circuit itself.

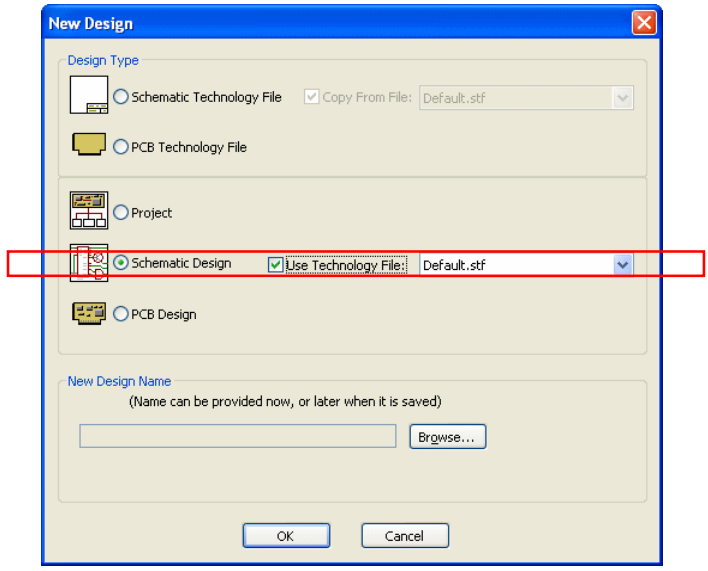
Starting a New Schematic

▶ To start a new Schematic design or a PCB design

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



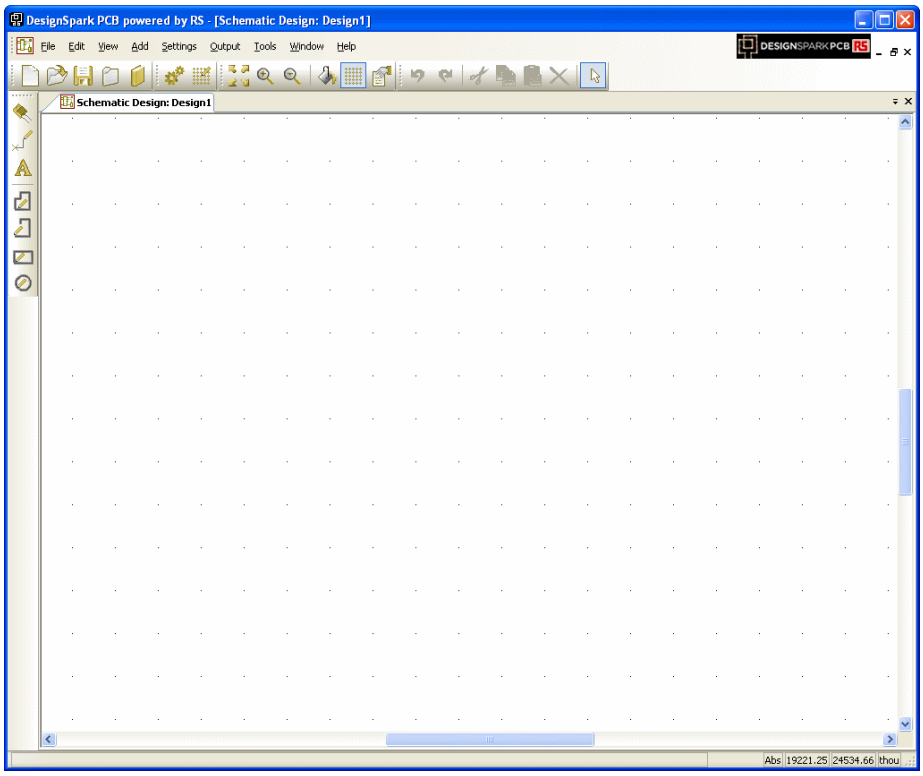
Check the **Schematic Design** button and the **Use Technology File** button. The **Default.stf** template will help you get started quicker, select this file. More detail about Technology files is available in the Online Help.



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 **PCB Design** button and **OK,** follow the New PCB Wizard. Then skip to the chapter headed, ***Starting a New PCB Design***.

Your DesignSpark window will look like this ready to start:



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

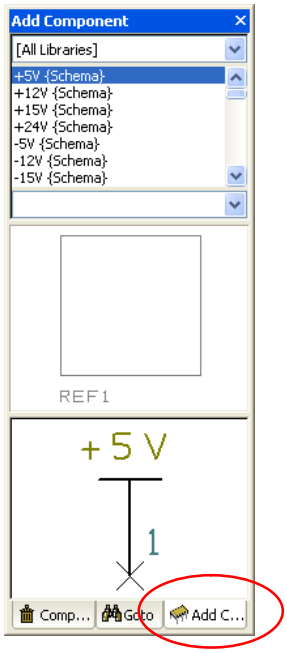
Adding Components

▶ To add a Component

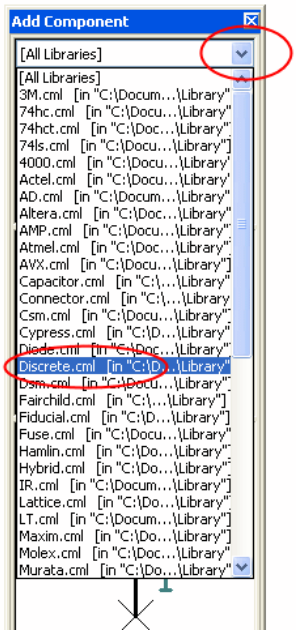
There are two methods for adding components; we will use the **Interaction bar** to start with, you can also use the **Add Component** option as well.

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.

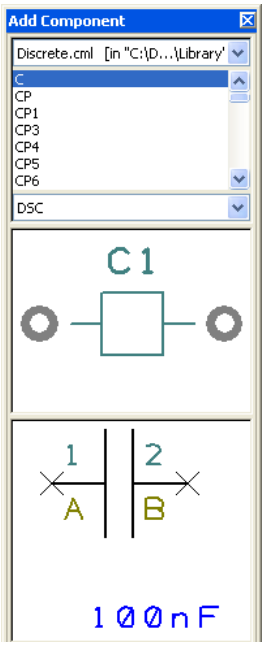
Select the **Add Component** browser tab at the bottom of the Interaction bar.



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.

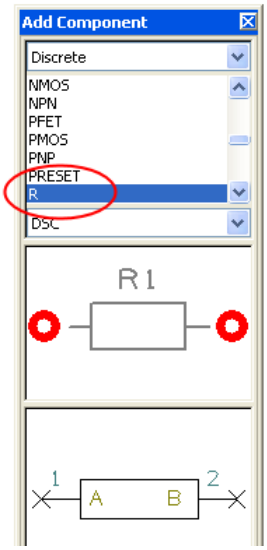


The **Component** list will be populated with components. Selecting this library will automatically select the first component in the list as well.



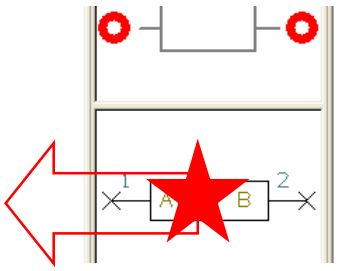
Scroll down the list of components using the vertical scroll bar. The first in the list is the generic '**C**' component. You can also click in this list and type the first and subsequent letters and numbers of the item required.

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 DesignSpark.



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

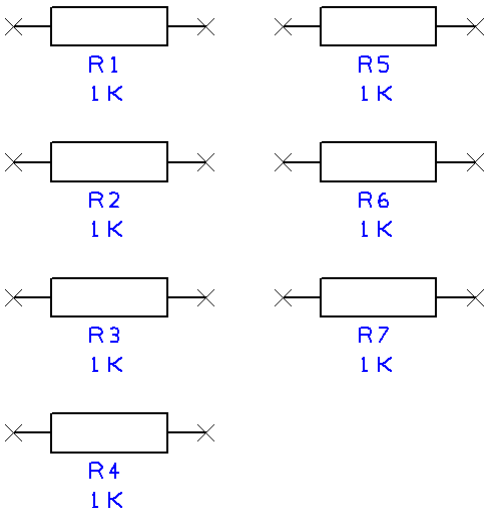
Once you have the correct component, you can add it by selecting the symbol preview and dragging it into the design. You can also add the component by selecting and dragging its name.



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.

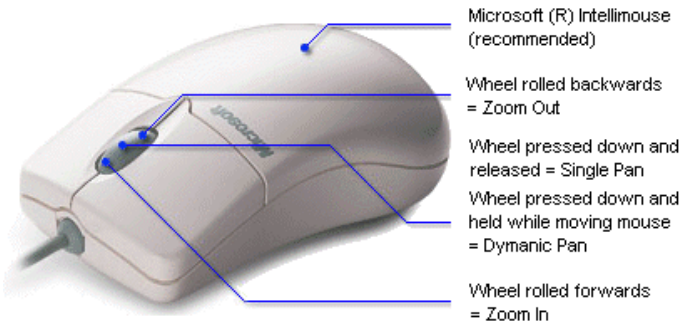
You need 7 resistors in total for the final circuit. Once the first one has been placed, pick and drag 6 more from the **Add Component** window. Position them in a stack like the picture below. For now, their position isn't important anyway.



▶ 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**>.

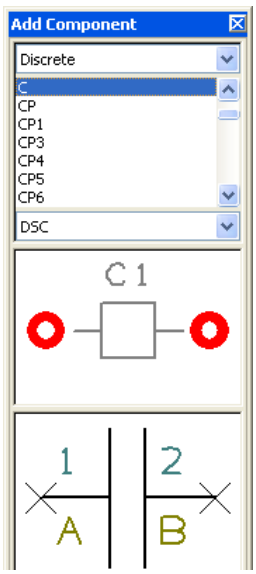
You can zoom while the component is active on the end of your cursor.



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

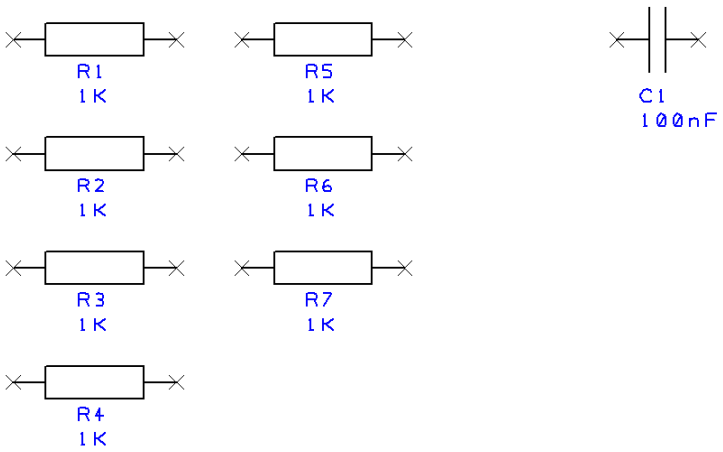
▶ To add the capacitors

We need 7 normal capacitors for this circuit plus 4 electrolytic capacitors. 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.



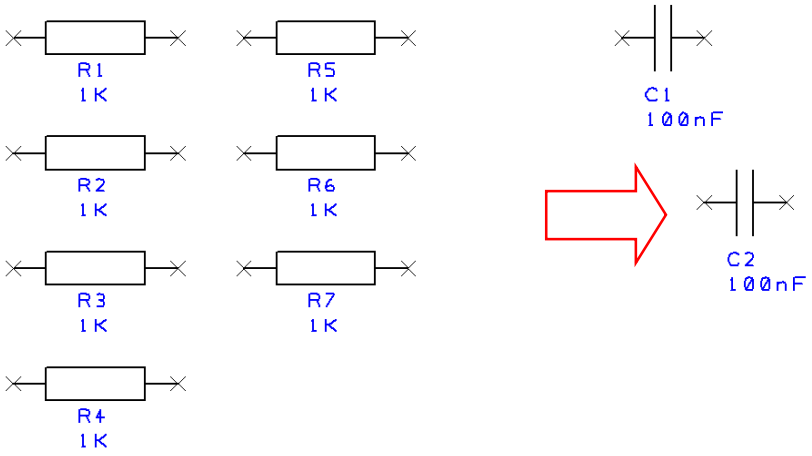
The preview windows confirm this is the correct component. As with the resistors, this is a generic component and we will change its value later on.

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:



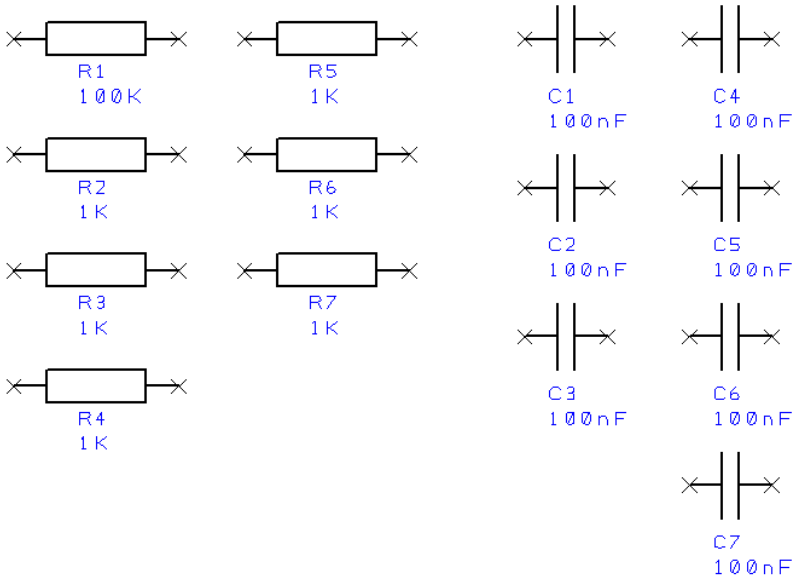
We will use a different technique to add more capacitors.

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 **C7**.

Your design now looks like this:



Obviously in a design as simple as this, it would have been quicker to drag out all 7 capacitors but the Copy/Paste principle is worth demonstrating as you will see its value later one.

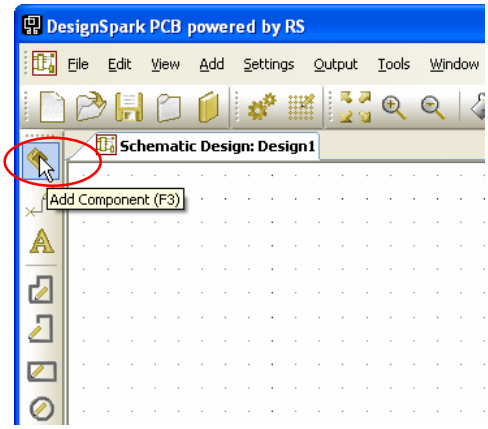
▶ To switch off the displayed grid

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 **555 Timer** and a **TL072**. We will use another technique to add these components. There will be other components to add also but we can add these later as you would in a real design; building the design up as you progress.

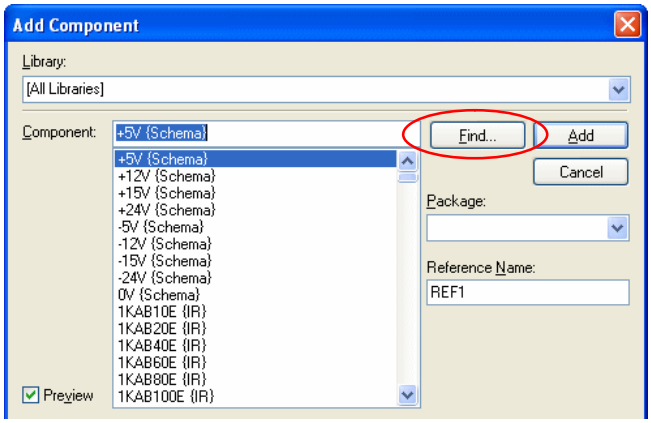
Click the **Add Component** button on the toolbar, shortcut key <**F3**>.



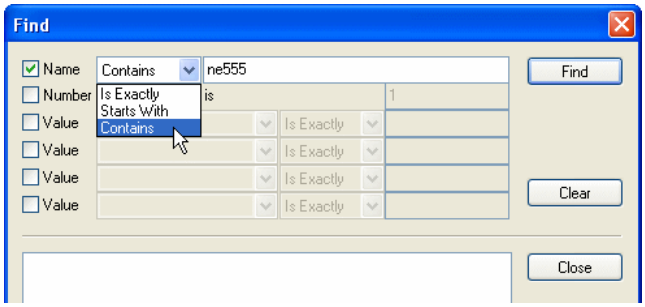
The Add Component dialog will display [All Libraries] in the library path. We will us the Find option to locate both of these components in the library.

▶ To Find components in the library

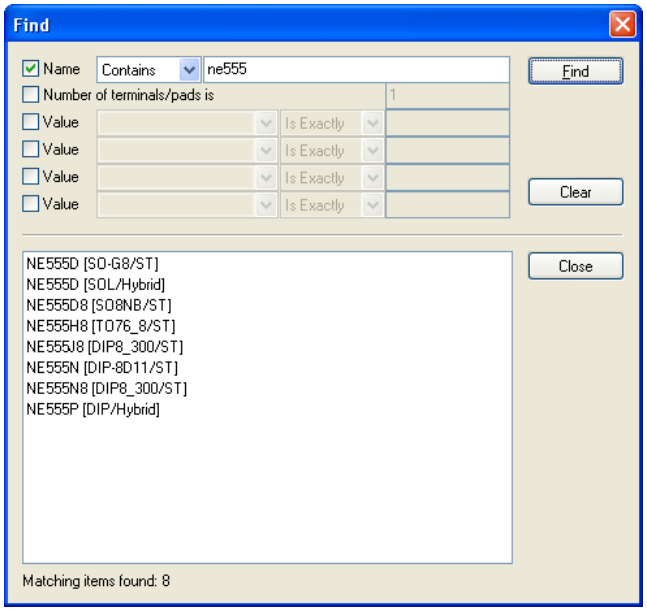
From the Add Component dialog, click the **Find** button.



We will find the NE555 timer first. For the **Name** selection, choose **Contains** from the drop down list. In the text entry, type **ne555**. This will be enough to get near a name to then match what we need.



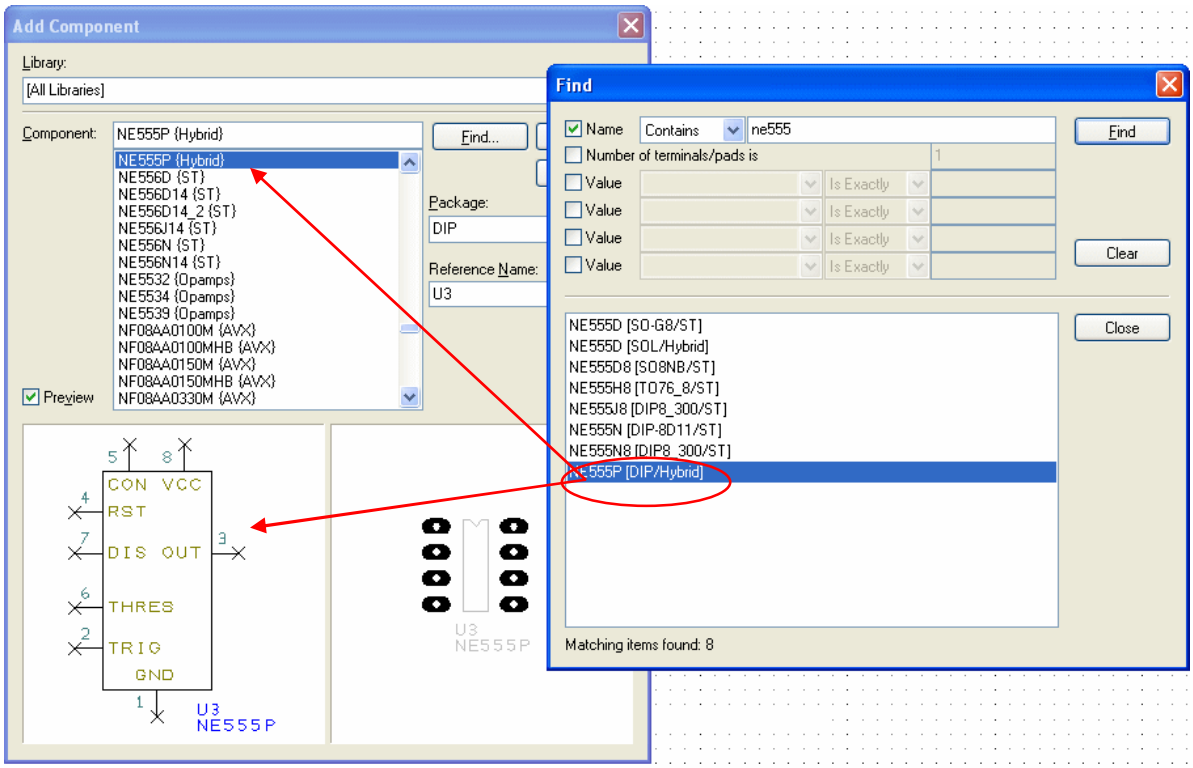
Click the **Find** button. The contents list will be populated with all matching items from all the libraries.



Select **NE555P** at the bottom of the list from the Hybrid library.

You can either send this component to the design by using a **double-click** or you can preview it in the **Add component** dialog contents list by selecting the **Close** button. We will use the Add Component method on the first device and the double-click method on the second.

After selecting, you'll notice that this component is now preselected in the **Add Component** dialog behind this.



Use the **Close** button on the Find dialog.

▶ To add components

With the **NE555P** selected from the **Hybrid** library, press the **Add** button and add it to the design.

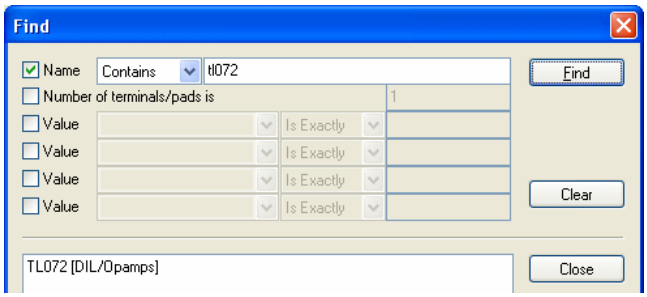
Depending on your current zoom level, you may need to roll the mouse wheel to zoom out a little. You will need to give yourself some space on the design in which to place this new component.

The NE555 (U1) is shown in the design on the end of your cursor. Position it next to the capacitors but it doesn't matter where it goes for now.

Go back to the **Add Component** dialog <**F8**> and click the **Find** button again.

Type in **tl072** – note the case typed isn't important.

Click the **Find** button.

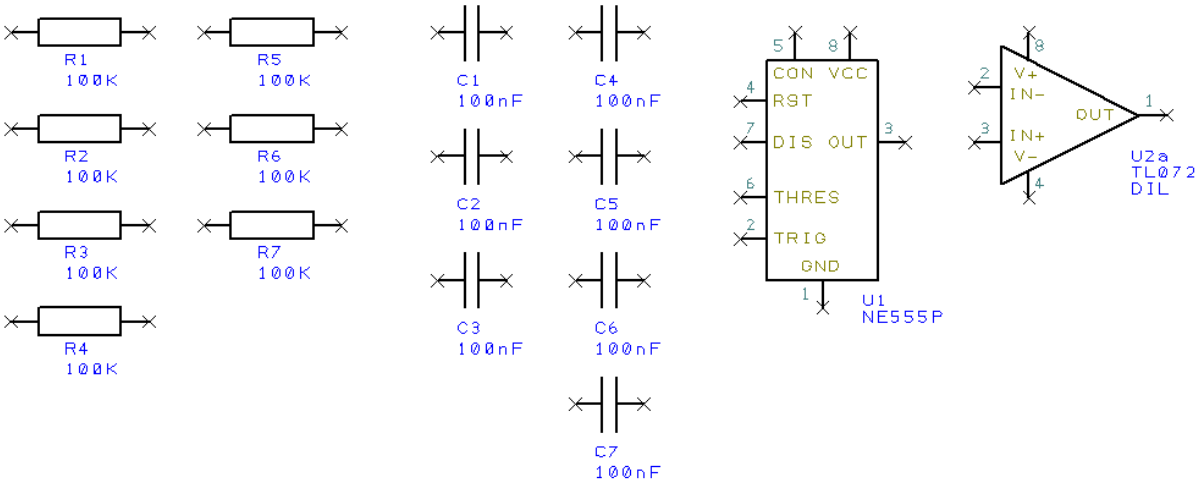


This time, double-click on the name in the found list. Both the Find dialog and Add Component dialog will close and the component will be on the end of your cursor ready to place in the design.

This is a two-gated device. Click the mouse once to place the first gate. The second gate will now be dynamic on the end of your cursor ready for placing. We do not need this gate so it can be discarded (into the Component Bin off the design sheet).

Click the <**Esc**> button to discard it.

The design now looks like this:

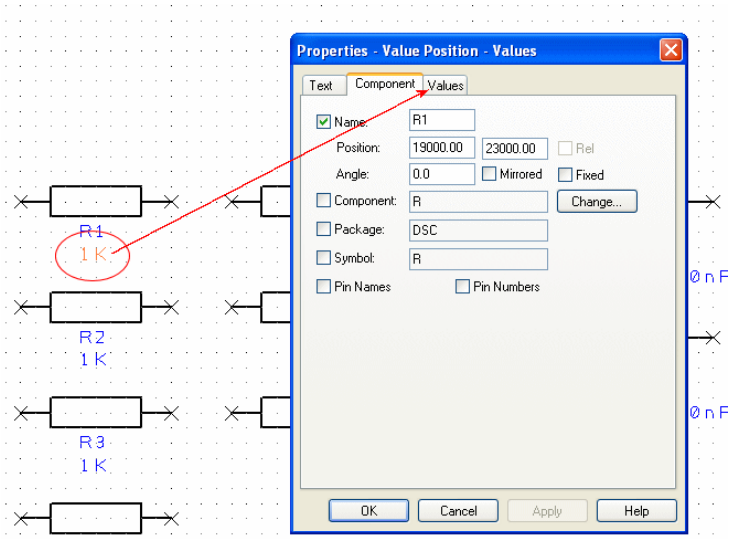


Before we make the connections we should change the component values to be the ones required. This is easy to do because we've used 'generic' components. If you prefer to use specific Part named components (a unique Component for each device you order and buy in), you can also work this way with DesignSpark.

All the resistors and capacitors we have used in this example circuit are generic. They all have the same values at the moment. In the next exercise, we will change their values to be unique and as required.

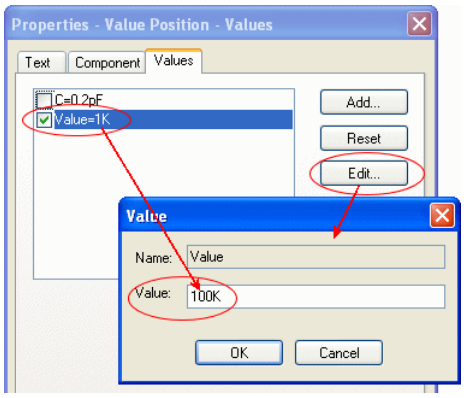
▶ To edit component Values

Double-click on the Value field (**1K**) on **R1**, the **Component** tab of the **Properties** dialog is displayed.



We need to use the Values table to change the values so 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 **100K**.

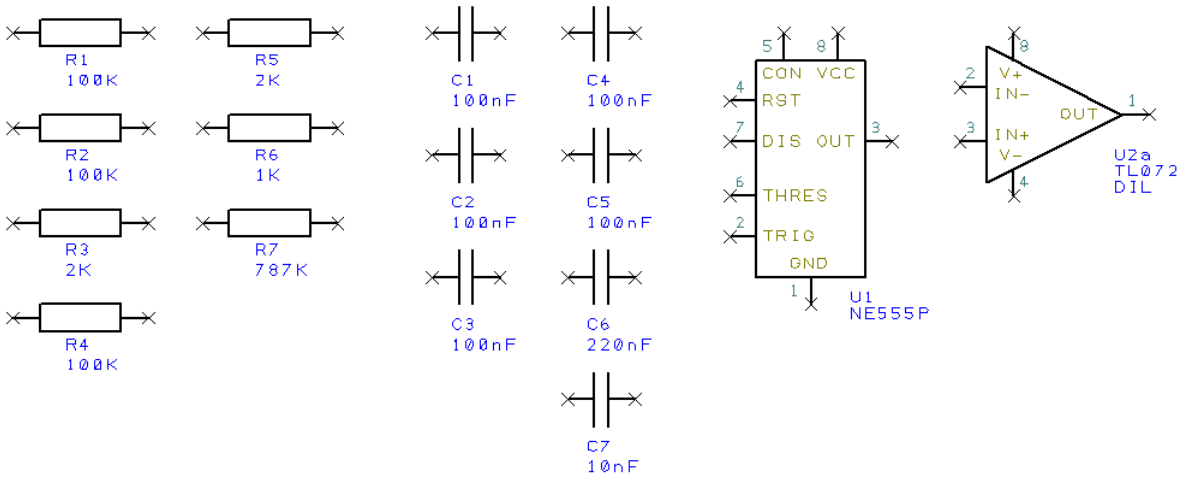
When done, click the **OK** button to exit this dialog and **OK** again to exit **Properties** dialog. When the value of R1 is changed to 100K the design is updated.

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 100K	**R2** 100K	**R3** 2K	**R4** 100K	**R5** 2K	**R6** 1K	**R7** 787K
C1 100nF	**C2** 100nF	**C3** 100nF	**C4** 100nF	**C5** 100nF	**C6** 220nF	**C7** 10nF

A quick tip here is that if there are a couple of components with the same value, edit the first one then copy it as many times as you need rather than editing each one.

Our design now looks like this:



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

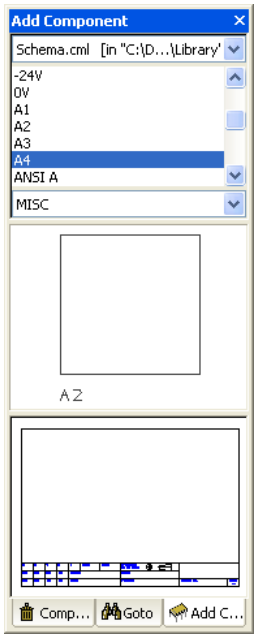
You can also use the **Values** option on the shortcut menu instead of **Properties**. This just takes you directly to the **Values** page of **Properties** to save time.

▶ 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.

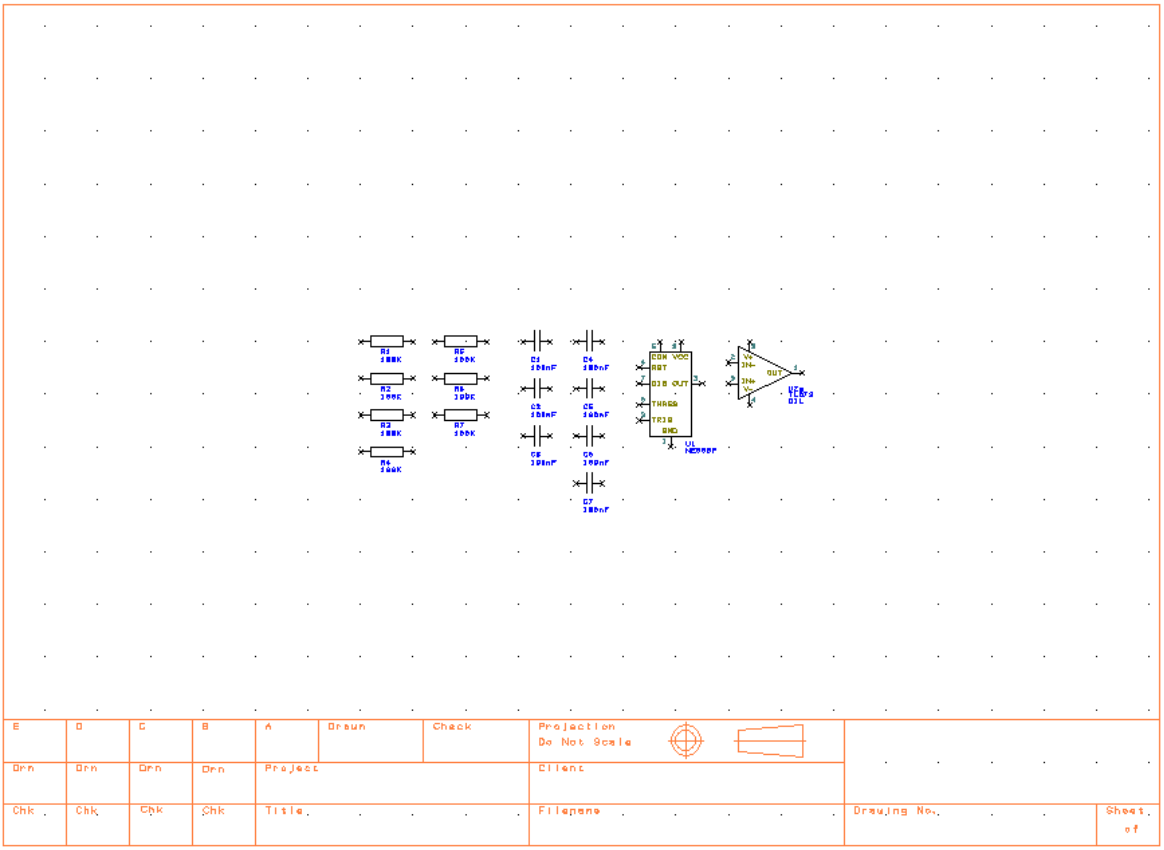
Using the **Add Component Browser** again (<**F9**>), select the **Schema** library from the drop down list. For reference, this is a schematic only component library and will not have any PCB symbols associated with them.

Scroll down the list and select the **A4** drawing border. This has been previously created to fit an A4 page size.



Note there is no PCB footprint for this component, it is Schematics only.

Like other components, select **A4** 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 all to place it somewhere near. Use the <**Z**> key to zoom out or <**A**> to view all of the design. For speed you can also roll the mouse wheel.



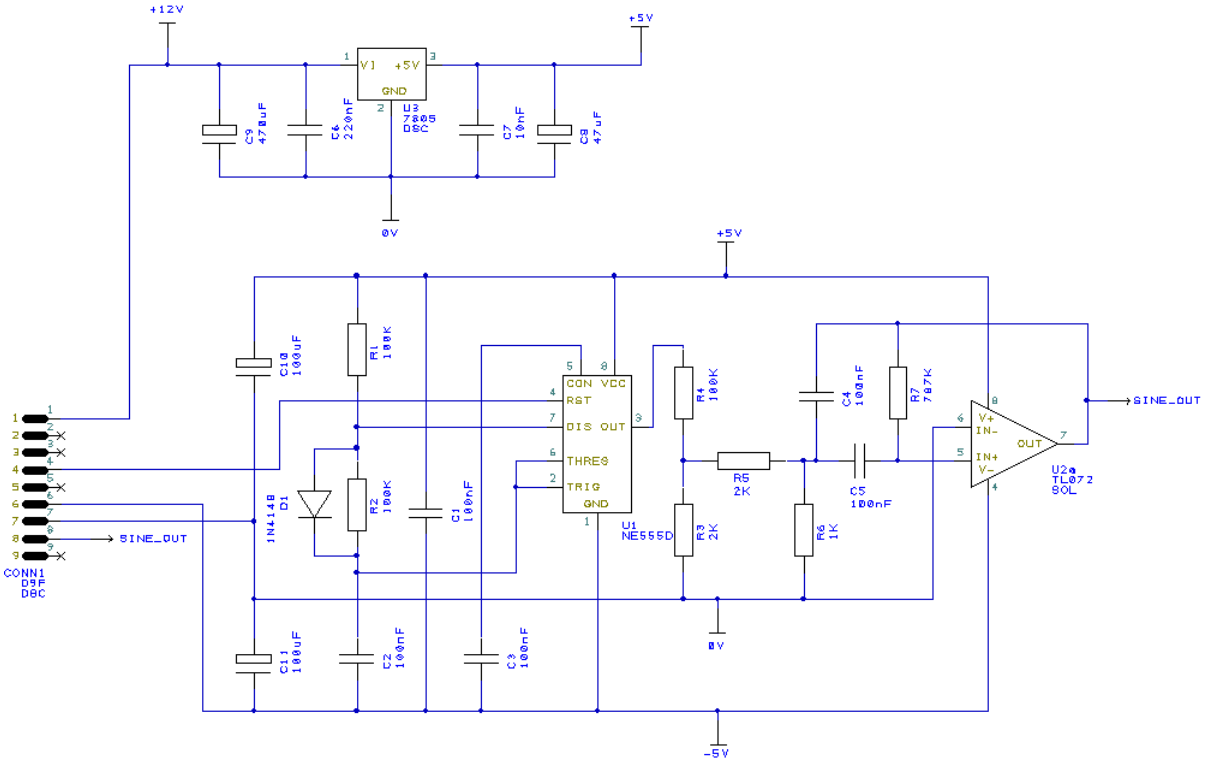
As with other components, this can be moved and edited later if you need to, extra detail can be added to it. Click once to release it.

▶ 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 U1 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 full circuit looks like this:

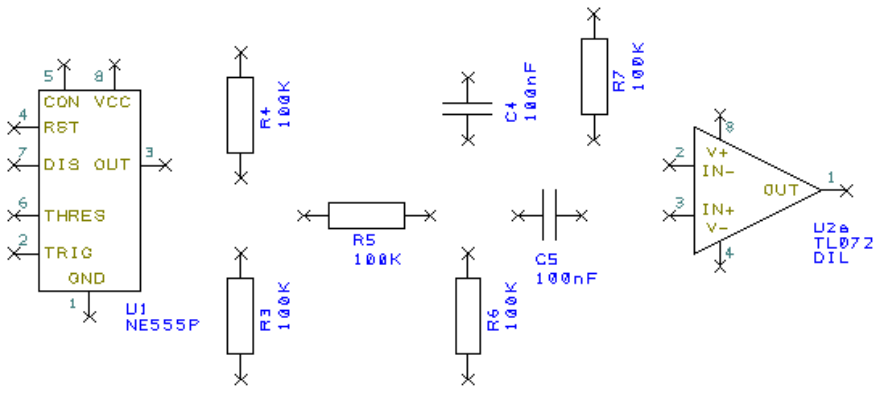


Move the components into position by **picking** and **dragging** them in one move using the mouse. This methodology is used throughout to save time and effort. It means, a continuous select and drag of the mouse without releasing on a component (or other items, this works for moving any item). When you release the mouse, the component placed and deselected in its current position. If you make a mistake, use **Undo** <**Ctrl-Z**> to restore its position.

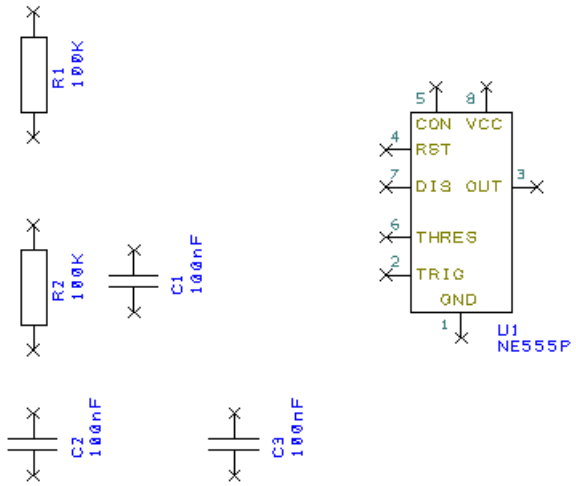
Components can be rotated during move by clicking the <**R>** key. A single click of the <R> key will rotate the component by 90 degrees. Click <R> more than once to rotate it by another 90 degrees and so on. There are other rotate modes and rotation increments available on the shortcut menu when moving the component.

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

The design will now look like this (with part of the circuit placed):



Place the other components. The order in which you place them doesn't make any difference. In a real design, you would add and place them logically as you build up the circuit.



Two capacitors are left unplaced (**C6** and **C7**), these will be placed at the a later time when we create the voltage regulator.

You are now ready to add some 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:

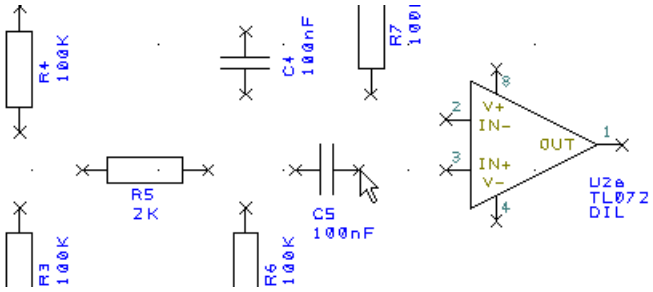
- Using the **Add Connection** option from the **Add** menu or from the **Schematic** Toolbar, click on the pin to start adding a connection, or;
- Double-clicking on a component pin to start a new connection, or;
- 'Dragging' off an unconnected component pin to start a new connection.

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

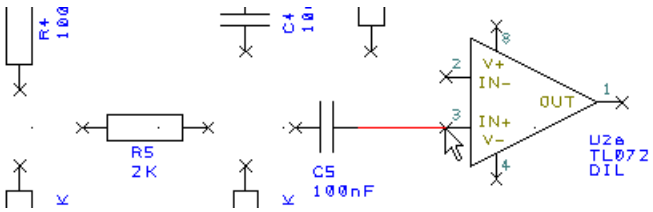
▶ To add connections by dragging off pins

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 C5 like the picture below.



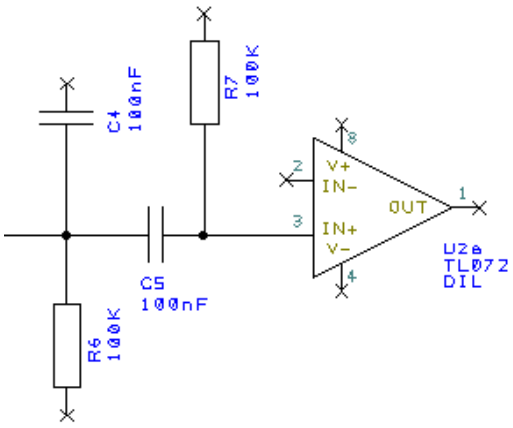
Click and drag (keeping the mouse button pressed down) off pin 2 of C5, this will start a new connection. Move the mouse over U2a pin 3 and single click to finish.



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

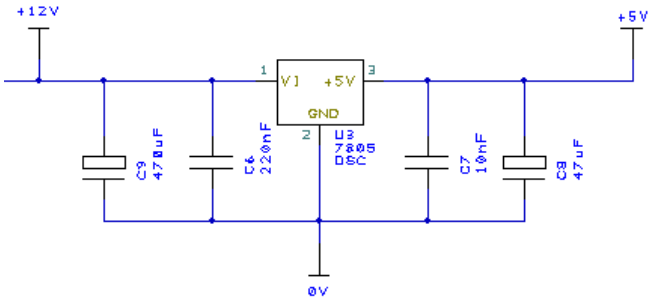
There are a couple of connections which must be connected together. Let's assume that you've made a connection and now you need to connect another to it using a junction dot. When you attempt to finish a connection on an existing connection, when you release the mouse, the connection will automatically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



Following the example design above, complete the design adding the connections required.

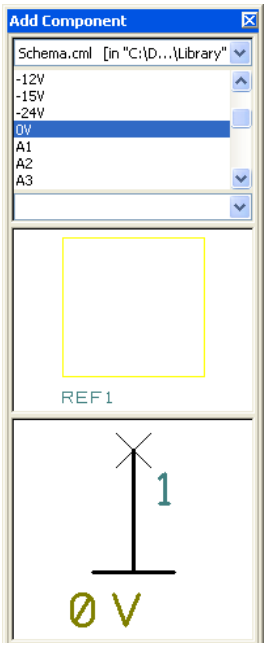
Adding Power & Ground Symbols

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

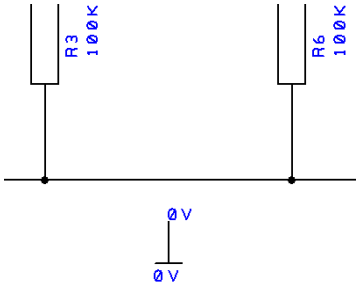
Using the **Add Component** browser on the **Interaction bar** <**F9**>.



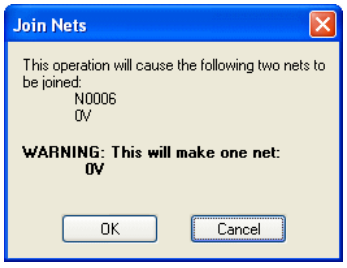
The **Schema** library will still be selected after using it previously when adding the Letter component. If for some reason it isn't, select it now from the library list.

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 between **R3** and **R6**, click the mouse to release it.



Using our working design, add more **0V** symbols to the design as shown. Connect these by dragging off the symbol to the existing connection. A confirmation dialog will be displayed to indicate a non-named net is being connected to a named net. This ensure you aren't joining nets which you don't know about.

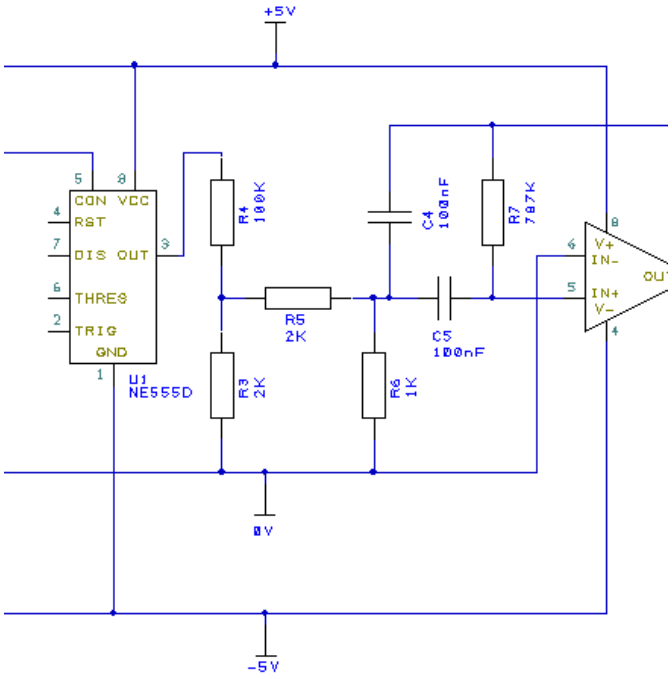


Click **OK** to confirm this.

From the **Add Component** browser, add one **+5V** symbol this time and place near **R4** and **C4**. Now add one **-5V** symbol, place it near **C3**.

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

Your design will look something like this:

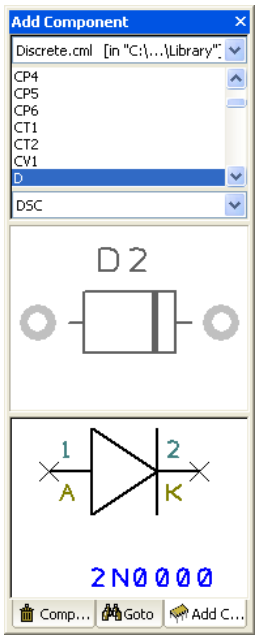


Remember to save the design. Use **Save** from the **File** menu.

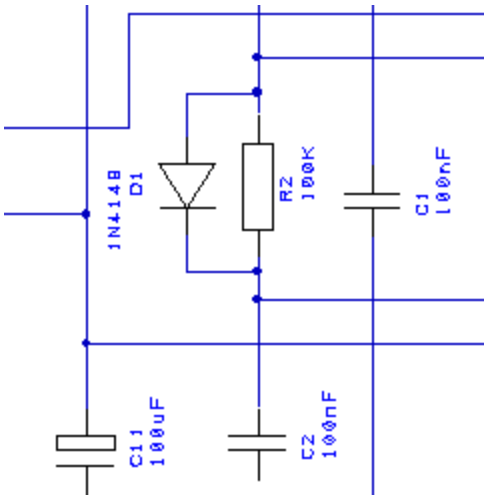
You will need to add a diode to the design.

From the **Interaction** bar <**F9**>, use Add Component. Select the **Discrete.cml** library and choose **D** from the list. This will give you a generic diode.

If you prefer, and now that you know how to do a **Find** in the **Add Component** dialog, search for a specific diode. You actually need a **1N4148**, but if you choose the generic dialog from the library, edit the value so that you make it a 1N4148.



Add the diode by dragging it from the browser into the design. Place it next to **R2** and rotate it by 270 degrees (click <**R**> 3x times.



Add two connections to connect it to both sides of **R2** as shown.

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

Adding Connectors

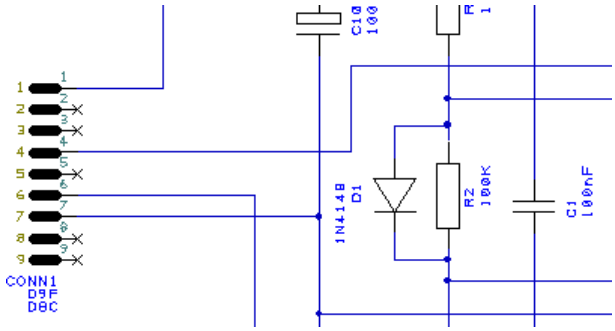
For this example, we will use a connector component from the library. Connectors are simply additional library items and not special devices as such.

▶ To add Connector pins

Using the **Add Component** browser from the **Interaction bar** <**F9**>. Select the **Connector.cml** component library.

Select the **D9F** connector component from the list. The PCB Symbol and Schematic Symbol previews will be displayed.

Drag it into the design and place it shown below. You will need to Flip it by using the <**F**> key. This will display it the way we need it.

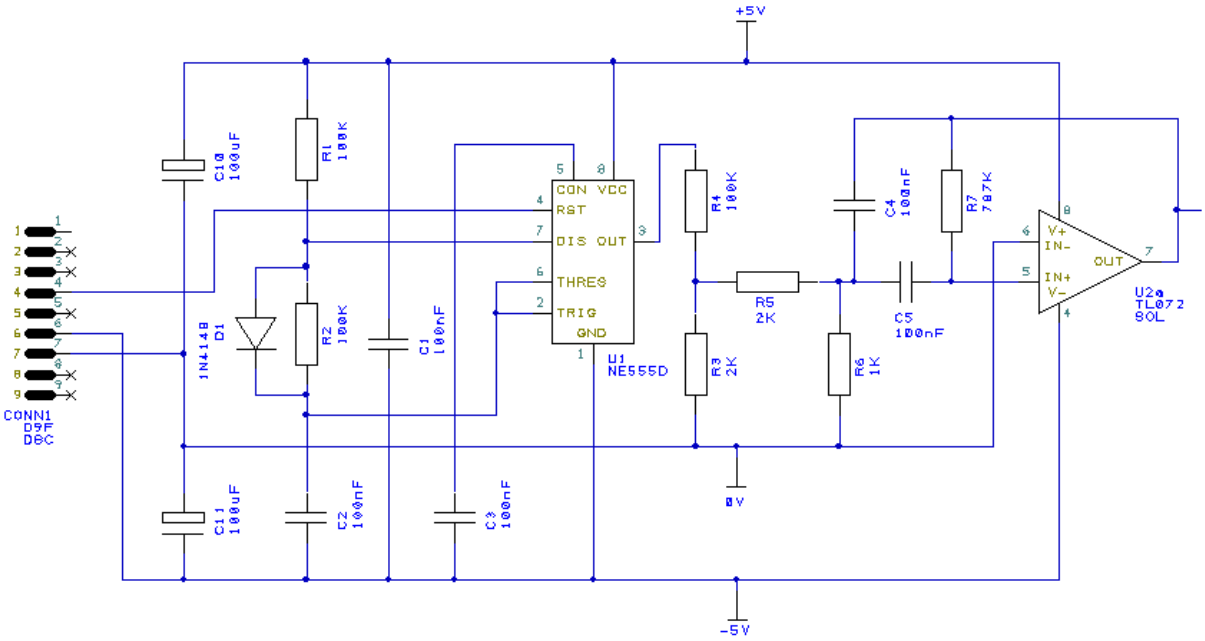


Connect in the connector to the design using **Add Connection** or by **dragging** off a pin.

▶ Add more components

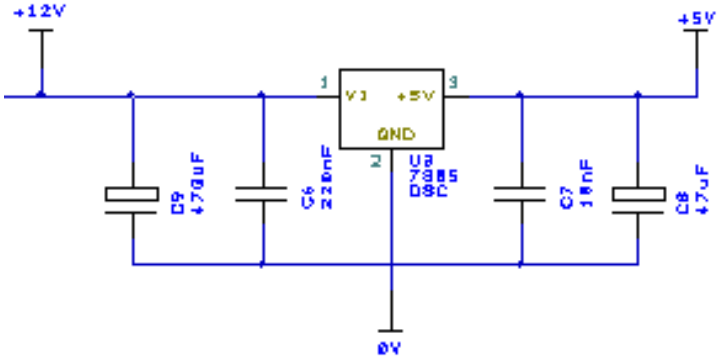
You will also need to add four more capacitors, but this time, add the Component **CP** using the **Add Component** browser or dialog and use the **Discrete.cml** library. Add **C8**, **C9**, **C10** and **C11**. We will use C8 and C9 a bit later on.

Place **C10** and **C11** near the connector **CONN1**. Connect these as shown below:



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

Using the picture below, complete the circuit by adding the voltage regulator circuit.



We already added the capacitors **C6**, **C7**, **C10** and **C11** but you will now need to add **U3** which is a **7805**.

Change the component values for the remaining capacitors to these values:

C8 47uF **C9** 470uF **C10** 100uF **C11** 100uF

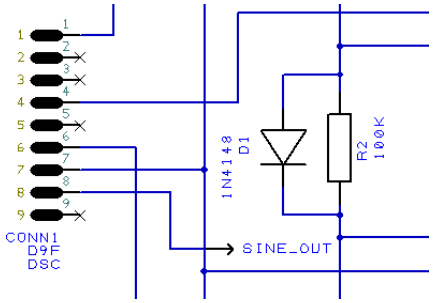
Add Signal references for **+12V**, **+5V** and **0V** from the **Schema.cml** library as you did before. If the symbol already exists in the design, simply copy and paste it for fast addition.

The input side of U3 on pin 1 is +12V. This will be connected to pin 1 of Conn1, the connector.

Adding Net References

As an exercise, we've added a Net Reference to the design. This is two pins on opposite sides of the design perhaps which need to be connected together but for which you do not need to add a full connection.

We will add a short connection for each pin, add a schematic only component as the signal reference and name the net on both ends. This will act as an 'implied' connection. Then when the Schematic is translated to PCB, these two points will be connected together.



From the Add Component browser, from the Schema.cml library, add two **To** components to the design.

Add a **connection** between **CONN1** pin 8 and the **To** symbol.

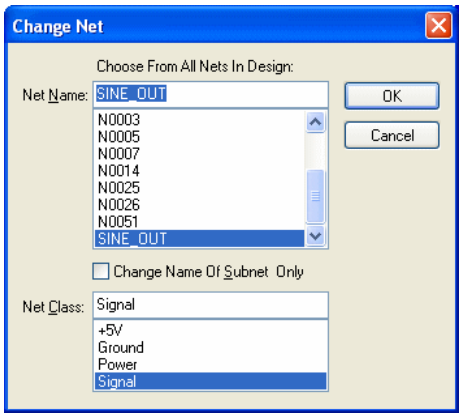
Remember, the intelligence in the implied net is the net name, so now we must name the connection.

▶ Using Change Net

Select the pin on the **To** symbol. From the shortcut menu, select **Change Net**.

From the dialog, type the **Net Name: SINE_OUT**

Leave the Net Class as Signal. Click **OK** to exit.



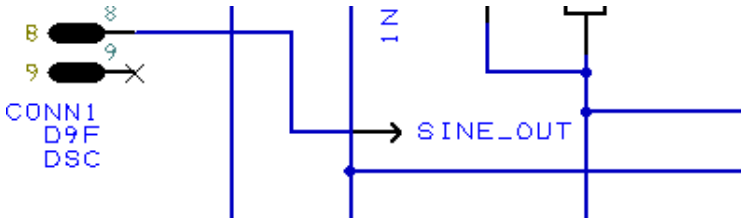
The Net is now named (SINE_OUT), now you need to display it.

▶ Displaying Net Names

Select the pin on the **To** symbol again.

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

Now that it is displayed, you can select it and move it into the required position.



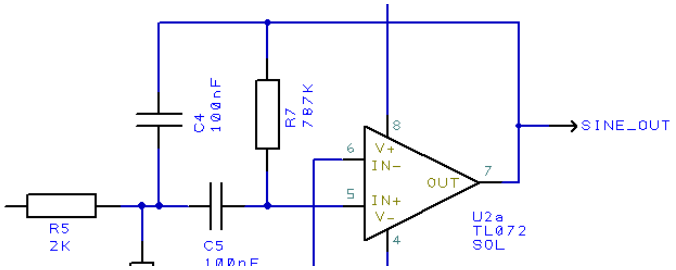
That is one half of the implied net completed. You must now do this procedure again for the other pin.

▶ Making the other half of the implied net

We will move the second **To** symbol to the right side of U2 (the TL072 component).

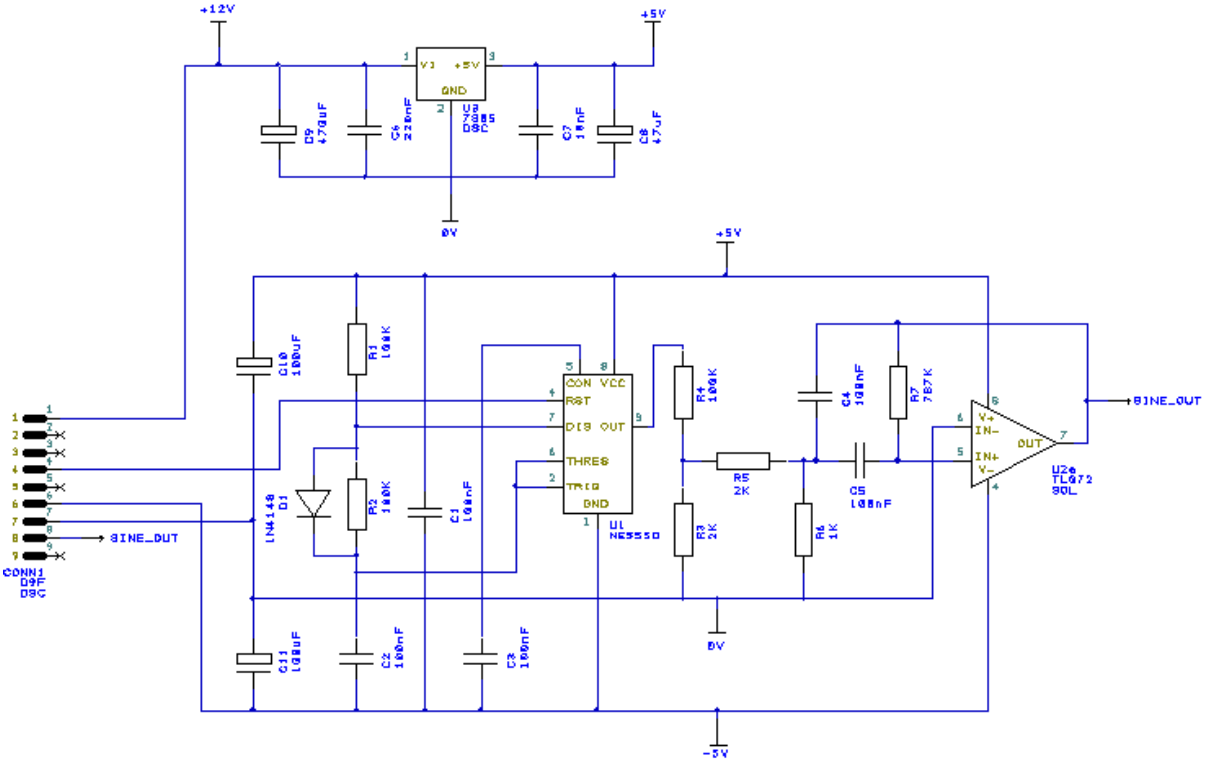
Add a small **connection** between the output **pin 7** of **U2** and the **To** symbol.

Now using the same technique as we used just above, change the Net Name to SINE_OUT and display it using the **Display Net Name** option from the shortcut menu.



Your Schematic is Finished

The schematic design is complete, remember to save it now. 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 *Converting Your Schematic To PCB*

Chapter 3. Converting Your Schematic To PCB

Overview

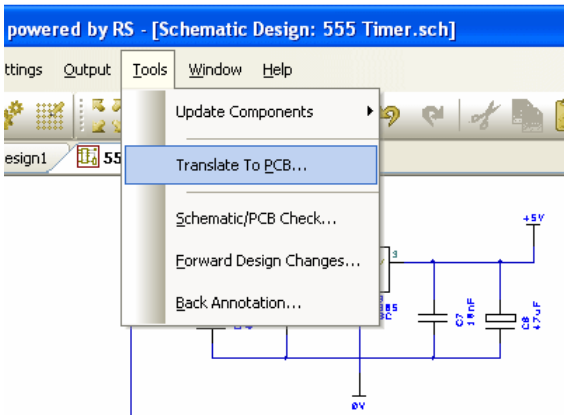
There are three methods for creating new PCB designs:

1. From a Schematic using DesignSpark as the capture 'front-end' to the PCB system. You will use the Translate To PCB option to take the netlist into the PCB editor. Nets and components used in the Schematic are converted to PCB in the integrated environment.
2. Use the DesignSpark 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.
3. From an external netlist and import it through the import mechanism to create a new PCB design.

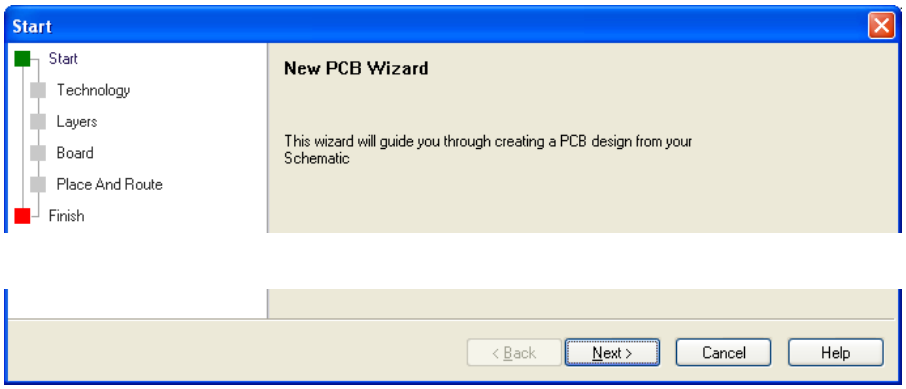
Converting the Schematic to PCB

If you created a **Schematic** in DesignSpark, 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 **Translate To PCB** from the **Tools** menu and use the **Translate To PCB** option.



This will display the **New PCB** wizard. Whether you create a PCB design by using the **Translate To PCB** option or by using the **New** option and **PCB Design**, you will be presented with the **New PCB Design Wizard**.



The New PCB Wizard

This wizard presents you with a set of dialogs which step you through a 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.

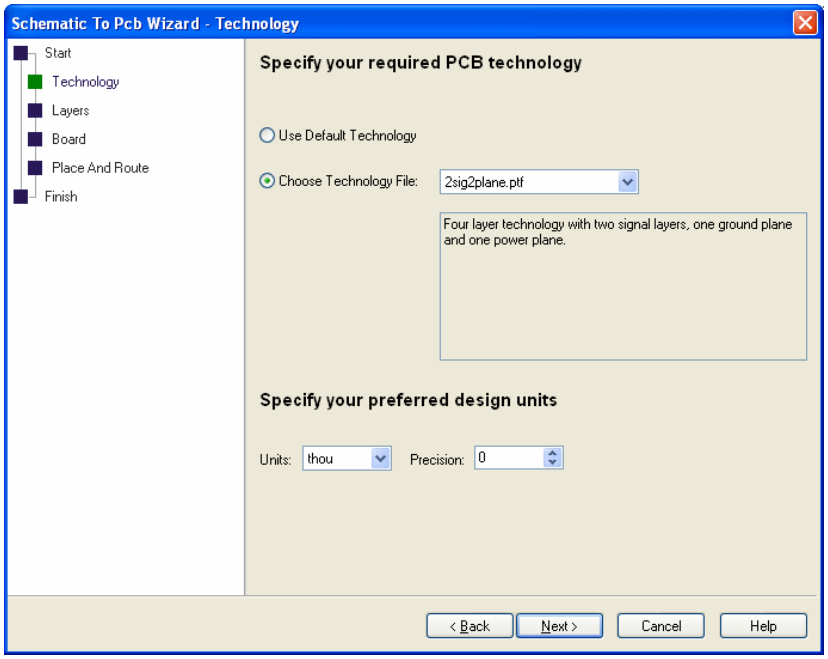
On the **Start** page select the check box to agree with the disclaimer. You must click this to proceed further. It is only shown once after you have selected it for the first time. Click **Next>** to proceed.

The **Technology** page allows you to predefine a Technology file and your design **Units**. Technology files allow you to create a set of sizes (styles) and design spacing rules that can be used time and time again on different designs. This enables you to be consistent with naming conventions and standards. It also provides you with a fast way to get your designs started.

The **Units** are defined here to get you started, as are the unit **Precision**, but both can be changed at any time during the design process.

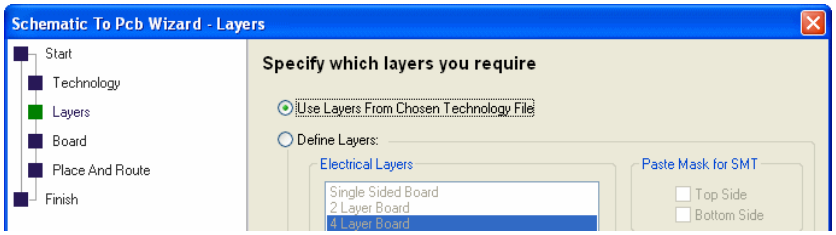
For our example, select **Choose Technology File:** (we want to use an existing one), and select **2sig2plane.ptf** from the drop down list.

For **Units:** select **thou** and **0** precision. If you prefer Metric units, choose mm and 3 as the precision.

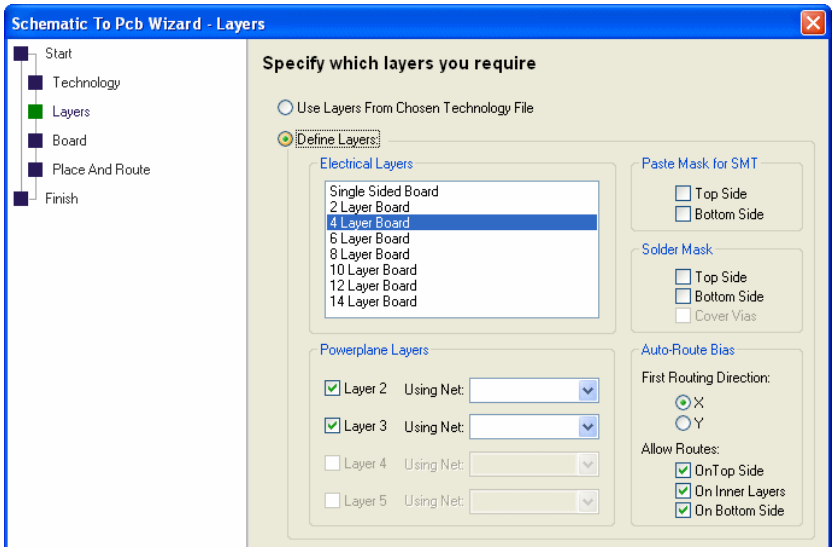


Click **Next >** to proceed.

The **Layers** page allows you to change the layer usage from that of the Technology file if you specified one. For our example, we will leave the selection as **Use Layers From Chosen Technology File**. The layers selection will display the contents of the Technology File but you cannot edit them, only view them. These can be changed once in the PCB design anyway, this dialog gives you visibility of the selection and the ability to make changes up-front.



If you decided to make changes to the Layers, the **Define Layers** button should be selected. Use this once you understand the implications of making changes to layers.



Leave the selection as **Use Layers From Chosen Technology File**.

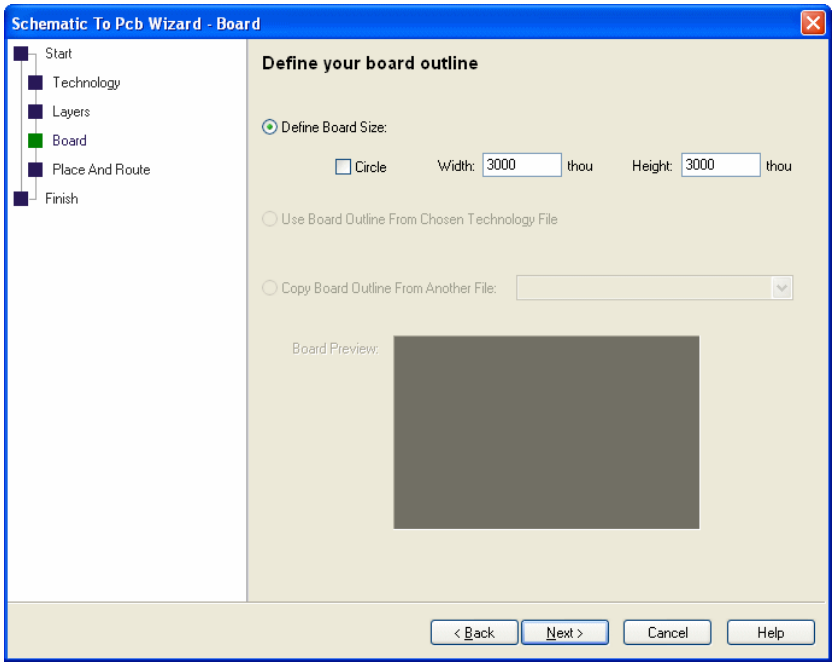
Click **Next >** to proceed.

You should now define the shape and size of the board outline using the **Board** page. You can **define** your board as a rectangle/square, or a circle and it's **Width/Height** or diameter in the current units.

If your chosen Technology file contains a board outline (which it can), you can elect to use this.

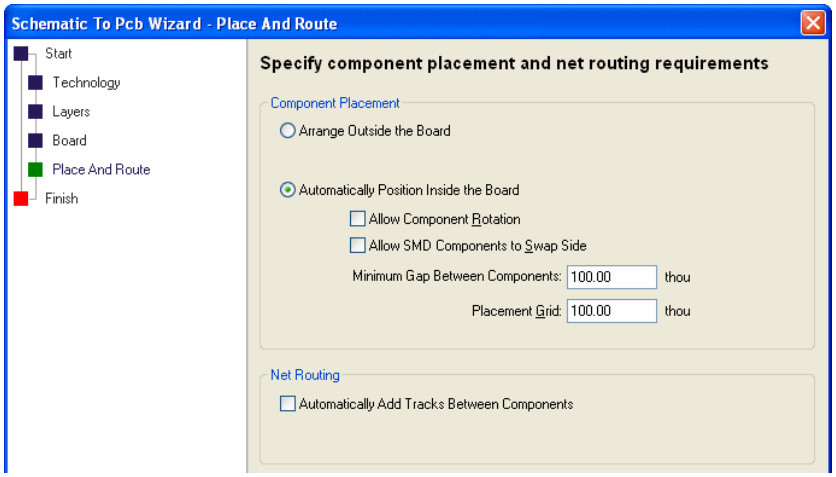
You can also choose to **Copy Board Outline From Another File** (another PCB design file or different Technology file). If using a PCB design for the board outline, the PCB file must reside in the Technology file folder.

For our example, choose **Define Board Size** and set the **Width** and **Height** to **3000** each.

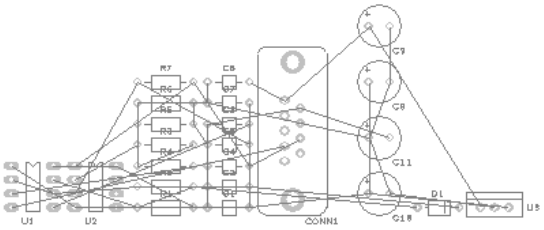


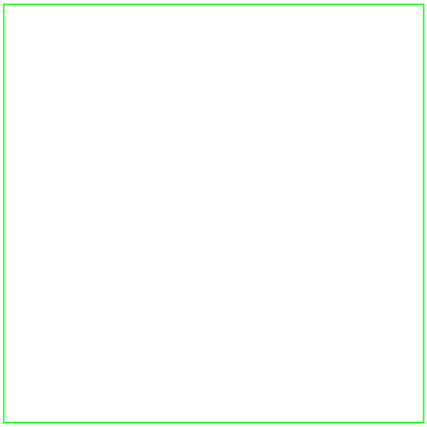
Click **Next >** to proceed.

The **Place And Route** page allows you to automatically let DesignSpark place components and then route their connections using the autorouter.

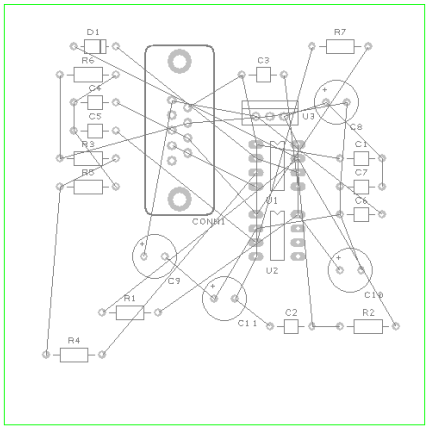


Under **Component Placement**, you can elect to place components in an **Arrangement Outside the Board** to stack them by type outside the board outline. This makes them easier to see and place manually.





Use **Automatically Position Inside the Board** to place the components on the board. It will attempt to work out the best way to arrange the components on the board to keep the total length of tracks as small as possible, without packing them too tightly together. The two controls will enable you to define how components are placed using your preference and possibly, your assembly or manufacturing processes.



For our example, change the **Minimum Gap Between Components** and **Component Grid** values to **100** (thou) each. We will allow plenty of space between the components. If you decide you'd like less space between each component, you can run the **Auto Place Components>** option from the

Tools menu once in the PCB editor, you don't need to go back through the New PCB Wizard. Before the design is routed, this can be an iterative process of trial and error.

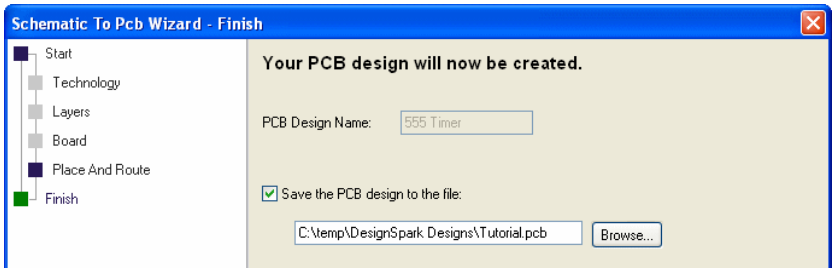
Don't forget also, even though the Auto Place option has made your placement for you, you can manually place components at any time to refine the placement by picking and dragging them (more of this later on).

Check the **Automatically Add Tracks Between Components** box to let the router convert the schematic netlist into PCB tracks. This is only available if you have chosen to Autoplace the components in the section above. Leave unchecked if you intend routing the board by hand, or need to alter the design before letting the autorouter finish it. You can run this option once in the PCB Design using the **Auto Route Nets>** option from the **Tools** menu.

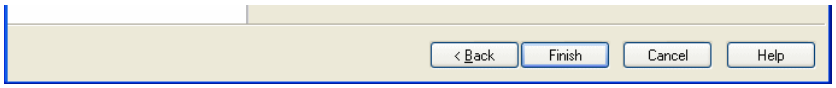
For our example, check this box to let the autorouter route the design for you (you can unroute it if you decide to have a go with the manual routing options).

Click **Next >** to proceed.

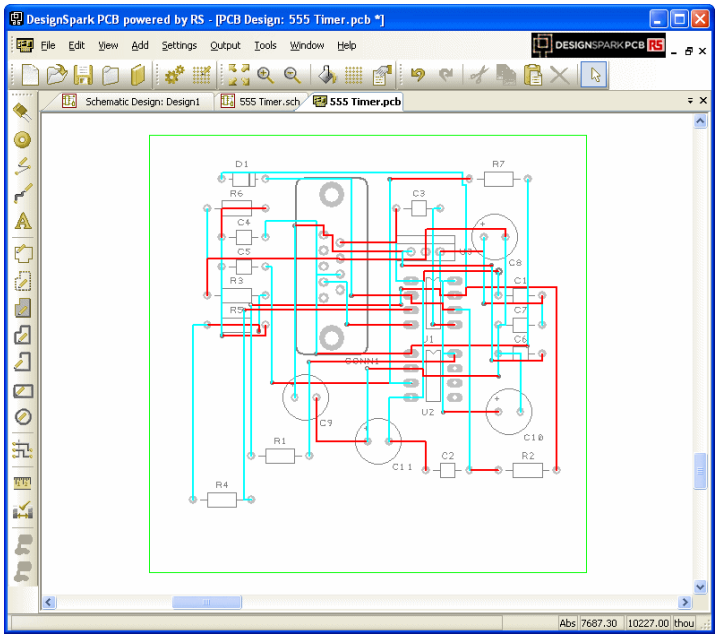
Use the **Finish** page to set your **PCB Design Name** and to **Save it to a file**. By default, the PCB design name will be the same as the Schematic design name. It doesn't have to be but it makes sense to keep them the same, especially when running options such as **Forward Design Changes** which will look for the opposite design type of the same name.



At any time (and before you hit the Finish button) you can go back to one of the previous pages by selecting the page name form the list displayed.

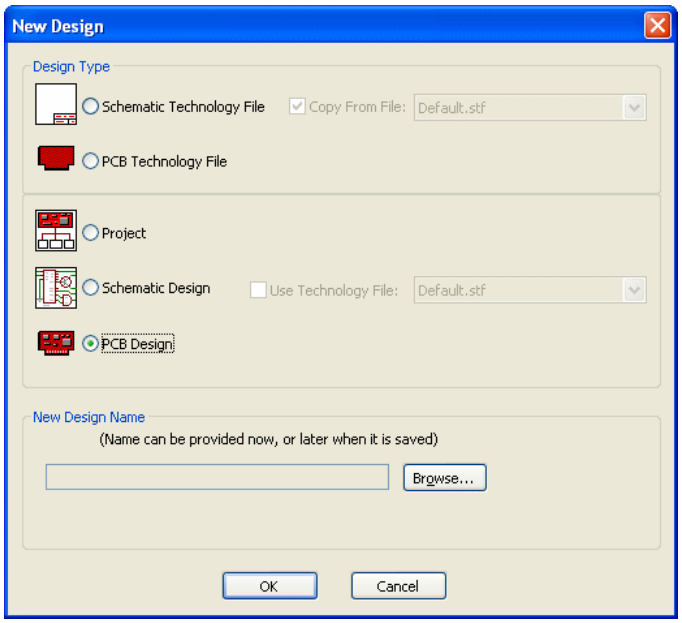


When you click the **Finish** button, the PCB design editor will run up and your PCB design will be ready for use. Your design background will be black, we've changed the colours for the picture so it is easier to view in the PDF file.



Starting a New PCB Design without a Schematic

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 **PCB Design** radio button.



Click the **OK** button and this will start the **New PCB** Wizard. This will run you through the wizard as described above.

Schematic Design Completed

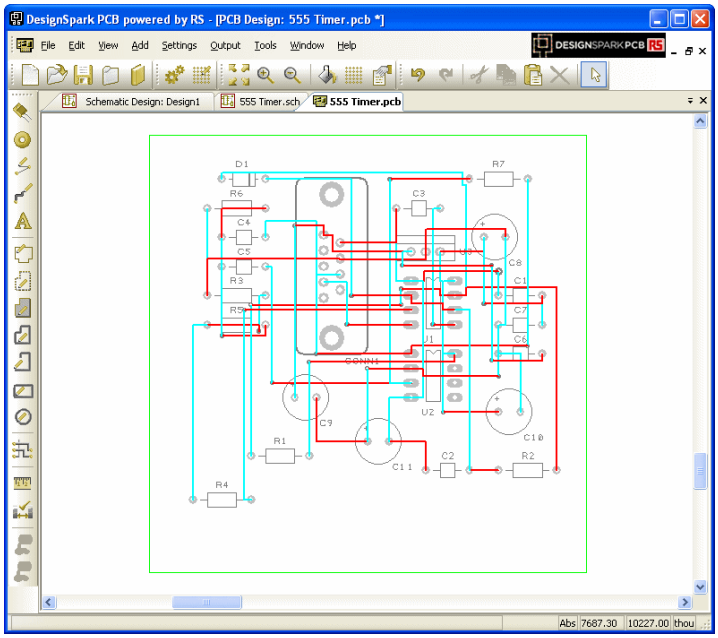
You have now completed the first section of this introductory tutorial so let's move on. Leave everything as it is and read on if you wish to continue with the PCB design tutorial.

Chapter 4. PCB Design Editor

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 **Translate To PCB** option and **PCB Wizard**. If you didn't do this, go back to the previous chapter (*Converting your Schematic to PCB*) and convert your schematic into a PCB design ready to carry on with this tutorial.

At this point you would normally be ready to start laying out the PCB design. For our example, we created the board outline, placed the components and routed the connections. This enabled you to see the full process working.



The board outline is displayed with the components placed (and routed), we are going to undo this and do it all manually.

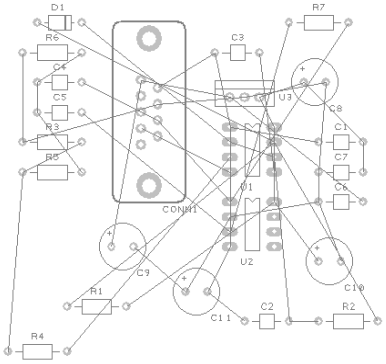
Unroute the Design

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

▶ Unroute the design

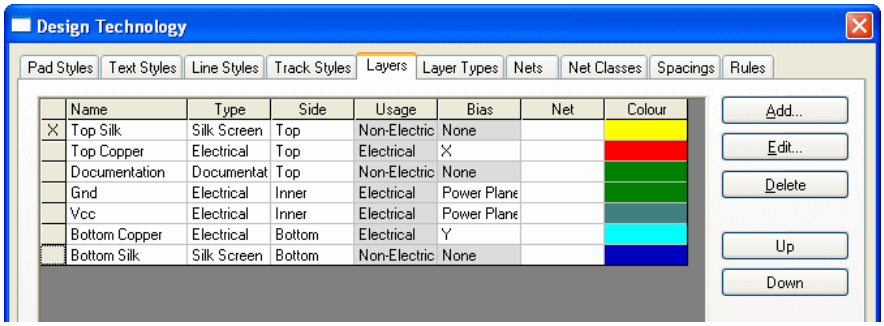
Select the **Unroute Nets> All Nets** option on the **Tools** menu to unroute the whole design removing all the tracks already routed. Three Unroute options available to control the amount of unrouting to perform – **All Nets**, **Browse** and **Selected Nets**.

Don't forget, once unrouted, you can always restore the routes using **Undo** <**Ctrl-Z>**.



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 **Design Technology** option, choose the **Layers** tab.



Be aware that reducing the number of layers using Delete could lead to information 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 our pre-defined shapes of rectangle or circle, this can also be easily achieved.

Deleting an existing board outline

DesignSpark 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 DesignSpark; 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. Four shape styles are provided: **Rectangle**, **Shape**, **Circle** and **Square**. 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, during the Schematic To PCB conversion process earlier we have specified the board outline as a square of 3000x3000 thou. If you require cutout shapes in your board, add these using the **Add Board Shape** option. DesignSpark understands that board outlines inside the main board outline will act as cutouts.

Importing a new board outline using a DXF file

Instead of drawing a board outline, you can also import a DXF file previously created in your mechanical CAD system, such as AutoCAD.

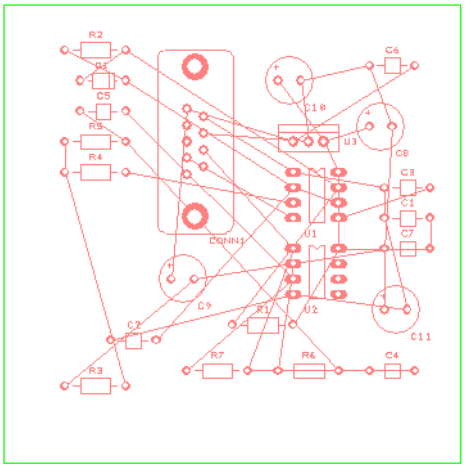
If you wish to import a DXF file, use the **Import** option on the **File** menu. You are presented with a dialog to map shapes and text in the DXF file into your DesignSpark layers. You can also change the import Units to adjust the import scale. More information about this option is available in the Online Help.

Placing Components

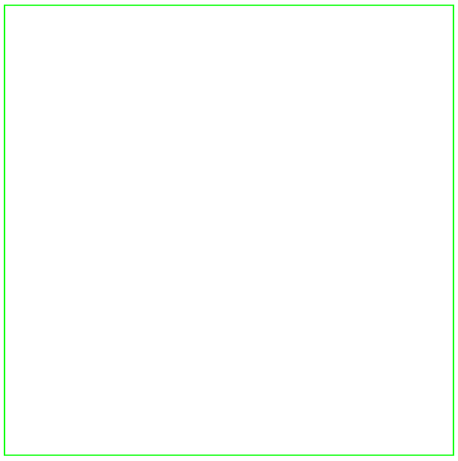
Placement can be an automatic or manually process. Automatic placement can be either during the conversion phase or by using the **Auto Place Components>** option from the **Tools** menu. Manual placement can be done at any point in the Design editor, even after using the automatic placement option.

We've started the design with the board outline and are now ready to try manually placing the components. Currently, our design has been placed during use of the New Board Wizard option. We will un-place the design first.

Drag a box around the components in the design to select them.



Now pick and drag them outside the board outline. Had you chosen the option **Arrange Outside the Board,** this would have stacked them neatly outside the board outline ready for manual placement. Once outside the board outline, release the mouse to place them. Click in free space to deselect the components.



▶ To place components

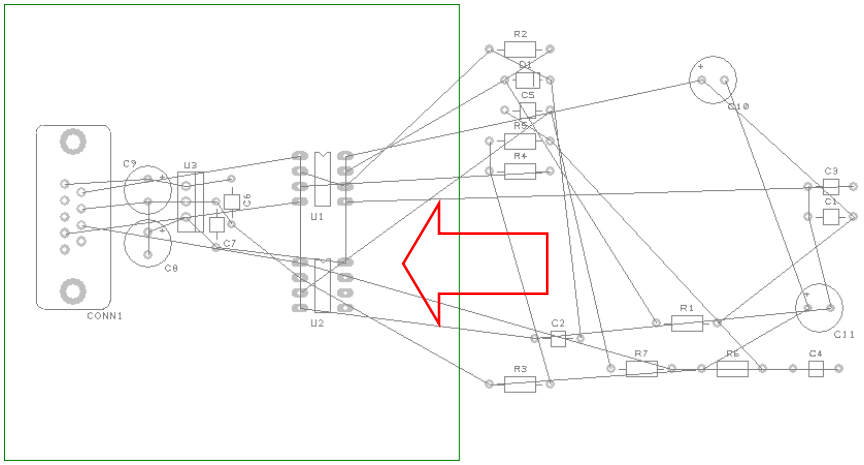
DesignSpark uses standard Microsoft Windows methodology throughout, moving or placing components is a simple case of **picking and dragging** the selected component. At any time during move, the drag may be cancelled by pressing the <**Esc>** key, or once released by using **Undo** <**Ctrl-Z>**.

Using the pick and drag method, place all the components to look like the picture below.

We suggest you place **CONN1**, **U1, U2** and **U3** first and place the other components around these. Placement on this design isn't critical but improve it if you feel you want to.

When placing **U3** we have rotated it to get it closer to the connector **CONN1**. We will also place the decoupling capacitors close to this device as they are pretty sensitive and are there to reduce noise of the voltage regulator.

Place capacitors **C6**, **C7**, **C8** and **C9** as close as you can to **U3**. To follow our example exactly as the picture below, you will need to rotate some of the components.

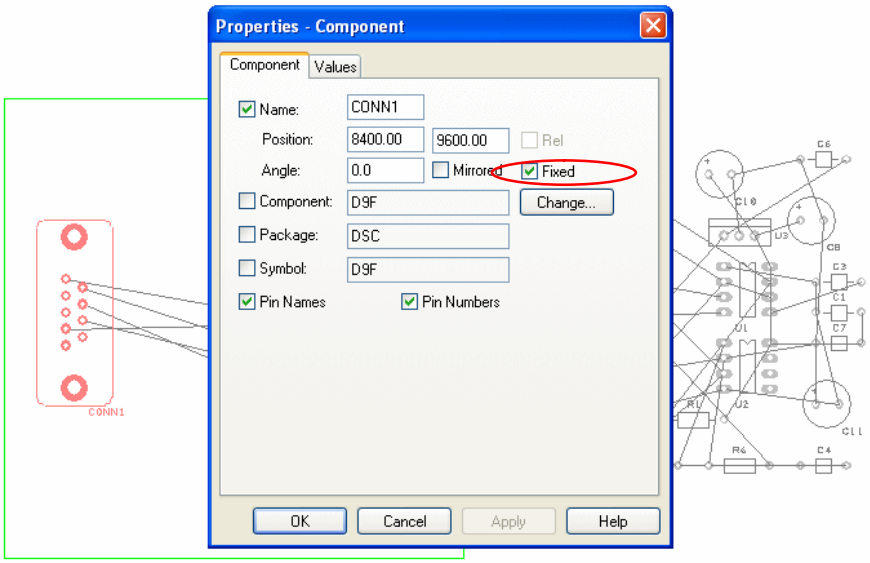


Once you have these components placed, we will fix them. On a real design, let's assume you might have some critical components which are placed specifically. You would then want to fix these so they cannot be moved accidentally.

▶ To fix components

To fix components, select the component and use **Properties** from the shortcut menu.

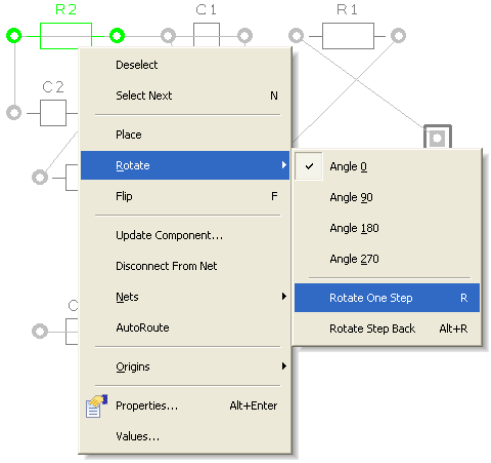
On the **Component** page, select the **Fixed** check box and press **OK**. This will fix the component.



The **Fix Item** option is also available on the shortcut menu for a selected component.

▶ To rotate components

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.



▶ To flip (mirror) components to the other side of the board

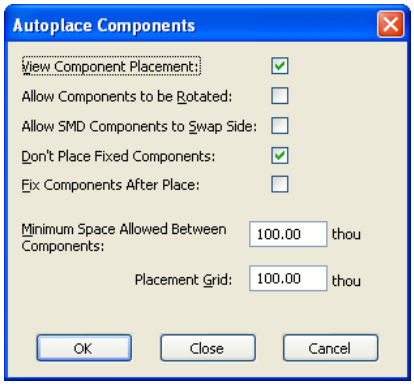
For our design, no mirroring (Flip) will be required. The placement and overall connection lengths could be improved by rotating some of the components. If you feel you would like the practice placing components, select a component and click <**R>** to rotate it. Component names can be rotated back to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

Once fully placed, the design should be routed.

▶ To use the autoplace option

We've placed some of the critical components and fixed them in-place. Now we will place the remaining components using the **Autoplace** option.

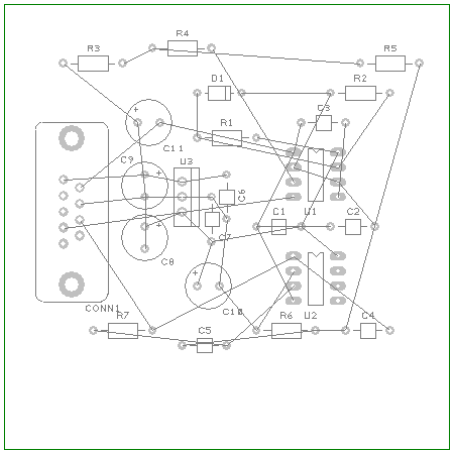
From the **Tools** menu, select **Auto Place Components>,** choose **All Components**.



From the selection available, we want to **View Component Placement** so check the box. Also check the **Don't Place Fixed Components** box, we want the fixed components to remain in-place.

Set the **Minimum Space Allowed Between Components** and **Placement Grid** values to **100** thou each. You can play about with these values if you like. Place the remaining components and use Undo if you wish to try a different placement pattern. Perhaps changing the Grid to 50 thou.

Press **OK** to make the placement. The remaining components are placed around the fixed ones in the space available.



▶ Save the design

Save the design using the **Save** option from the **File** menu or click the **Save** icon on the toolbar.

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.

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 it back to the 'rats nest' and use the manual routing tools to route it.

Automatic Routing

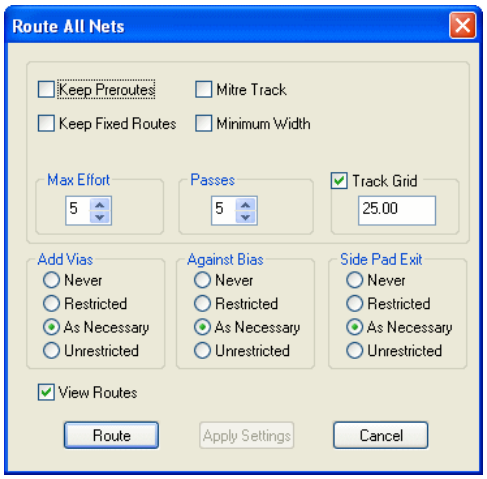
The **Autoroute** option is used to convert your nets into electrical tracks as you can with manual routing. However, on dense designs it can save you many hours or days of work. The autorouter can take an unrouted design and route it within minutes quite often.

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**.

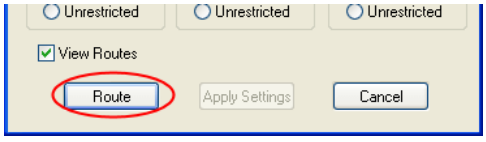
▶ To use the autorouter

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

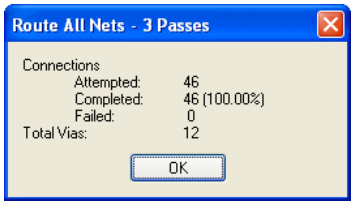
Leave the default settings as they are:



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

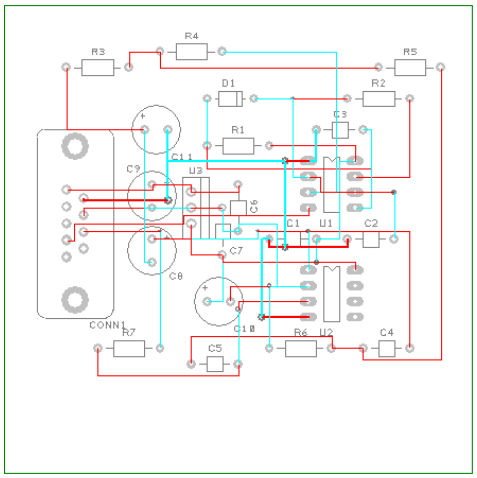


The results are quickly displayed:



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

Click the **Save** icon on the toolbar or **Save** from the **File** menu.

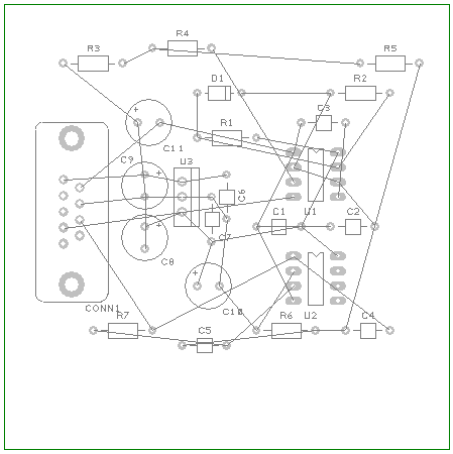
Unrouting the design

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

For our exercise, 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>**.



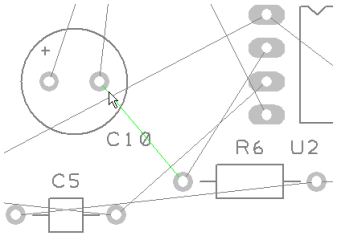
Manual Routing

DesignSpark 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.

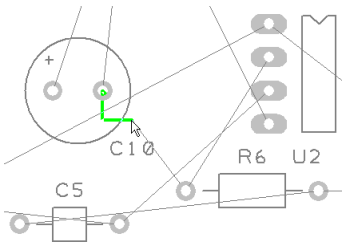
For our example design, we will now add the routes in by hand (manually).

▶ To start routing tracks manually

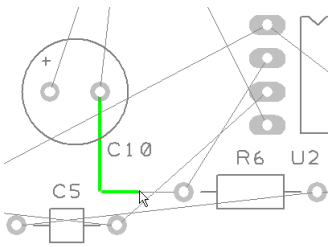
To start adding tracks, **double-click** on a connection, this is the quickest method to start routing, other methods are available.



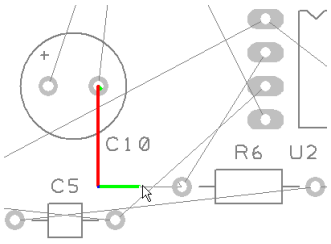
If you followed our design placement loosely, double-click the connection between C10 pin 2 and R6 pin 1.



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 (R6.1). In this mode, the track is placed orthogonally at 90 degrees with two floating segments.

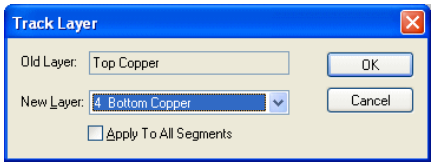


Move the cursor down to draw the vertical track segment. The track is still being placed with 2 segments and a 90 degree corner.

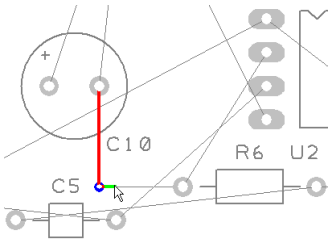


Click the mouse once, this places the first track segment (the straight track segment coming out of C10.2). This is temporarily 'locked' in place for this routing session. It can be moved or edited later on.

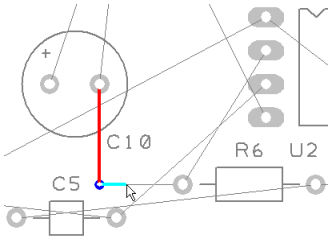
Now we will add a track layer change.



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.

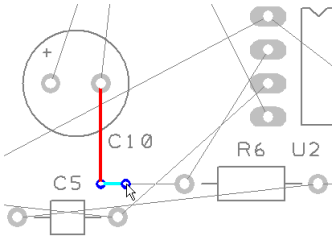


A via has been added automatically and the track layer is now on the Bottom side. This will be displayed on the Status bar at the bottom of the DesignSpark window.

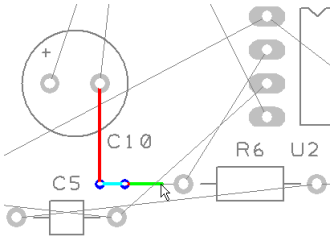


Move the cursor right and click again.

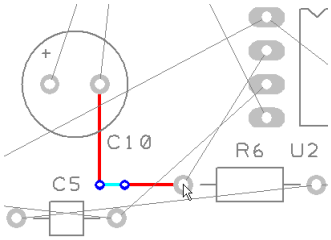
This will either be the next corner or the next location for a via for a layer change. Notice the track has changed color to show it is on a different layer.



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 R6.1. 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. Note the track colour is now red again, showing it is on the top layer.

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.

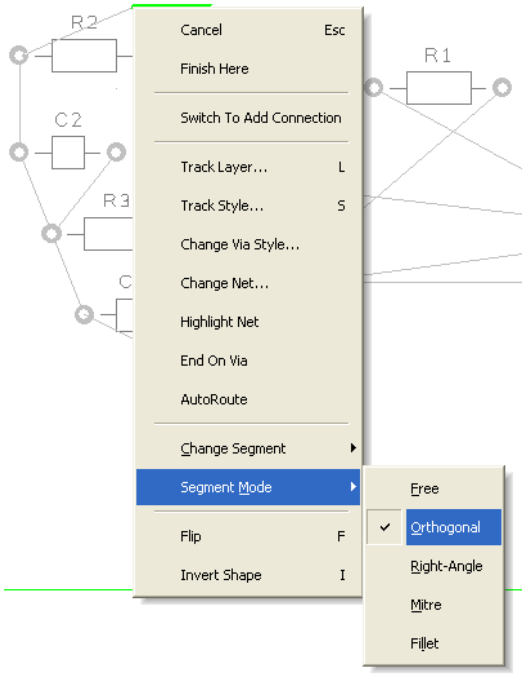
Summary of the basic modes of routing

During track editing, the basic commands for use are:

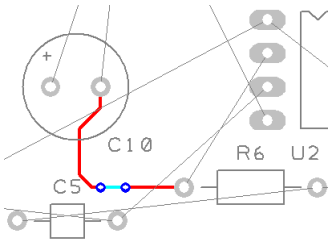
- **Single-Click**, once editing, this will insert a corner to change routing direction.
- **<L>** followed by <**Enter>** will enable a layer change to the opposite side of the design. This can be used during editing, or afterwards on selection of a track.

▶ 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).



during routing, this mode can be changed by accessing the routing mode. 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 DesignSpark 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 (0V) or +5V. 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.

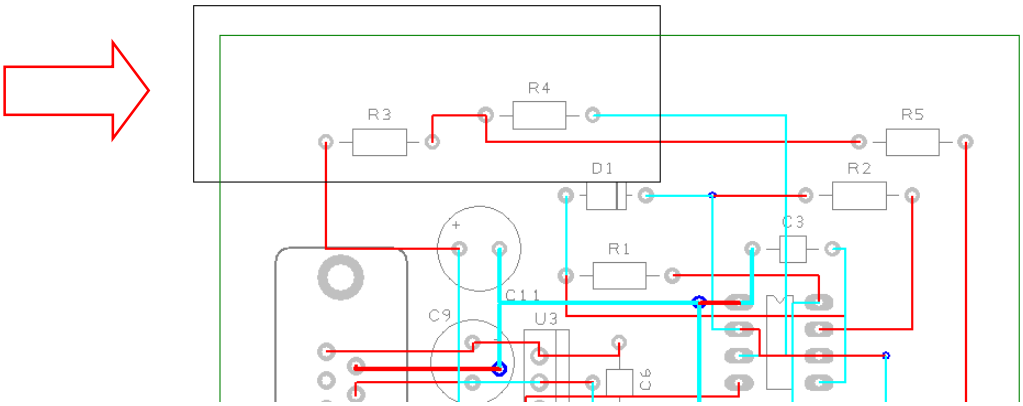
The principle of this facility is to insert a Copper Pour Area and 'pour' the copper into this area.

▶ 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. DesignSpark will pour the copper using the Spacing rules and rules defined for the Pour area.

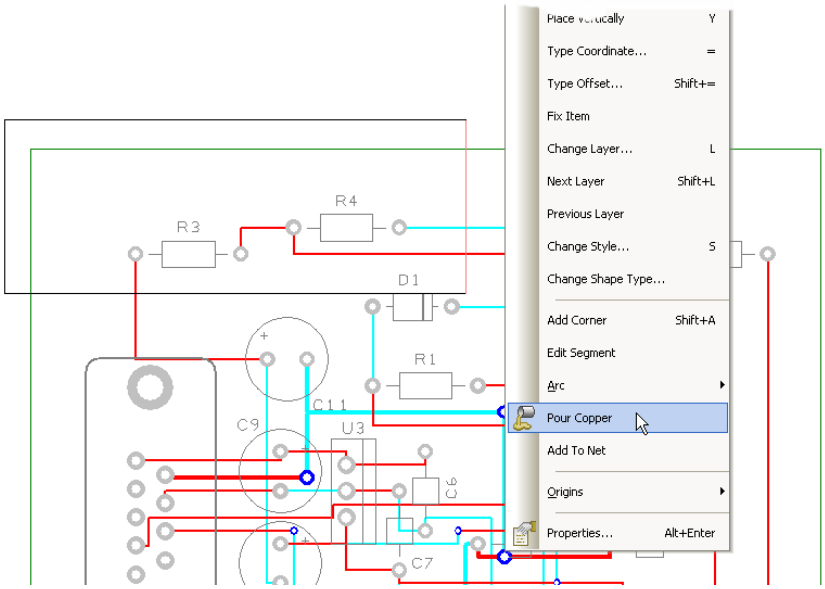
Following the picture below of our example, draw in a basic rectangle pour area shape, this is shown in black in the top left hand side of the design:



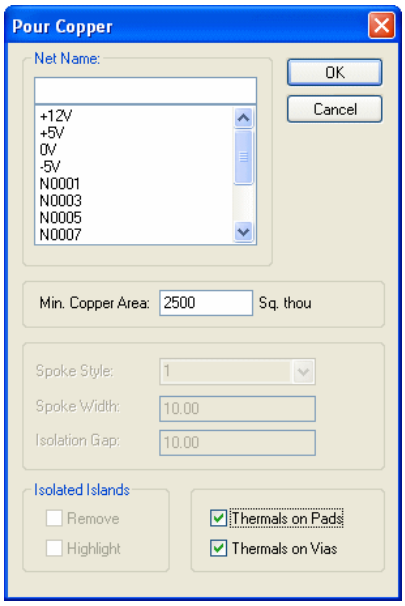
For this example you will see how easy it is to add a poured copper area.

Right click on the pour area.

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

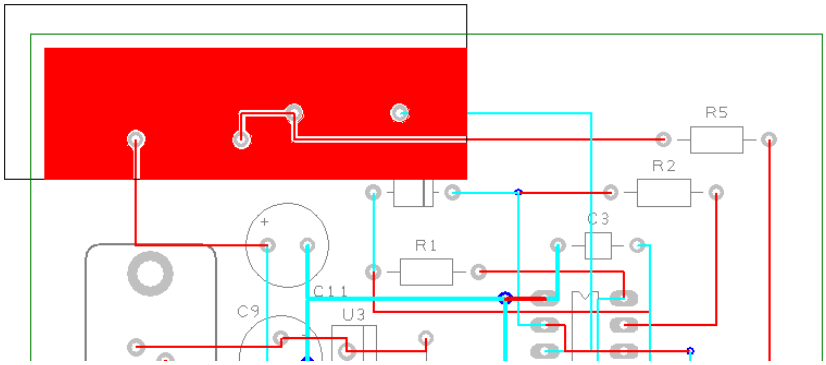


A dialog is displayed from where you can select parameters for the poured area, such as the net name association and thermal connections etc.

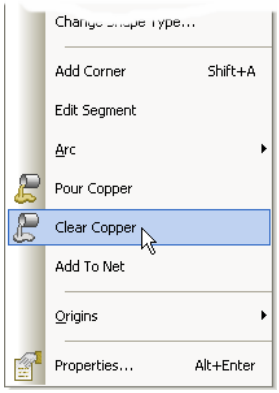


For this exercise, leave the **Net Name** blank and simply click **OK**.

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

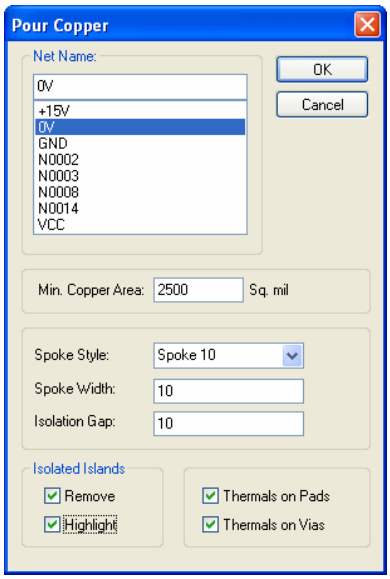


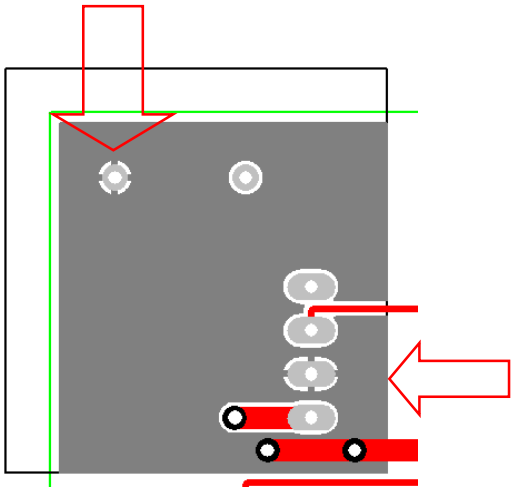
To remove a poured copper area (including any thermal spokes), simply select the area outline and select **Clear Copper** from the shortcut menu. This removes the copper but not the Copper Pour area.

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 **Spoke Style** check box settings for **Spoke Style**, **Spoke Width** and **Isolation Gap,** the style of how the copper is connected to pads can be defined.

In **Spoke Style:** type **Spoke 10**. Type **10** for **Spoke Width:** and **10** for the **Isolation Gap:.** Press **OK**. When requested, add the new style, Spoke 10 to the Technology.

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 **Schematics/PCB Check** option 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).

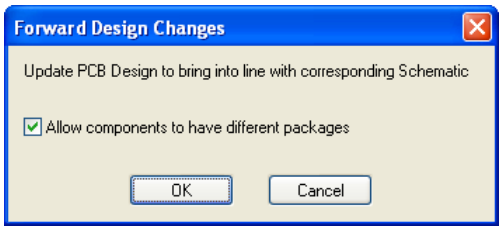
When run, if there are no differences, the following dialog is displayed:

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.

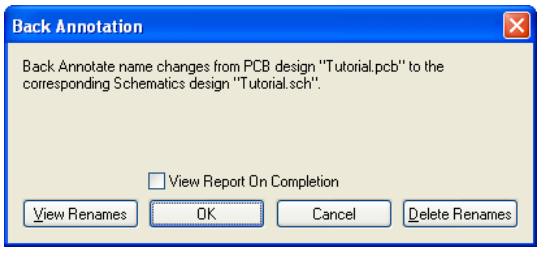
▶ To bring Schematic changes into the PCB

If at any time you want to 'push' Schematic changes into the PCB design, run the **Forward Design Changes** option from the **Tools** menu. This will run a comparison of the Schematic against the PCB and will only make the changes required to the PCB. It will report all the changes made.



▶ 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'.



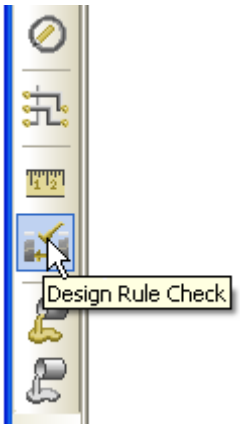
Design Rules Checking

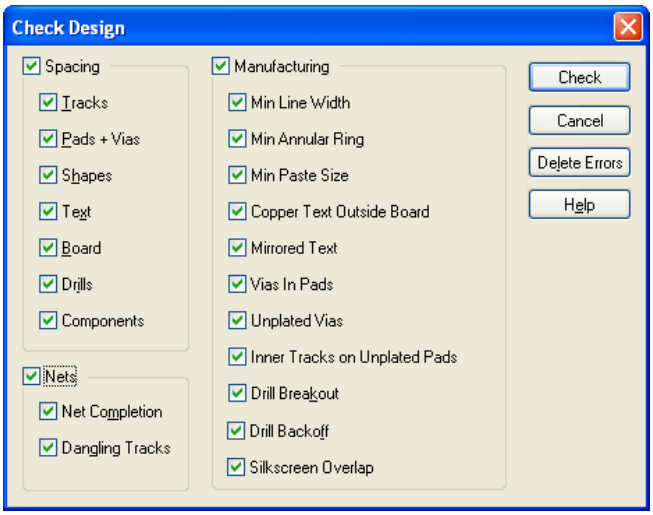
Before sending your design for manufacture, you must run the **Design Rules Check** option. This dialog contains check boxes that enable various checks to be made. The checks are made against the **Spacing** rules defined. This is also used to check various manufacturing rules have been obeyed.

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**.



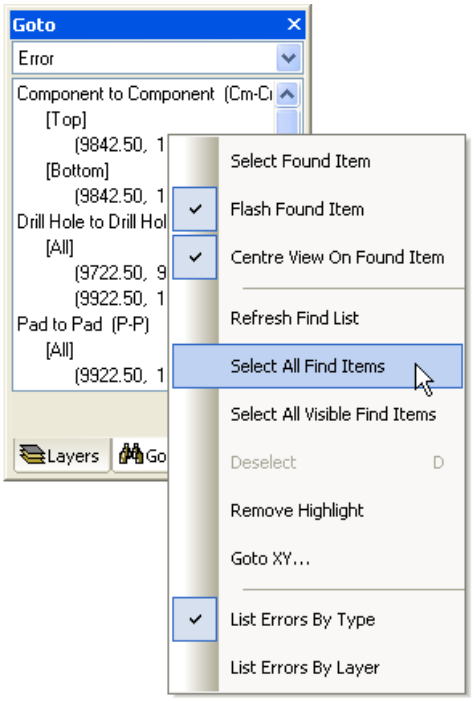


A dialog is displayed that allows you to control which items are included in the check. Click **Check** to run the DRC.

▶ 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.

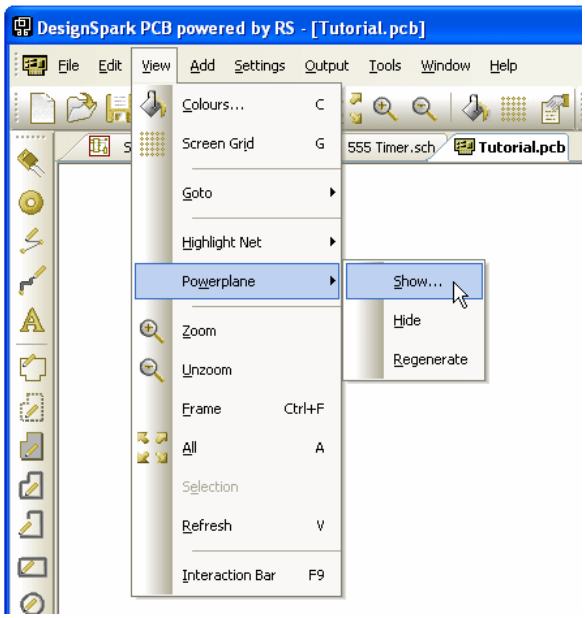
If at any time you right click on the Errors bar, you will see a shortcut menu displayed. From this you can choose various locate options to aid the finding of error markers in the design.

You can choose to list errors by **Type** or by **Layer** also on this menu.

Viewing Power Plane Layers

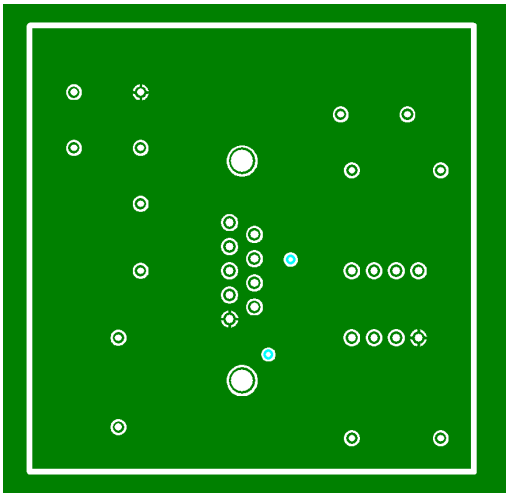
If you have included powerplane layers in your design on the inner layers, before you send off the PCB design for manufacture, it is worth running a visual check of the thermal connections. This visual check will validate any potentially missing thermal spokes where a pad to copper connection has not 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:



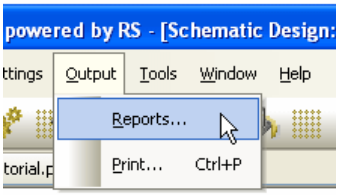
Once finished, select the **View** menu and use **Powerplane > Hide** to switch it off.

Reports, Part Lists and Net Lists

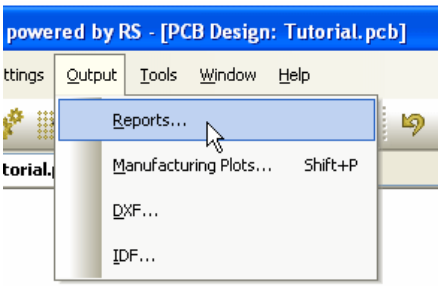
DesignSpark can generate reports where all vital aspects of the design can be output in detail. The reports also include a set of 'standard' reports like Component lists and net lists.

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

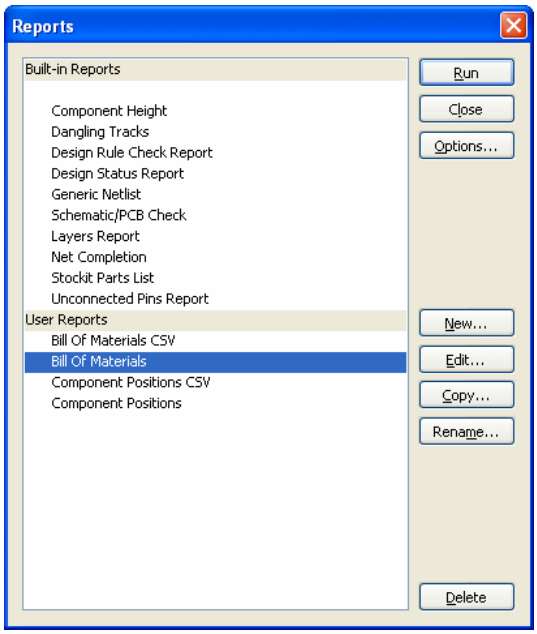
In Schematic editor:



In PCB design editor:



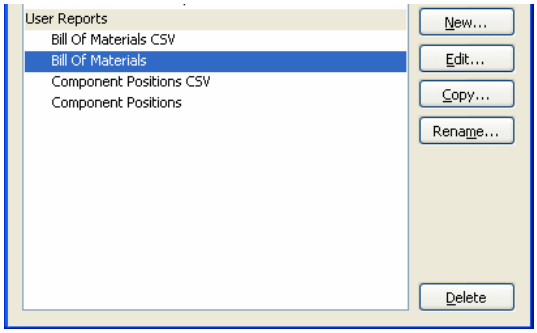
The **Reports** dialog is displayed. From here you can access standard reports under **Built-in Reports** and the custom report builder option under **User Reports**.



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

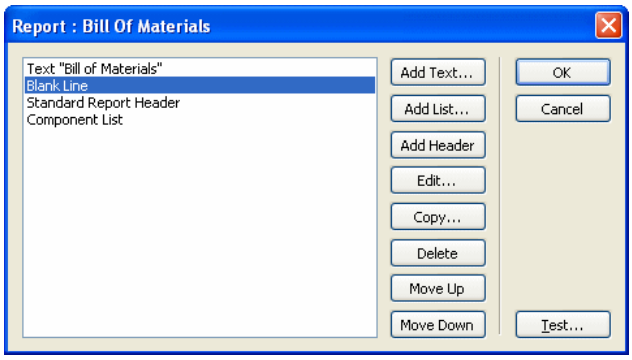
▶ To create custom reports

The bottom half of the **Reports** dialog is dedicated to the creation of custom reports.



Use the **New** and **edit** buttons to access the **Report Builder** option.

From within this dialog, you can add commands to construct your own report and format it in the way you require. Some basic reports and outputs have been created as examples for you to see.



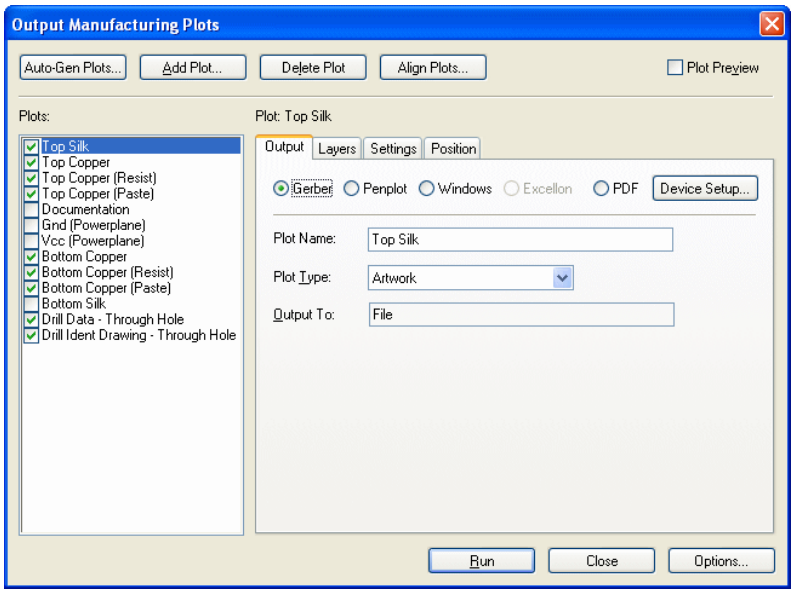
Chapter 5. Manufacturing the PCB

Plots and NC Drilling - Overview

DesignSpark provides you with a mechanism for plotting your designs in standard manufacturing formats, such as Gerber and Excellon NC drill. You can also print the design to Windows printers and PDF for documentation purposes or for checking.

The Plotting Dialog

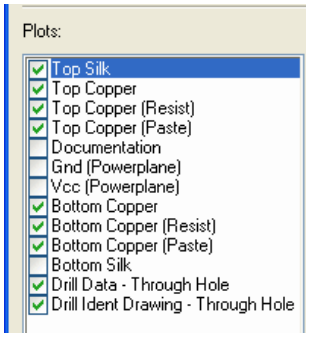
The **Manufacturing Plots** plotting dialog is accessed from the **Output** menu.



The dialog is split into three main areas:

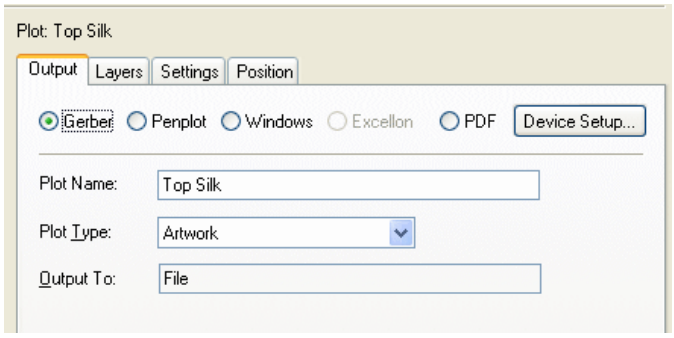
Plots

From here, choose the plots and the plot selection for output and plot name.



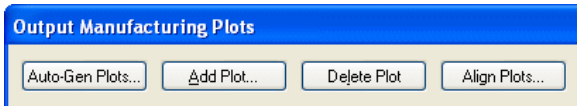
Output

From here, choose the output type, layer choice, plot contents (Settings) and plot scale.



Auto-plot generation

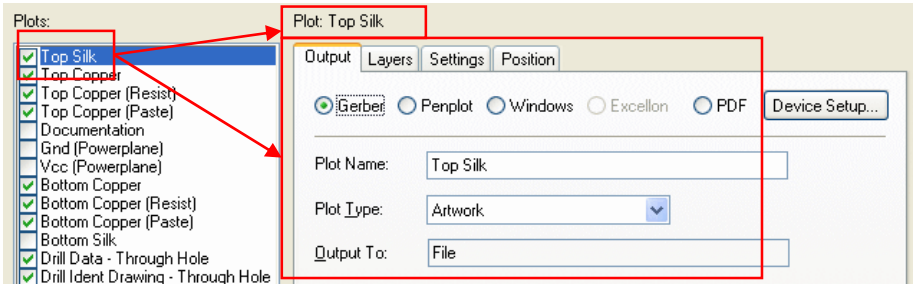
From here, choose to Auto-generate plots, add new plots to the set, delete plots and align plot contents with each other (position, scale etc.).



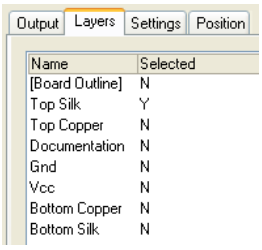
Using the dialog

For each plot selected in the **Plots:** list (Top Silk for example), the name is displayed above the **Output** pane. The contents of the **Outputs** pane, including the **Output**, **Layers**, **Settings** and **Position** tabs, are used to create the contents of the plot.

So in our example below, the **Top Silk** plot will be output to **Gerber** in an **Artwork** style and will be output to a file.

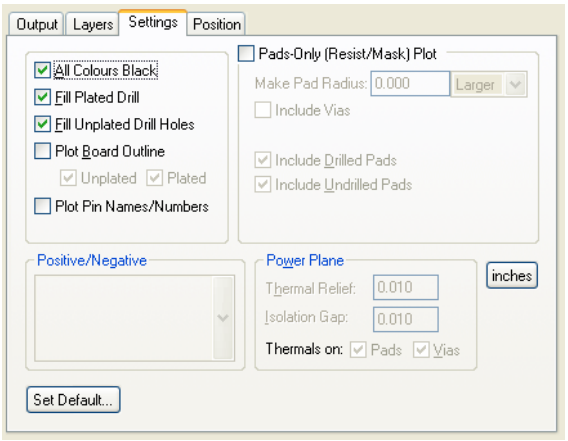


Still for the Top Silk plot, let's look at each of the four tabs:

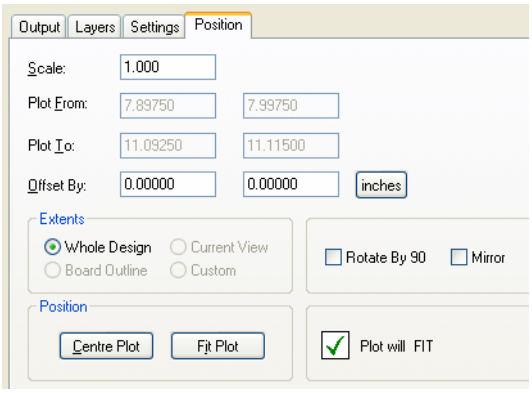


On **Layers**, choose the layer or layers (you can combine layers if you wish).

You can see that for our Top Silk plot, the Top Silk layer has been selected.

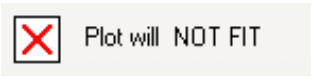


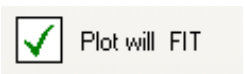
On the **Settings** page, the parameters here define the contents of each plot. It defines more specific items for inclusion and how they are plotted.



The **Position** tab allows you to define the scale and position of the plot.

A (non-editable) check box at the bottom of the dialog displays whether the plot will fit or not. If it doesn't fit, the box changes to a red cross.





The Plotting Process

To generate plots for manufacturing, you need to plot **Gerber** plots for the electrical and non-electrical layers and **Excellon NC** drill files for drilling the board.

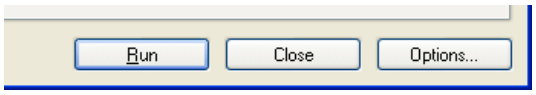
On opening the **Manufacturing Plots** dialog, you will need to first set up each plot required to create you an overall set of plots. For a new design, the plots will be automatically created for you based on the layers and layer classes in the design and intelligence built into the product.

Once a set of plots has been created, the next time you enter this dialog for the design, the settings will be remembered.

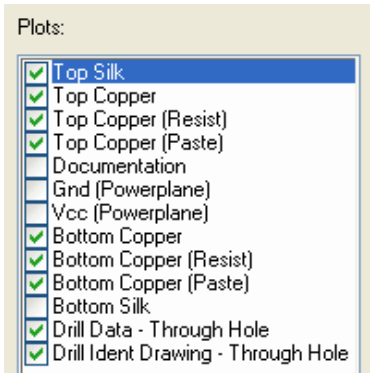
Creating plots

Once all the parameters have been set for each plot, you can output them.

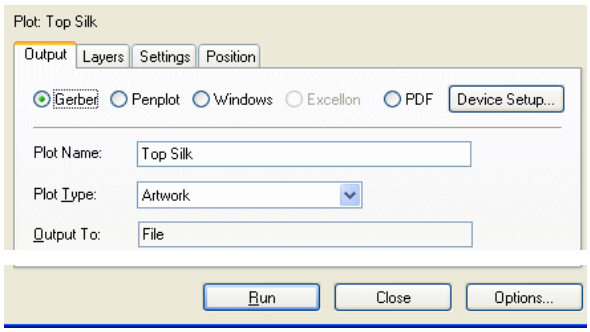
Use the **Run** button to create plots of all the plots in the **Plots:** list, the ones that have a 'tick' next to them.



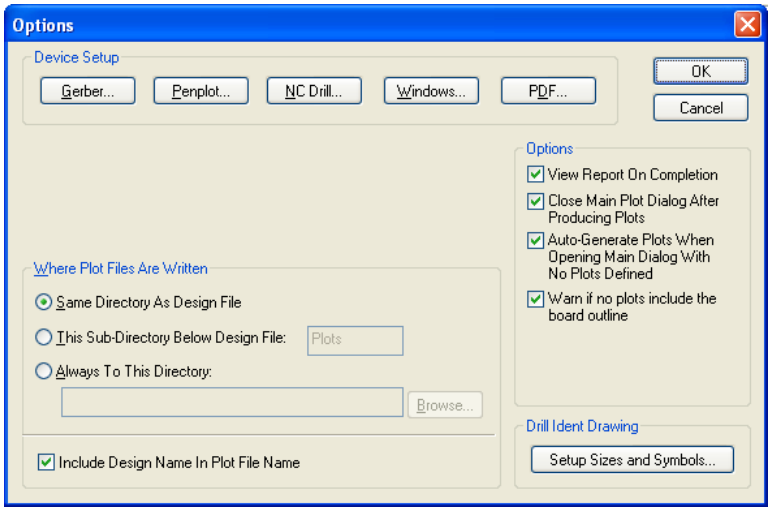
Plots with a tick are output:



For each plot, the **Output To:** selection will define where the output goes to.

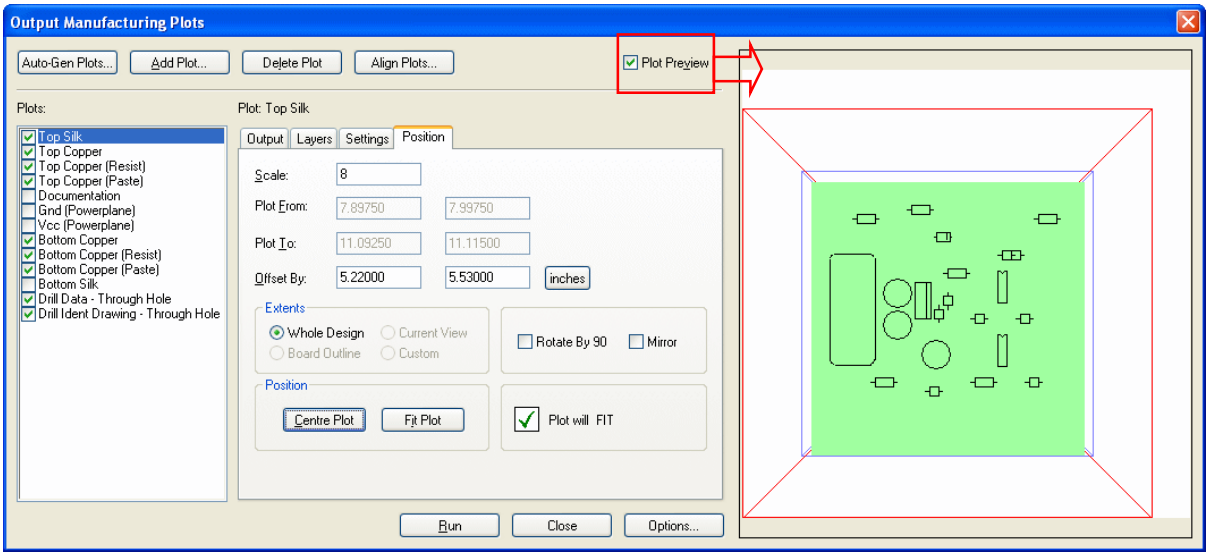


If you select the **Options** button on this page, your can direct the plot to a specific folder under **Where Plot Files Are Written**: By default, the files are written to the same directory as the design file. Another option is to specify the **Plots** sub- directory under the design directory.



Previewing the plots

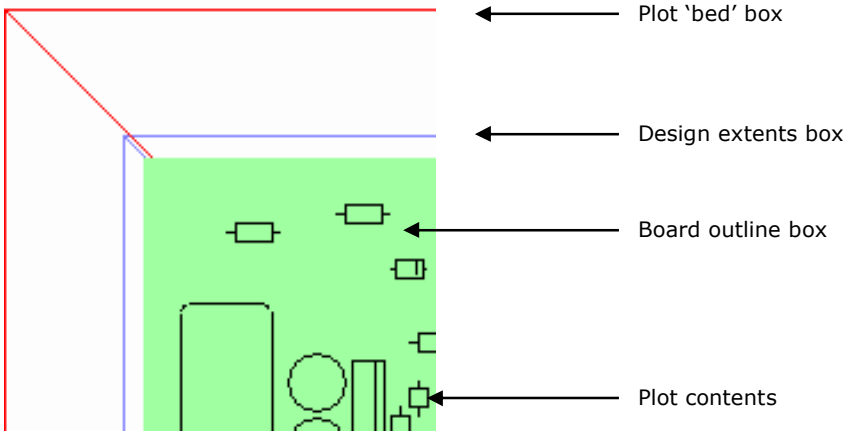
If you wish to view the plot to see it's position, size, contents etc. check the **Plot Preview** check box. A Preview pane on the right side of the plots dialog is displayed.



Within the Preview box are three main boxes:

Red box – this is the plot 'bed' or paper size. This will be the maximum size (minus any hardware edges) that the plot can occupy when fitted to the page.

Blue box – These are the design extents. If for example, you have a drawing border with a board outline inside it, you will see the box box is the same sieze as the drawing border.

Green box – This is the board outline.

Contents – the contents of the plot will be displayed inside the blue or green boxes. These are approximate contents to give you a quick preview of the final contents. For example, it will clearly show you a silkscreen outlines plot rather than a plot being a power plane.