



Guide & Tutorial

DesignSpark PCB PRO

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. 1997-2021. All rights reserved. E&OE

Issue date: 25/01/21 iss 3

RS Components Ltd

Birchington Road

Corby

Northants

NN17 9RS

United Kingdom

Tel: +44 (0) 1536 444215

Contents

CHAPTER 1. GETTING STARTED	4
Introduction to DesignSpark PRO.....	4
System Requirements.....	4
Installation.....	4
Introduction to the Tutorial	4
Additional Help and Information.....	4
Starting DesignSpark PRO	5
The DesignSpark PRO Desktop.....	5
Toolbars.....	6
Customising DesignSpark PRO	6
Commands and Shortcuts.....	6
Cancelling Commands	6
Units	6
Grids.....	7
Shortcut Menus	8
Undo/Redo	8
Properties of items.....	8
Status Bar	9
Design tooltips	9
Dockable Windows and Browsers.....	9
CHAPTER 2. CREATING THE SCHEMATIC	10
Schematic Design Tutorial	10
Starting a New Schematic.....	10
Adding Components.....	11
Placing Components.....	19
Adding Connections	19
Adding Power & Ground Symbols	20
Adding Connectors.....	22
Naming Connections	23
Translating the Schematic to PCB.....	25
Schematic Design Completed	25
CHAPTER 3. PCB DESIGN EDITOR	26
Getting Started with the Design.....	26
Translate to PCB.....	26
Setting up a PCB Technology	26
Layers.....	26
Creating a Board Outline Interactively	27
Using the Component Bin	29
Moving Components	30
Routing the Design	31
Picking in tight areas	33
Unrouting Tracks.....	33
Using the Autoroute Option	33
Pouring Copper into areas.....	34
Adding Dimensions	37
Checking the Design Integrity	37
Design Rules Checking.....	38
Reports, Part Lists and Net Lists.....	40
Plotting the Design	40
Concluding The Tutorial.....	42

Chapter 1. Getting Started

Introduction to DesignSpark PRO

Welcome to the DesignSpark PRO Schematic and PCB design system. The system is built on an integrated design environment providing all the tools required to capture a schematic, output it to Spice simulation and pass it through to the PCB design environment for the printed circuit boards (PCBs). The PCB design process is aided with a shape-based autorouter. Outputs to industry

System Requirements

The table below shows the hardware requirements for running DesignSpark PRO. It shows both the minimum and the recommended specification. It should be said that this product will run pretty much on any off-the-shelf PC available in most computer stores.

	Recommendation
Processor	Pentium i5 or better, such as i7
RAM	2048MB or more
Hard disk space	650Mb for product
Operating System	Windows 10
Graphics	Better than 1024x768
Mouse	Wheel Mouse

* MAC O/S or Linux are not supported. All supported operating systems are supported in both 32 and 64 bit formats. Please visit our web site for the latest news on supported operating systems.

Installation

It is assumed that you have successfully installed the DesignSpark PRO product or demo onto your computer. If you haven't, then use the product CD or a download from the numberone.com web site. The installation is straight forward using the wizard provided. Follow the instructions and use the default settings provided. The tutorial designs can be run even using DesignSpark PRO in demo mode if the demonstration version is being used. Note, the demo version and full product are not the same, the demo cannot be 'converted' to a full version, it requires a new installation.

Introduction to the Tutorial

You can quickly familiarise yourself with the DesignSpark PRO 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 to PCB design cycle. This tutorial can be used with both the full purchased product and the trial alike.

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 PRO has outputs to Gerber 274-D and 274-X (with the embedded apertures), NC Drill, pen plotter, full Windows printer support, PDF, ODB++, Bill-of-materials, netlists, DXF and IDF formats. Assembly machine outputs for pick & place machines and suchlike can also be output.

Additional Help and Information

There are many more exciting aspects to DesignSpark PRO which are not covered in this document. Further information can be obtained in the Online Help by pressing the **<F1>** key on your keyboard at any time, or by reading the DesignSpark PRO Users Guide. You can also obtain additional information by viewing the FAQs on the DesignSpark web site.

Starting DesignSpark PRO

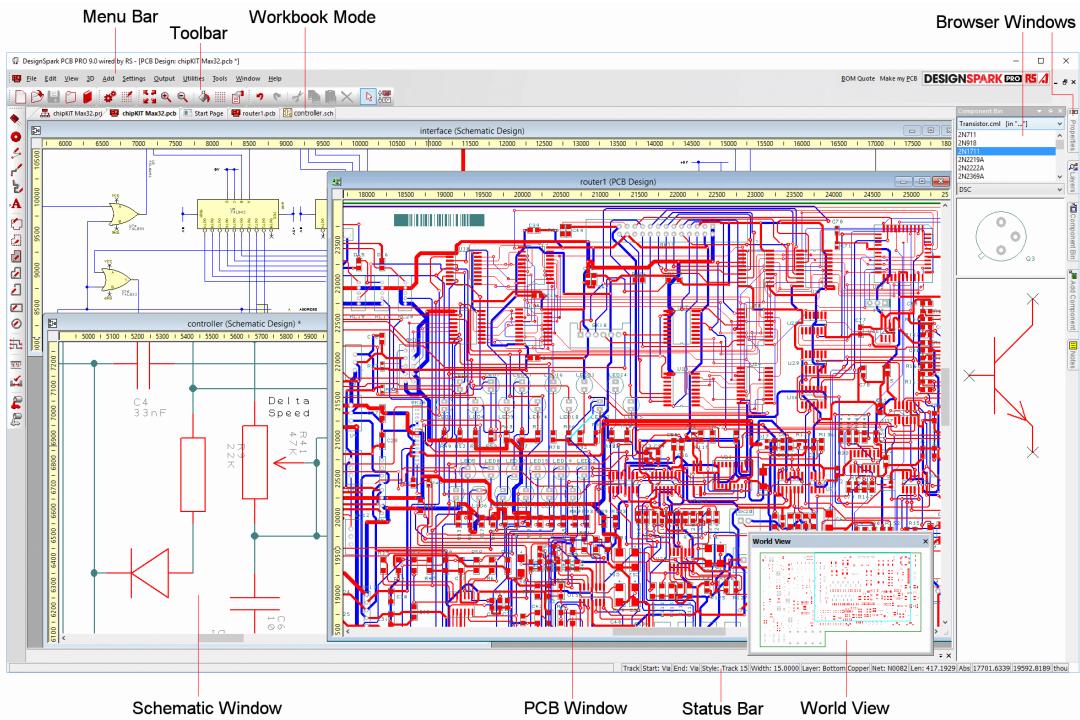
During installation, an DesignSpark PRO program group added to the **Start** menu, under **Programs** then **DesignSpark>**.

Select the DesignSpark PRO menu item to start the program.



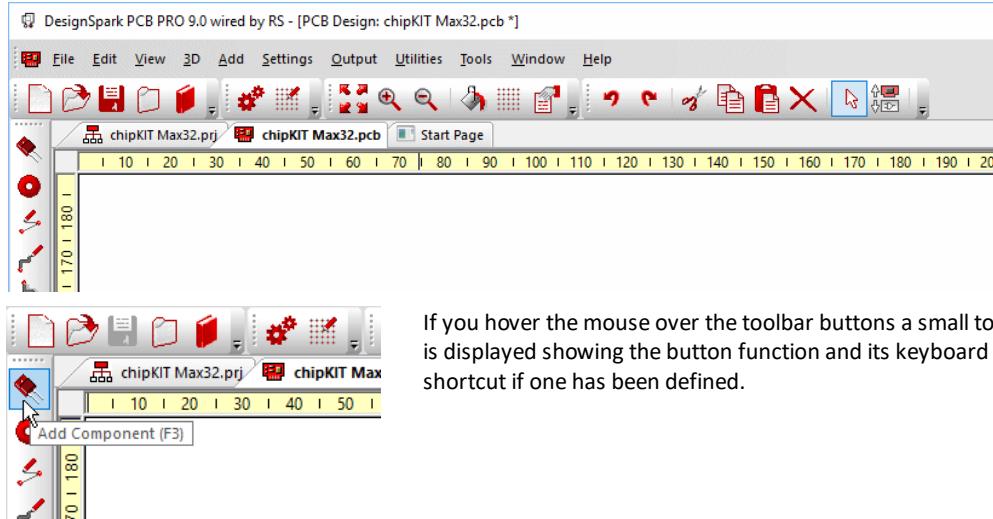
The DesignSpark PRO Desktop

When you run DesignSpark PRO, the main application window appears. You can open any number and combination of different designs and libraries together in this. DesignSpark PRO is broken down into many facets as are most Windows applications. 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 PRO is installed with a set of commonly used toolbars for you to use. These tools can be fully customised to suit your own design practice once you become familiar with the product.



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.

Customising DesignSpark PRO

DesignSpark PRO provides you with the ability to customise the user interface with your own **Toolbars**, **menus** and **keyboard** commands. You also have a complete command set of all the options within DesignSpark PRO which can be assigned to a shortcut key on the keyboard. Other aspects of DesignSpark PRO can be customised using the **Customise** option on the **Settings** menu.

Commands and Shortcuts

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

When DesignSpark PRO is first installed, you will find a set of shortcuts already defined for you. These are only a default set and can be modified to your own preference through the **Keyboard** tab on the **Customise** dialog. Almost all of the keys on a standard keyboard can be utilised giving you extensive coverage.

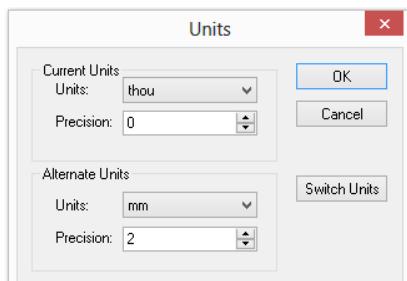
A full report of all the assigned keys can be obtained from this dialog using the **Shortcut Report** button on the **Keyboard** tab.

Cancelling Commands

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

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 between **Imperial** and **Metric** design units. You can quickly toggle between Metric and Imperial units by using the **Switch Units** button on this dialog or by clicking the shortcut **Shift-I** when working in the design.

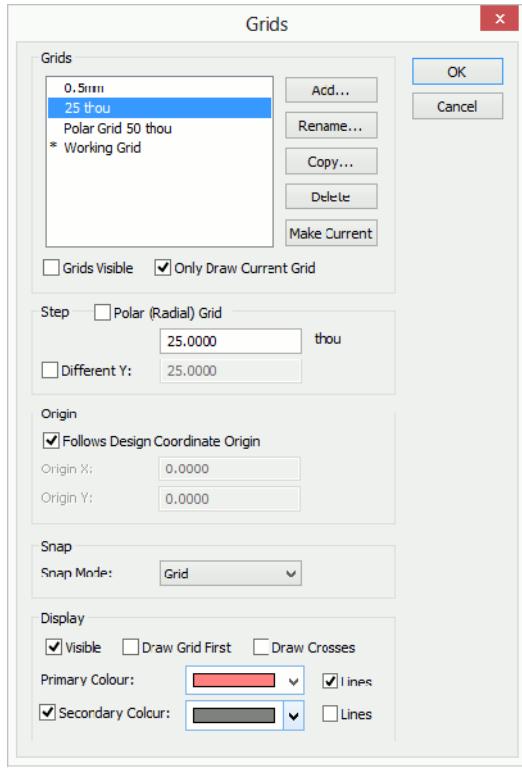


The design units are stored in an internal database with a precision of 1/100 micron regardless of what the unit precision is set to.

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 PRO 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 can contain any number of named grids:



The **Grids** area will allow you to define the grid used for placement of design items during manual interaction.

Grids can be defined as regular X Y grids or polar (radial) grids.

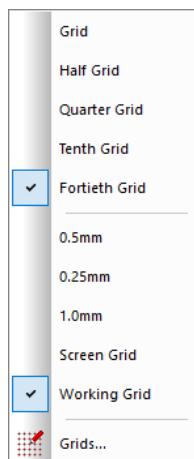
They can be referenced relative to the **Coordinate Origin**.

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>.

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

Snapping to Grid

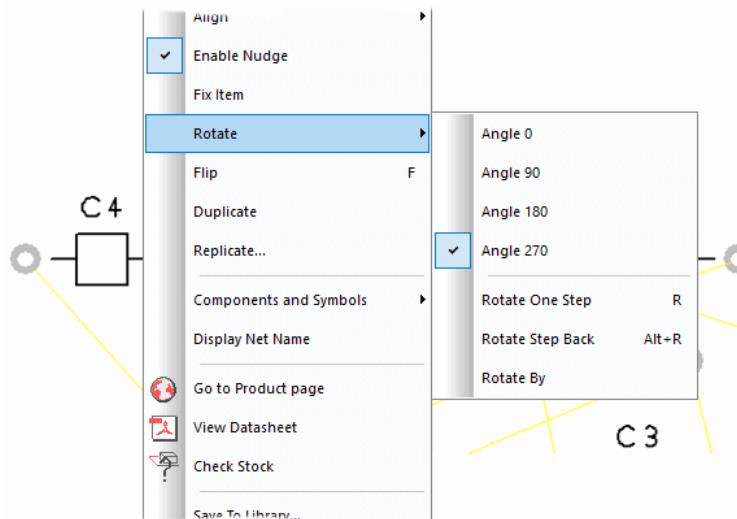
The **Snap Mode** shortcut can be displayed during editing by selecting the direct keyboard command <Ctrl-G>. By selecting the snap mode setting you can use predefined multiples of the Working Grid to move items for example.



Shortcut Menus

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

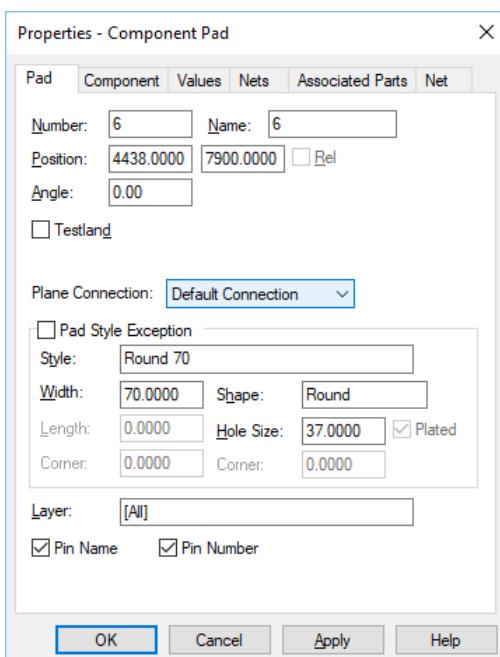


Undo/Redo

DesignSpark PRO contains unlimited multi-level Undo and Redo capabilities for use through the product. These are invoked using the standard Windows shortcut keys **Ctrl-Z** and **Ctrl-Y** respectively, these are also available on the **Edit** menu and as **toolbar** icons.

Properties of items

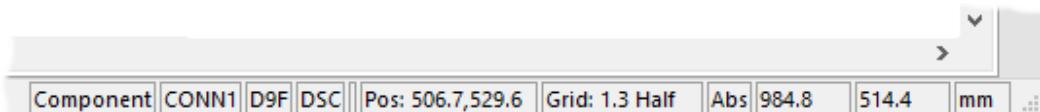
Each **item** in the design contains a set of characteristics that can be viewed or edited using the **Properties** option. Some items within the design are made up of several entities, each of which is individually selectable and has its own properties that can be viewed, components for example.



Design 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 Pad properties of a component pad.

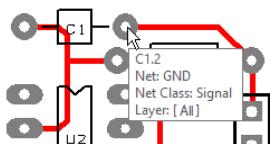
Status Bar

The status bar at the bottom of the design window displays useful information about the design item selected. When using an option, like track editing, it also provides you with a command prompt to show you what action is expected. More detailed information is also available by selecting the design item and using the **Properties** dialog.



Design tooltips

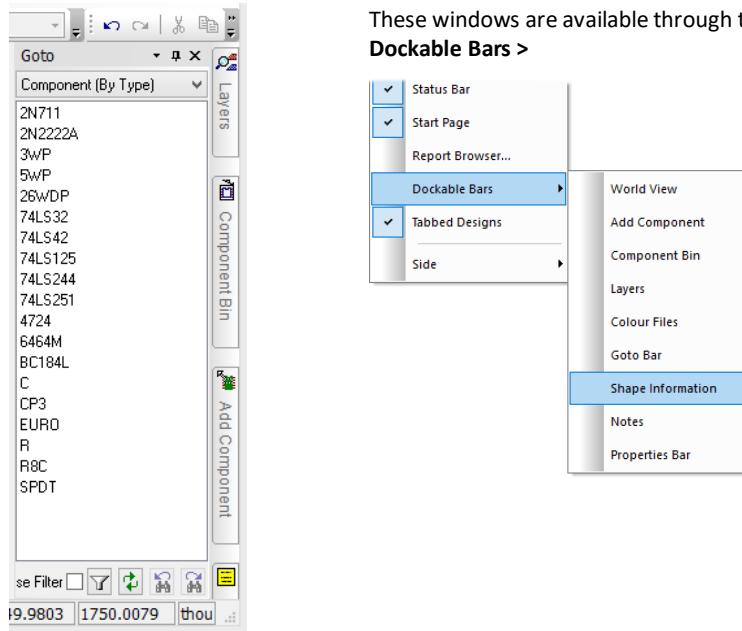
As well as the status bar information and Properties dialog, by hovering the mouse over a design item, information about that item will be instantly revealed on a design tooltip.



Dockable Windows and Browsers

As part of the design suite, you have a number of dockable windows and browsers that can be used.

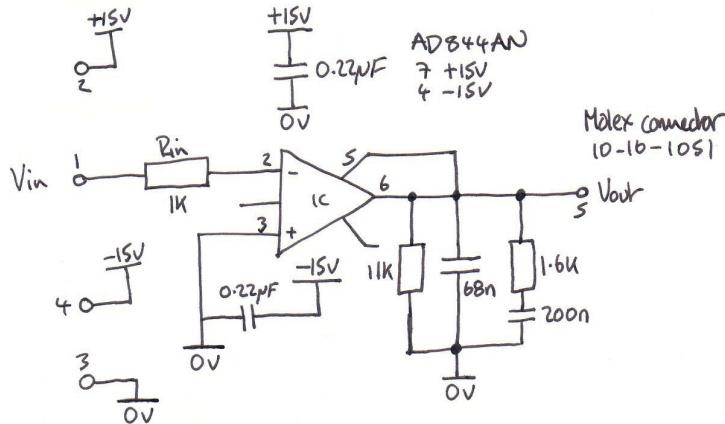
Each individual window can be specified to **Allow Docking** (enabled through the shortcut menu for each window or browser).



Chapter 2. Creating The Schematic

Schematic Design Tutorial

During the tutorial we will create a Schematic design, this is how the engineer ‘conceived’ it:

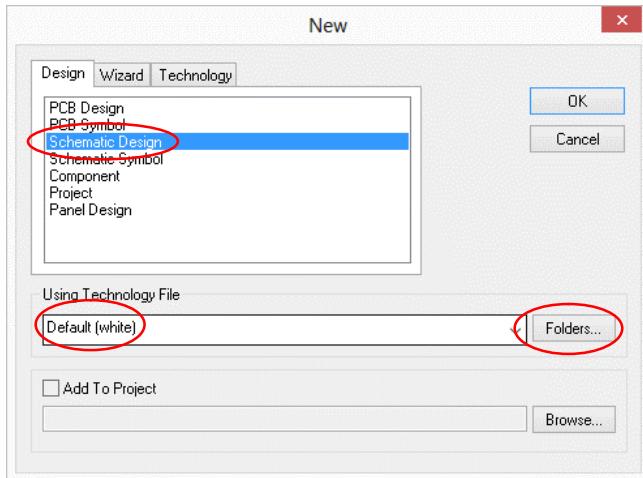


Obviously, once the sketch has been drawn properly as a DesignSpark PRO Schematic it will look quite a bit different.

Starting a New Schematic

► To start a new Schematic design

On the File menu, click **New** and **Schematic Design** from the **Design** tab.

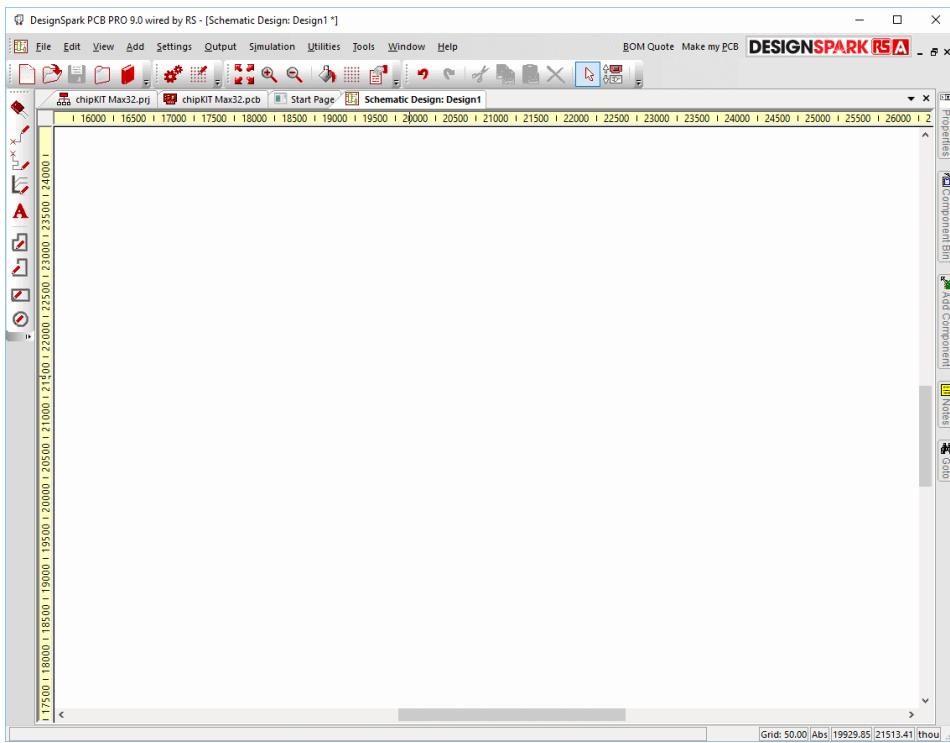


Under **Using Technology File**, for the **Name**: click on the small 'down' arrow to reveal the drop down list, select **Default.stf** More detail about Technology files is available in the Online Help.

Leave the **Add To Project** box unchecked. We will create a simple single schematic sheet without a project for this exercise.

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

Your DesignSpark PRO window will look like this, you'll notice there is already a design prepared for you to view in program called ChipKIT Max32.prj



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

Adding Components

You can use two methods to add components; by using the **Add Component** option and using the **Add Component Browser**. For this tutorial, we will use the **Add Component Browser**.

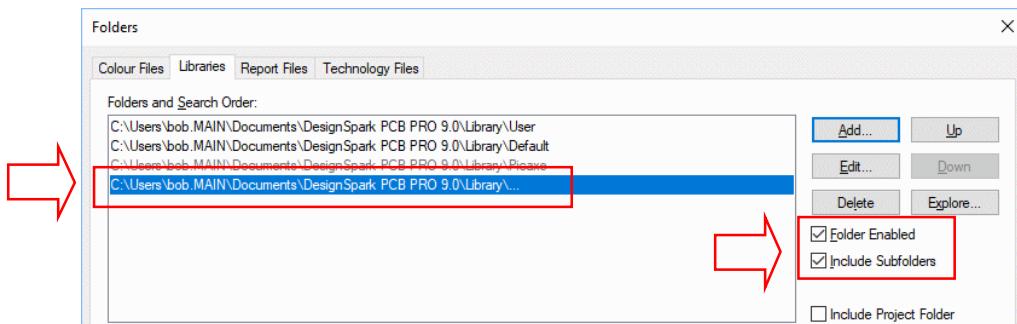
For our tutorial we will enable more libraries. For simplicity, the standard DesignSpark PRO installation only enables a small selection of libraries to start with.

► To enable more libraries

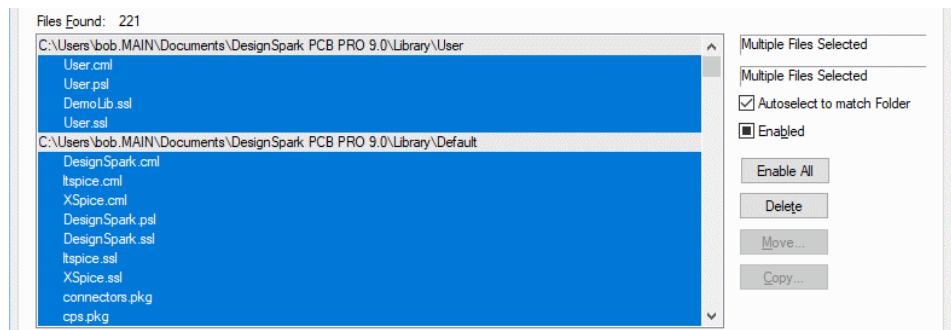
From the **Setup** menu, choose **Folders**. Click on the **Libraries** tab.

You need to enable the folder at the bottom of the **Folders** list - \Library

It is greyed out currently (not enabled), select it and press the **Folder Enabled** check box.



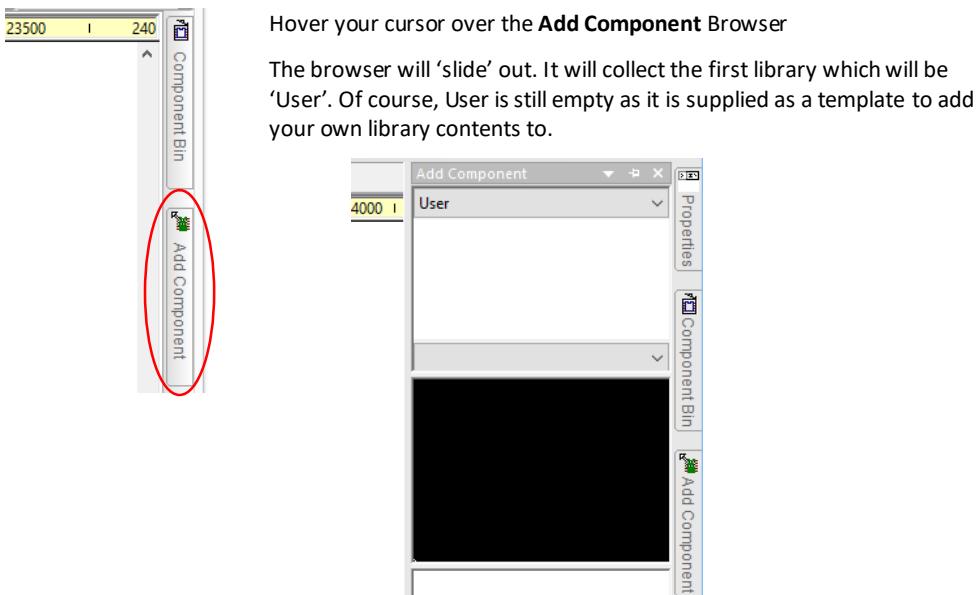
You will now see more libraries have been enabled in the contents list.



Press **OK** to confirm the selection and exit.

► To add a Component

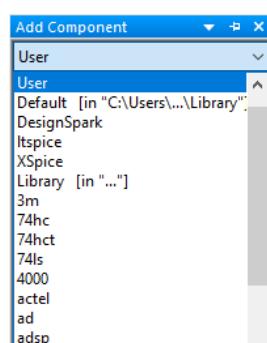
The **Add Component Browser** bar has automatically been run when you started a new Schematic design.



From the list of Libraries, which currently shows **User**, select the small 'down' arrow to reveal the full list of libraries.



The list displays all the libraries enabled.



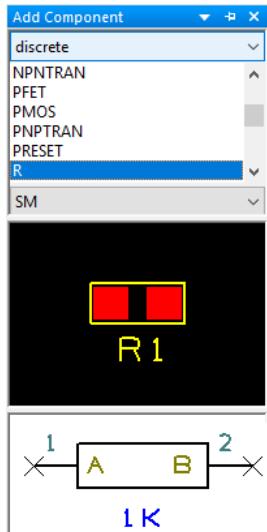
From the list, choose **discrete** You may have to scroll down to see the name.

The **Component** list will now be populated with components from that library. The first component found in the library will be automatically displayed.

From the list, select the component **R**, this will be a resistor. Either scroll down using the slider bar or pick in the list and type R, the list will then automatically scroll down to the first component starting with the letter R.

This is a generic resistor with no specific value now, we will change the Value for later on.

For now, we will only use generic components from the standard library and not worry about them having RS part numbers, availability, cost etc.



A preview of both the Schematic and PCB symbols is displayed.

Once you have the correct component, you can add it to the design by selecting either of the symbol previews and dragging it into the design. You can also add it by selecting its name (R) and dragging it into the design.

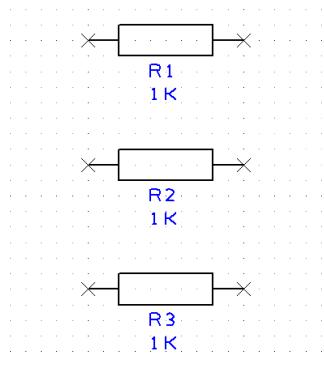
This is a one-shot process. If you wish to add more components you must drag them from the bin or copy an existing one in the design.

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

You need three resistors in total for the final circuit. Once the first one has been placed you can copy it two more times. Select the resistor **R1** in the design. Hold down the **Ctrl** key on your keyboard at the same time as 'dragging' the mouse away from the resistor. As you drag you will see that an identical copy of R1 has been made and is named **R2**.

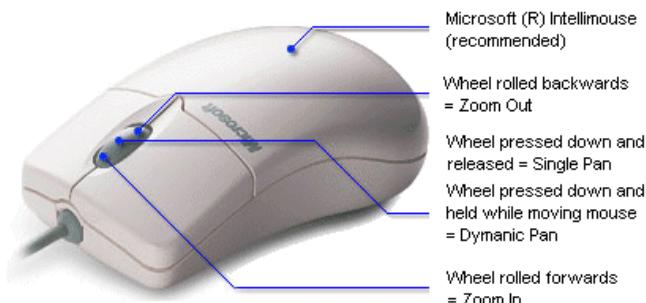
Copy the resistor one more time to add a total of three.

Position them to look like this:



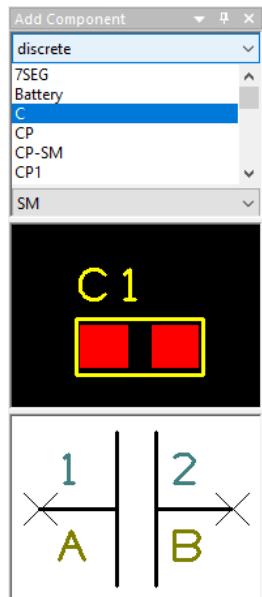
► To pan & zoom in the design

To view the resistor 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 again. Shortcut keys are also available for Zoom in and Zoom out.

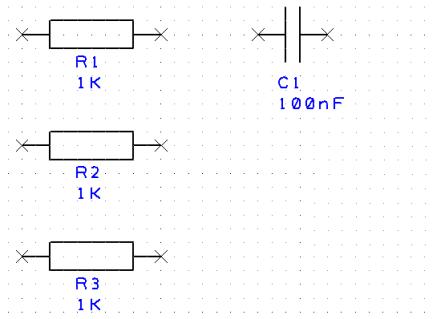


► To add the capacitors

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



Pick and drag one component from the browser to the design to add it.



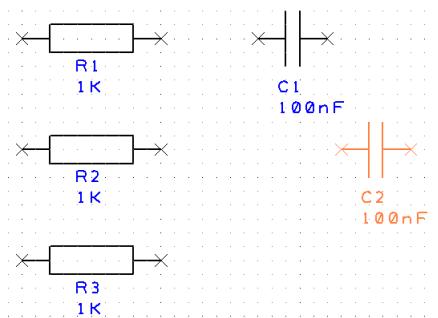
Just the one capacitor will be added.

We will use a different technique to add more capacitors.

Select the capacitor **C1** in the design.

The standard Windows commands of **Ctrl-C** (Copy) and **Ctrl-V** (Paste) can be used to add components by copying them. You can also use the DesignSpark PRO command **Duplicate <D>** to make copies.

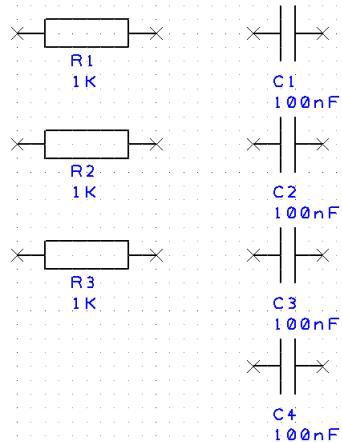
Select **C1** and press **Ctrl-C** to copy it. Now press **Ctrl-V** to paste a copy (now called **C2**) back into the design.



Because the capacitor is in the paste buffer, you can add more of the same capacitor.

Move C2 away just underneath C1 by moving the mouse. Single click to release. Now do the same again, this time to C2 to add another capacitor C3 and then again to C3 to add C4.

Your design now looks like this:

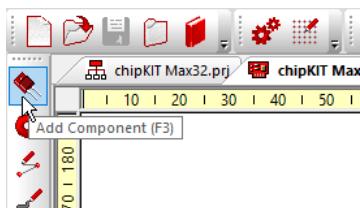


► To switch off the displayed grid

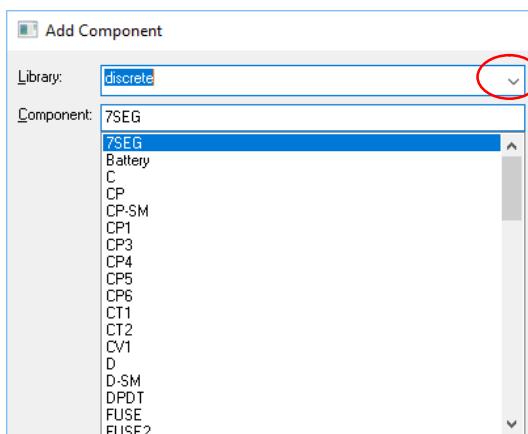
At this stage we don't really need to see the displayed grid dots so let's switch them off. Click <G> to toggle the grid off. Press this key again will switch it back on at any time.

► To add the Op-Amp

We will now add an Op-Amp. This time, click the **Add Component** button on the toolbar, shortcut <F8>.

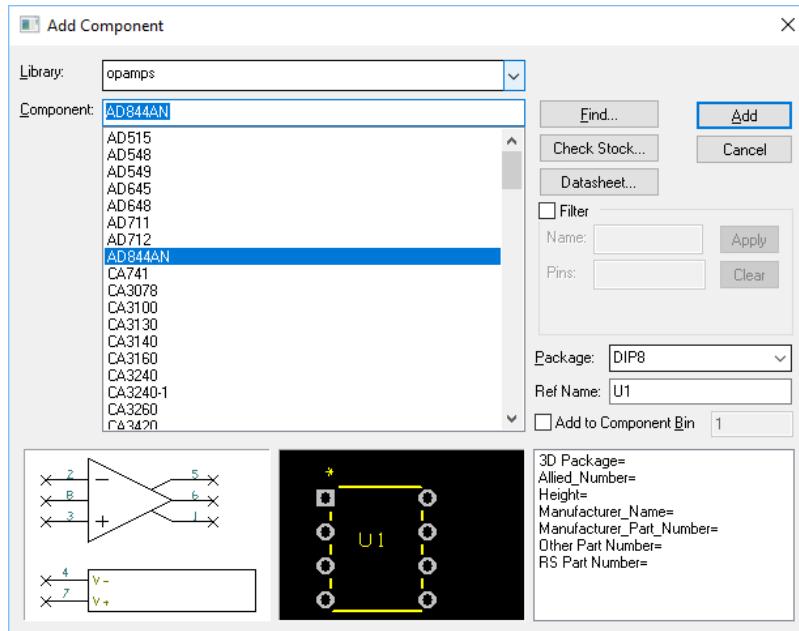


The **Add Component** dialog will display the discrete.cml library that you've already been using with the Add Component browser. We need to swap libraries to another one. Click the small 'down' arrow to reveal the fully libraries listing.



From the list, select the **Opamps** library.

In the **Component:** list, type in **AD844**, this will jump down to the name **AD844AN** on the list. This is the DIP8 version of this device but for this exercise it isn't important.



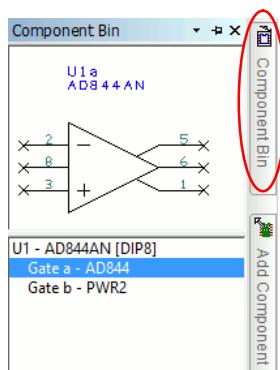
This time we will place the component into the **Component Bin** and not directly into the design to show you another mechanism for adding components to the design.

Before adding it to the design, roll the mouse wheel to zoom out. You will need to give yourself some space on the design in which to place this new component.

Check the **Add to Component Bin** check box on the dialog and ensure **Count:** is set to 1. Click the **Add** button once then click the **Close** button to exit this option.



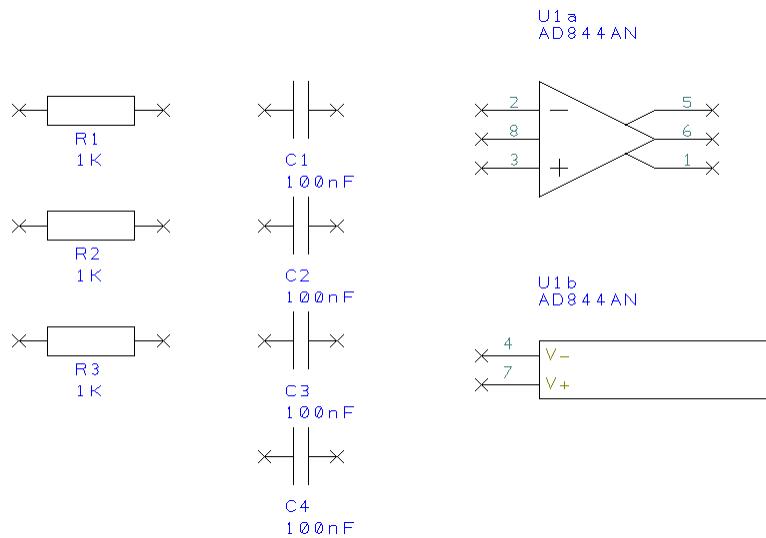
To display the contents of the **Component Bin**, move your cursor to the right side of the DesignSpark PRO window over the **Component Bin** tab. Hover the cursor to reveal the Bin. Moving the cursor away will slide the window back in to close it.



Our Op-Amp (U1) is shown in the Bin as two gates (Gate a & Gate b). Click on **Gate a** under the word **U1 – AD844AN {DIP8}** to see a preview of the symbol above it.

Select **Gate a** and drag it into the design (pick and drag at the same time). Position it near the capacitors. Now go back to the Component Bin (hover the cursor over the Component Bin tab). Drag **Gate b** from the bin. This is the Op-Amp power gate. Drag it into the design, it can go anywhere and can be moved later on.

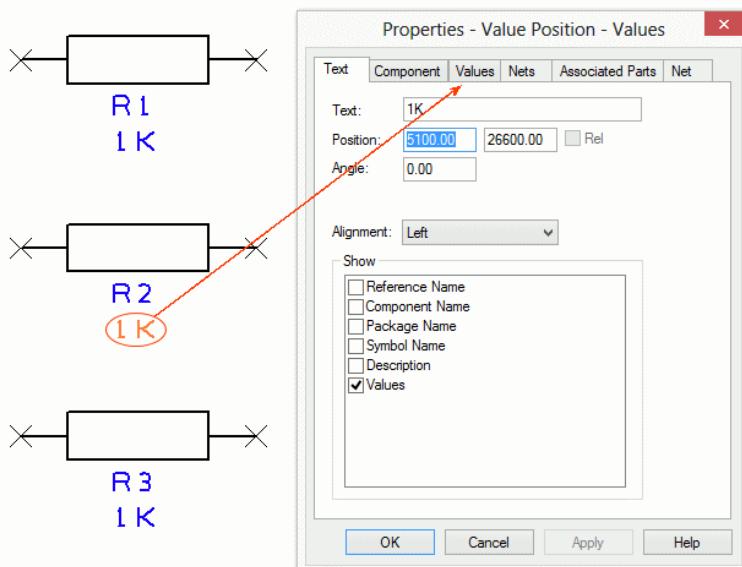
Your design should look something like this:



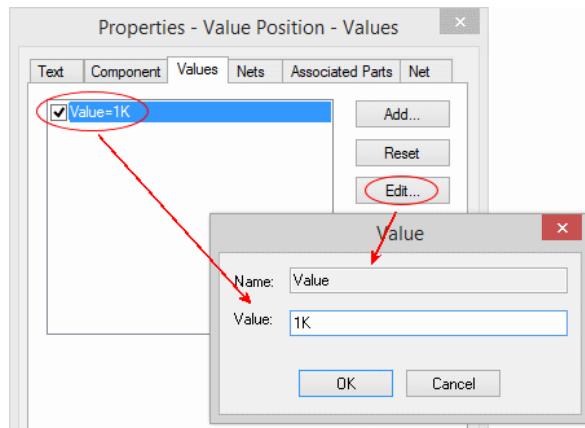
Before we make the connections, we should change the component values to be the ones required. This is easy to do because we've used 'generic' components. If you prefer to use specific Parts, you can also work this way with DesignSpark PRO.

► To edit component Values

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



The **Value** field shows the current 1K value and is already highlighted blue. Simply click the **Edit** button or double-click the blue highlighted field to edit the value required. Type in **11K**.

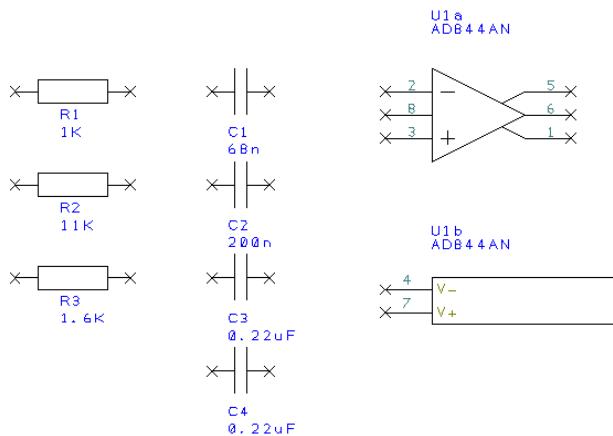


When done, click the **OK** button to exit this dialog and **OK** again to exit **Properties**. When the value of R2 is changed to 11K the design is updated.

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

R1 1K	R2 11K	R3 1.6K
C1 68n	C2 200n	C3 0.22u
		C4 0.22u

Our design now looks like this:



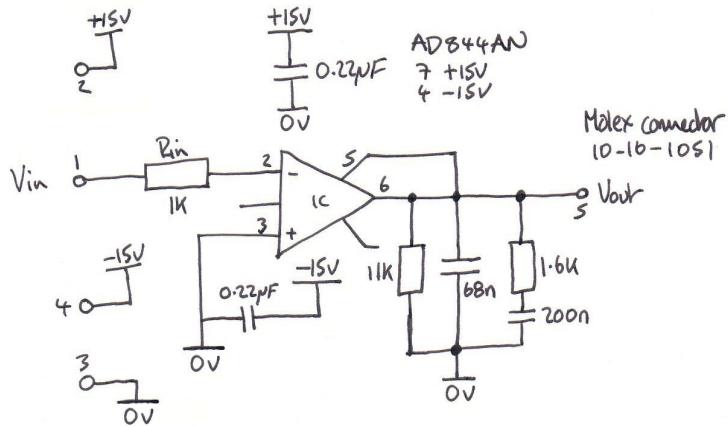
If you select the whole component by mistake, you can still edit the Value field by clicking **Component Values** tab on the **Properties** dialog.

► To save the design

At this stage it might be a good idea to save your design. If you are using the evaluation tutorial with DesignSpark PRO in demo mode, you will not be able to save the design. If you have installed the purchased product go to the **File** menu and **Save As**. Save it somewhere you can find it later on if needed and call it something sensible, like **Tutorial.sch**.

Placing Components

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

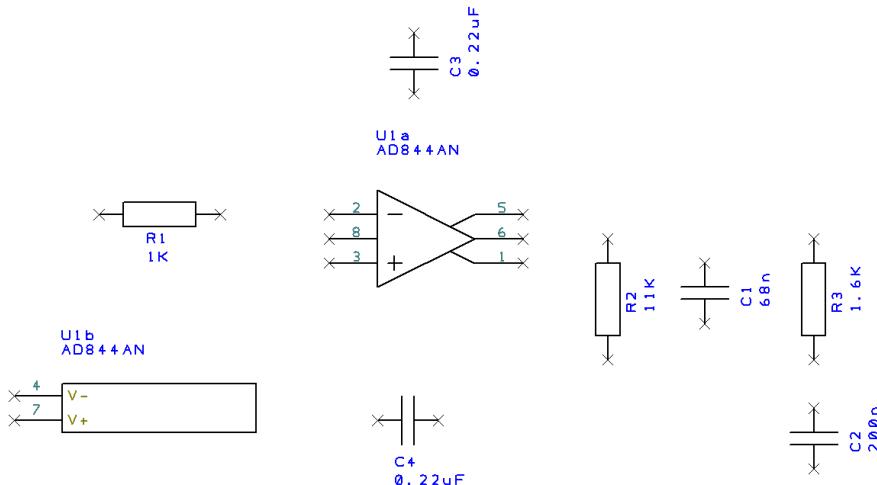


As before, move the components into position by **picking** and **dragging** them in one move using the mouse.

Components can be rotated during move by clicking the R key (while using pick-drag). 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.

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

The design will now be looking like this:



You are now ready to add connections.

Adding Connections

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

By adding a connection when you start this tutorial, default properties are already set up. All you need to do is add the connection. Later on as you become more familiar with DesignSpark PRO, you can change and customise the connection properties to your own settings.

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;

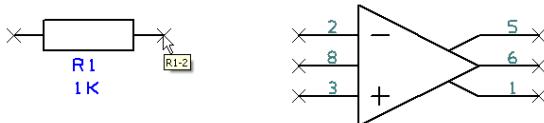
- 'Dragging' off an unconnected component pin to start a new connection.

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 show a finish marker; a small illuminated blob over the pin. This shows the item as a legal landing site to finish the connection.

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

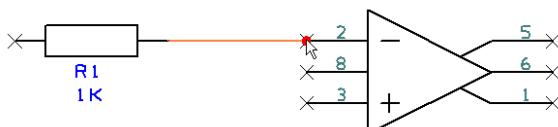


When you hover the mouse over the right hand terminal of the resistor R1 a small design tooltip label is displayed. This confirms the name of the device and the pin number (R1-2). Although on this resistor it is a bit excessive, on a BGA pin which has a connection and net name, it becomes very useful (and essential!). These design tooltips are available for all designs items, not just pins.

Click and drag (keeping the mouse button pressed down) off pin 2 of R1, this will start a new connection.

Move the mouse over U1 pin 2.

You'll now see a finish indicator with a small solid circle on it. This indicates that the connection can legally be finished here.



You do not have to be exactly over a pin for the finish marker to be displayed, as long as the cursor is within range then it will be shown.

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.

Following our 'sketched' design, complete the connections in the design.

► To join connections together

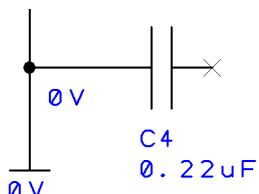
There are a couple of connections which must be connected together. Let's assume that you've made a connection and now you need to connect another to it using a junction dot.

When you move over the existing connection and hover momentarily, a finish marker will be displayed. Click to release the connection here and this net is now connected to the one already there. A junction dot will automatically be added to show the connections are connected.

Complete the design adding the connections required.

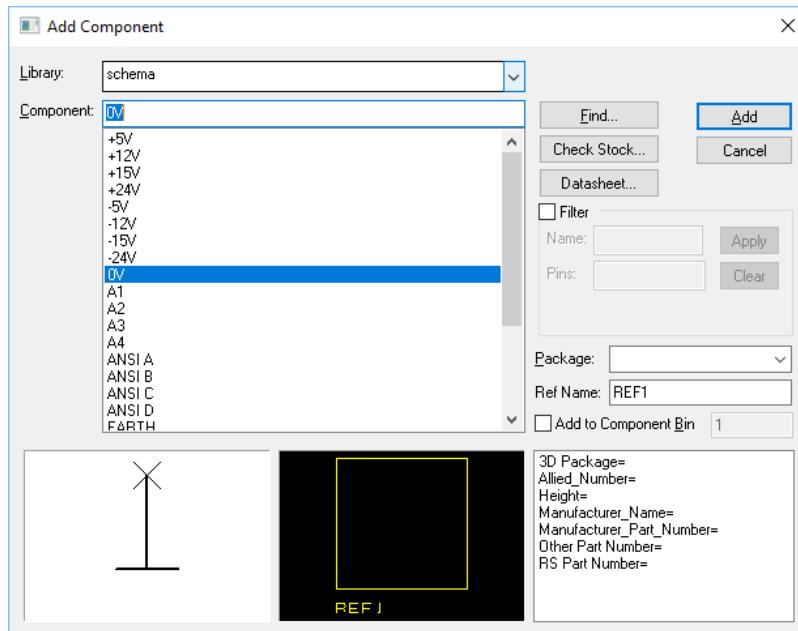
Adding Power & Ground Symbols

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

Click on the **Add Component** icon on the toolbar.

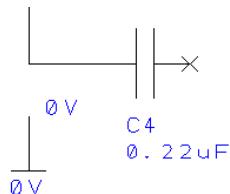


In the **Library:** drop down list, select the **Schema** library. Any selected symbols will be previewed as being Schematic symbol with no PCB footprint.

In the **Component:** list select **0V**. You will see the symbol selected in the **Preview** box.

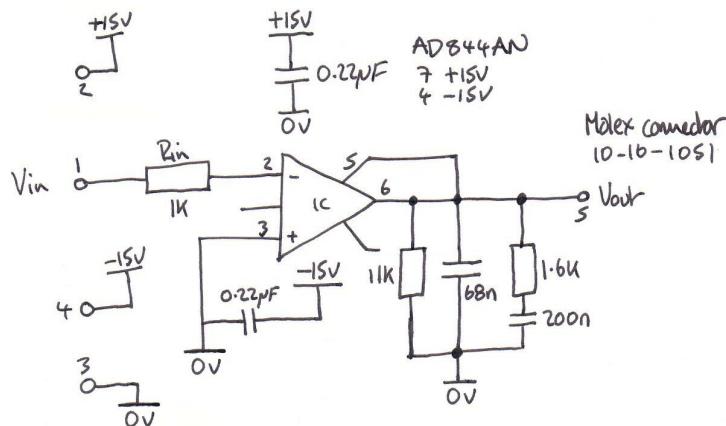
Ensure the **Add to Component Bin** check box is unchecked, click the **Add** button.

The 0V symbol is on the end of your cursor. Move it near to **C4** and click the mouse once to release it.



Using our working design, add two more 0V symbols, then click the <ESC> key to cancel. Select the **Add Component** option again.

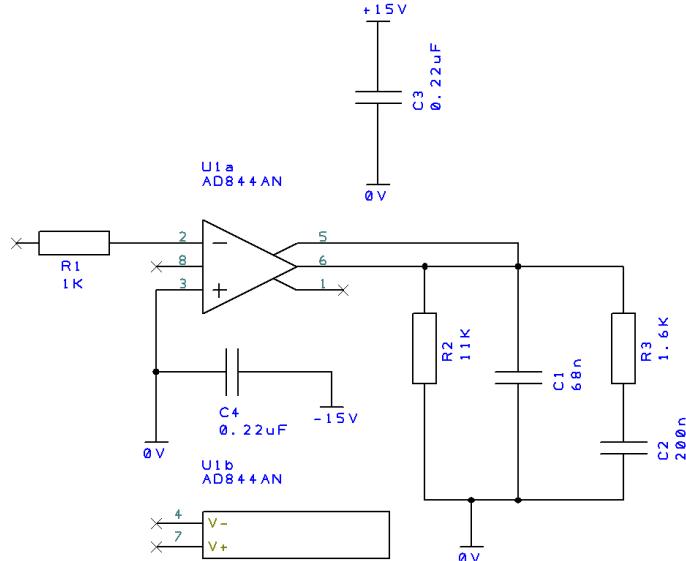
Our original design is shown below. Your component names may differ slightly, it doesn't matter, their relative position and values are more important.



Add one +15V symbol and place near C3, then <ESC> again. Now add one -15V symbol, place near C4. Click <ESC> to exit.

Drag a connection off the end of the symbols and connect as shown below. Move the power gate for U1b to the position shown below just under C4.

Your design will look something like this:



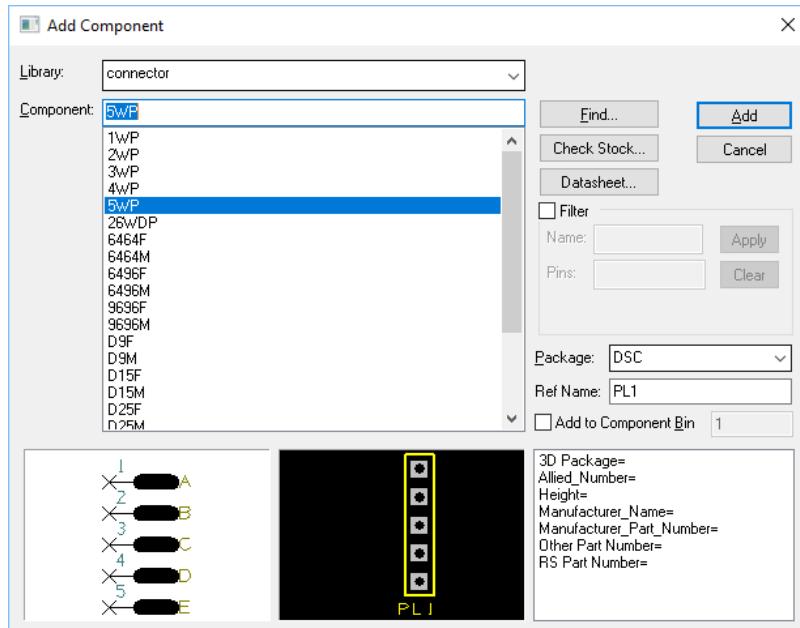
Generally speaking, you would now need to add some connector pins to connect the circuit to the outside world.

Adding Connectors

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

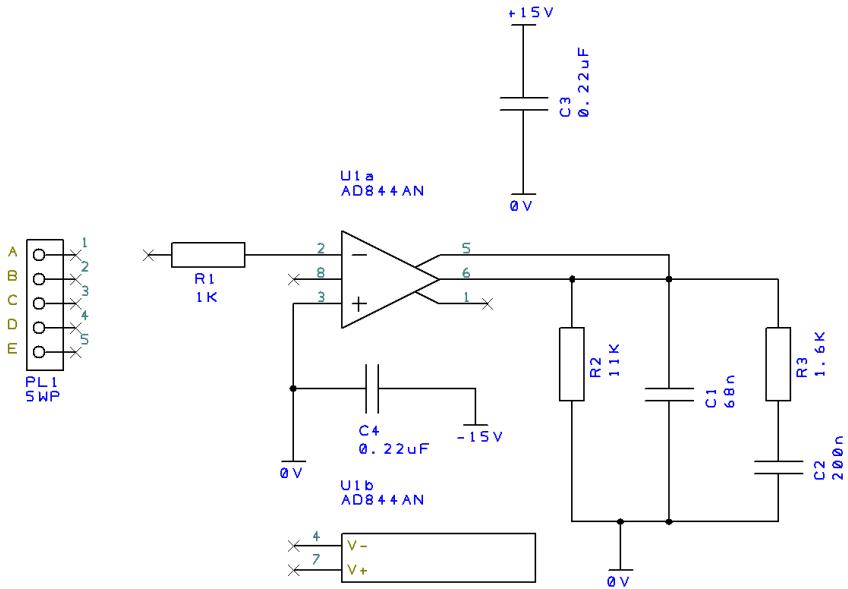
► To add Connector pins

Click on the **Add Component** option from the **toolbar**.

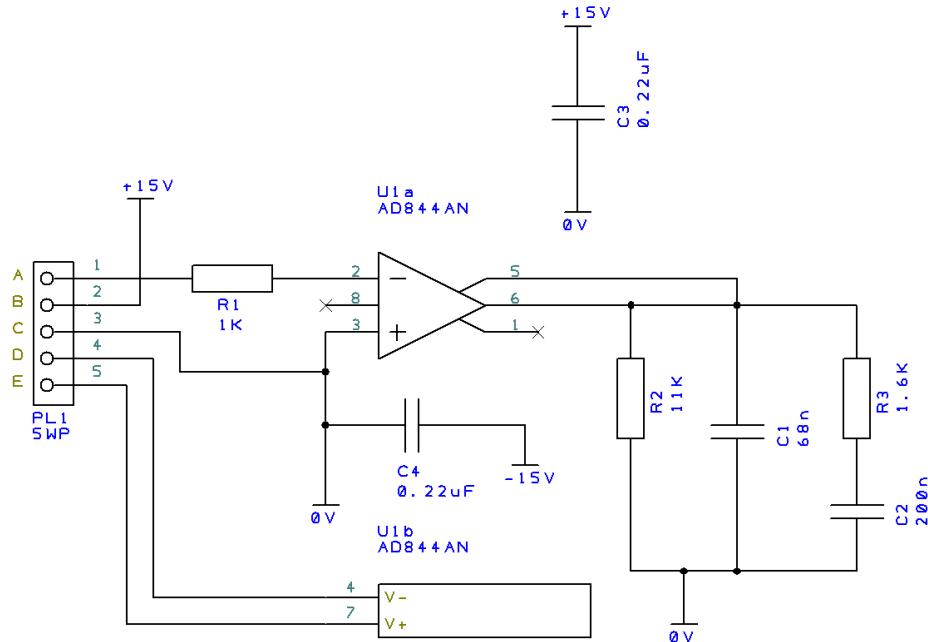


From the dialog, select the **Library: Connector** from the list. In the **Component:** list select the **5WP** connector, this will be shown in the preview windows along with it's PCB Footprint. It will be automatically named PL1.

Click the **Add** button to add it to the design, place it near the resistor R1. Click the **<ESC>** key to cancel Add Component. If it comes in mirrored to what is shown here, mirror it back using **<F> Flip**.



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



Click the **Save** button on the **File** menu to save the design. If you are using the tutorial in demo mode you cannot save it.

Naming Connections

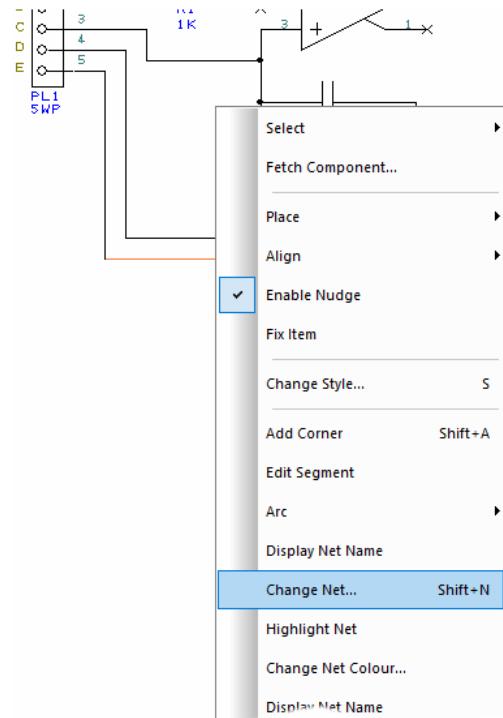
One last task that is often required is that of naming connections in the design. We've added 'normal' connections which have been named using the default naming convention and have been given 'system' names like N0005, N0100, N008 etc. We've also added connections between pins and components that have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

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

► To edit net names

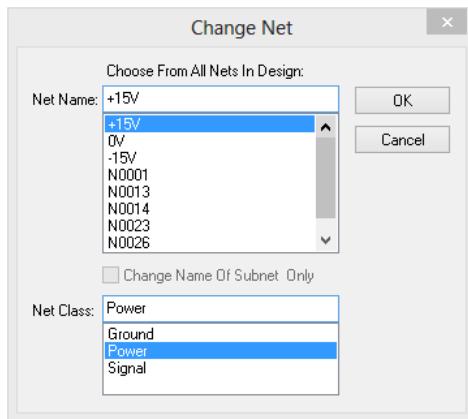
Select the connection just off pin 7 of the Op-Amp's power gate, the gate marked as U1b.

Right click and from the shortcut menu, select **Change Net**. If you use this a lot, you can assign it to a shortcut key.



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

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



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

If you wish to see the Net Name in the design, right click again with the connection selected and choose **Display Net Name** from the menu (this is the option just above the previously used Change Net option). The net name will now be displayed in the design, you may reposition it if required by picking and dragging it.

Change net name on the power gate

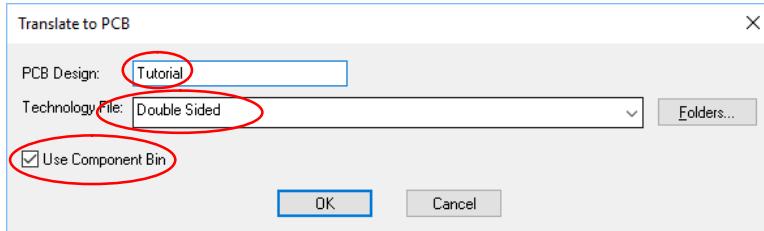
► Change net name on the power gate

Now select **pin 4** on **U1b**. Using **Change Net** again, this time select the net name **-15V**. Select the Power **Net Class** again. Click **OK** to exit this dialog. Do the same with **pin 7** on **U1b** and make this **0V**. The Net Name can be displayed using the method above if required.

Translating the Schematic to PCB

The final process before starting on the PCB design itself is to translate the Schematic design into the PCB design environment.

When you are in a position where you wish to go-ahead with this translation you should use the **Translate To PCB** option from the **Tools** menu. This is discussed in the following pages in the PCB Design section. Select this option now to carry on.



Type in a **PCB Design**: name, for example **Tutorial**

Select a **Technology File** to be used for the PCB design. Use the **Double Sided** technology file. This will give you a basic double sided design.

Check the **Use Component Bin** box. Click **OK** to translate the components and netlist to a PCB design.

The PCB design editor will run up. All of your PCB footprint components will be in the Component Bin, the design will initially appear empty, don't be alarmed this is normal!

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 3. PCB Design Editor

Getting Started with the Design

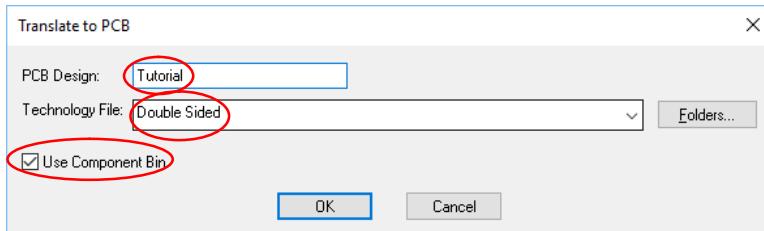
DesignSpark PRO PCB is flexible in that you can start your layout in a number of ways:

- Use DesignSpark PRO Schematic Capture as the 'front-end' to the PCB system. The nets and components are translated to PCB in the integrated environment.
- Use a netlist that is derived from another CAE or Schematic system.
- Use the DesignSpark PRO PCB design editor interactively creating the design on-the-fly without the need for an initial netlist. You just add connections and components as required to create the PCB design.

Translate to PCB

We will take the 'logical' Schematic design created in the previous section and convert it to a 'physical' PCB layout.

You can skip this bit if you have already translated your design, go onto **Setting up a PCB Technology** below. If you need to translate the Schematic to PCB - from in the Schematic design editor with our tutorial design, select the **Translate to PCB** option from the **Tools** menu.



Type in a **PCB Design**: name – **Tutorial** (no file extension is required, just the name).

Under **Technology File**, choose **Double Sided** to be used for the PCB design. The Technology predefines 'rules', such as the layers, pad styles, track styles, colours and other aspects of the 'make-up' of the overall PCB design.

Technology files make the initial set-up of a design much quicker but you can modify all aspects of this during the design stage.

Select the **Use Components Bin** box, the Translate option will put our components into an off-page location for clarity for now.

Click **OK** to convert the components and netlist to a **PCB design**. Your design will appear blank, all the components are ready to place in the Component bin which is not visible.

At this point you are ready to start laying out the PCB design.

Setting up a PCB Technology

When choosing or setting up a Technology file in your design you must think ahead and think about the final manufacturing process and whether your manufacturer can handle it. The technology used by you can have an impact on the cost of the final PCB. Changing your track styles from 8 to 6 thou for example, or adding blind and buried vias could have significant costs implications. You should always check that the boards can actually be manufactured.

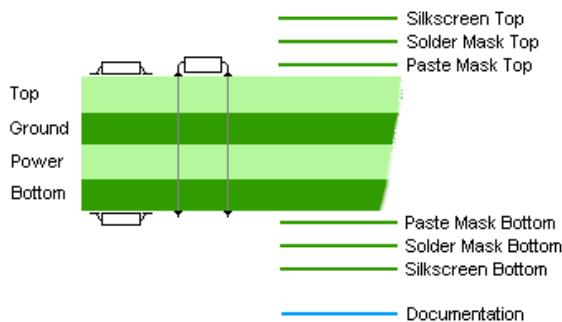
All aspects of the technology can be edited and added to as you go, this includes the number of layers in the design, styles and names etc.

For this exercise, you will use a pre-defined technology file, **Double Sided**. This provides you with a two-layer PCB with a set of general purpose styles and sizes. It also uses a black background; this can be easily changed if you prefer a lighter colour background such as white or light grey which seems to be easier on the eyes.

Layers

The DesignSpark PRO PCB design system supports an unlimited number of layers in any combination of Electrical, Non-electrical, Wire, Construction and Documentation layers. These layers are used to detail the physical 'make-up' of the manufactured PCB.

Below is an illustration of a typical 4 layer PCB showing the 4 electrical layers, 2 outer layers - Top and Bottom and the inner power plane layers - Ground and Power.



If you wish to view the layers dialog, from the **Settings** menu select **Layers**. Click **OK** to close the dialog once you've viewed it.

Creating a Board Outline Interactively

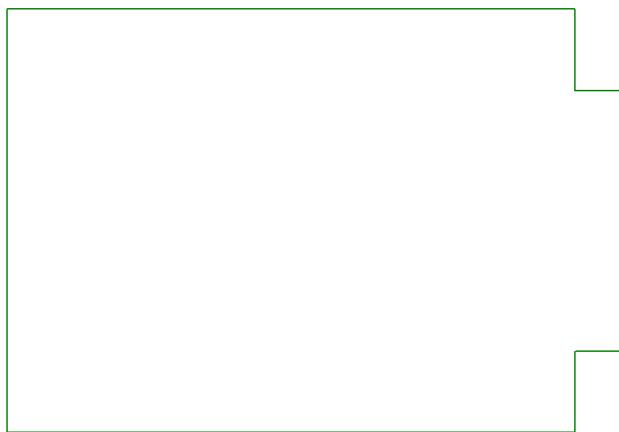
Board outlines can be created easily in DesignSpark PRO; by adding board shapes (including circles) or by importing DXF files from your mechanical system which contain board outlines.

Add Board allows you to interactively create a board outline by drawing it into the design. Three shape styles are provided: Rectangle, Circle and Polygon.

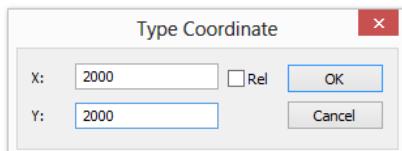
► To insert a board outline

On the **Add** menu, click **Board> Rectangle**.

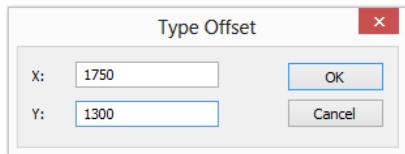
This is the shape you will create; the overall length is 1900 thou and the height is 1300 thou:



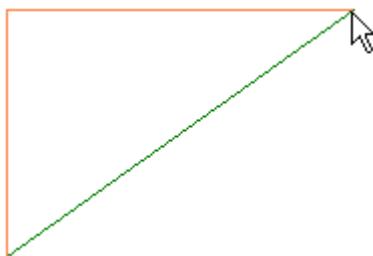
With **Add Board** selected, in the design area right click and select **Type Coordinate** from the context menu. On the dialog, type **2000 X** and **2000 Y**, then click **OK**. This starts the shape.



Now right click again and from the context menu, select **Type Offset**. On the dialog, type **1750 X** and **1300 Y**, then click **OK**.

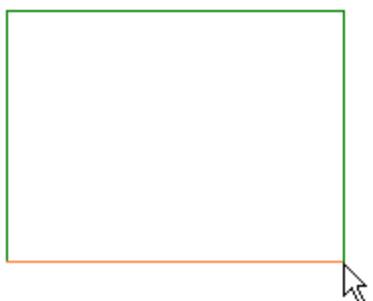


Your board will look like this so far:

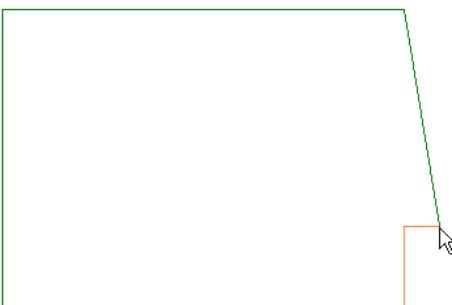


Click the mouse once to add the corner at the top right side.

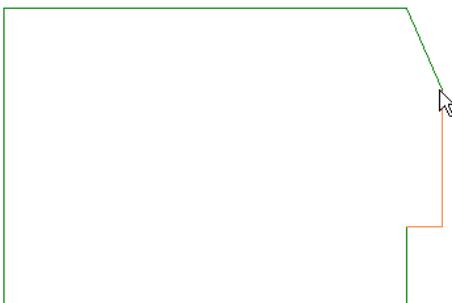
Drag the corner down so that a 90 degree corner is formed and the remaining two sides are straight. Double-click to finish.



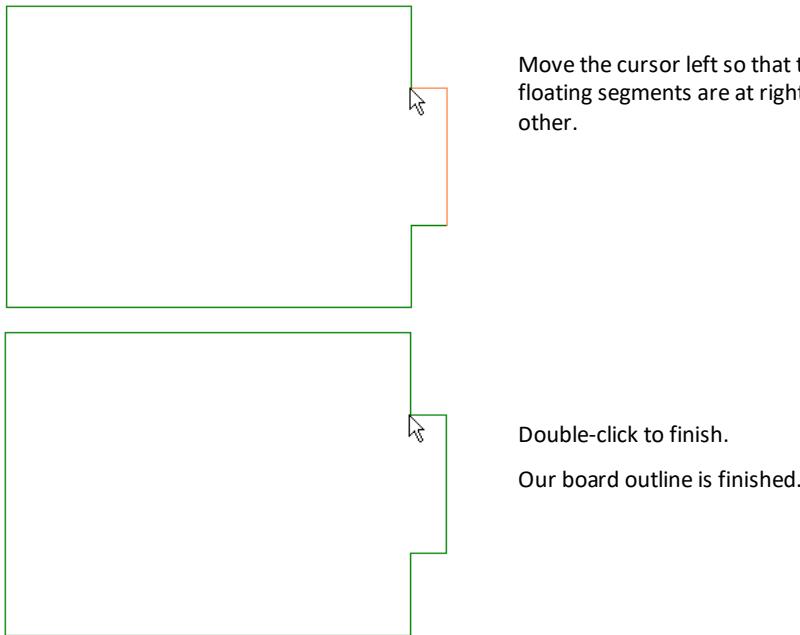
Once a board outline has been added, it can be modified afterwards, extra segments added and the corners can even be mitred or curved.



Add the 'notch' now. Double-click on the right side to insert a corner. Move the cursor right 150 thou and up 250 thou. Single click to add the first corner.



Move the cursor up 800 thou. Single click to add a corner.

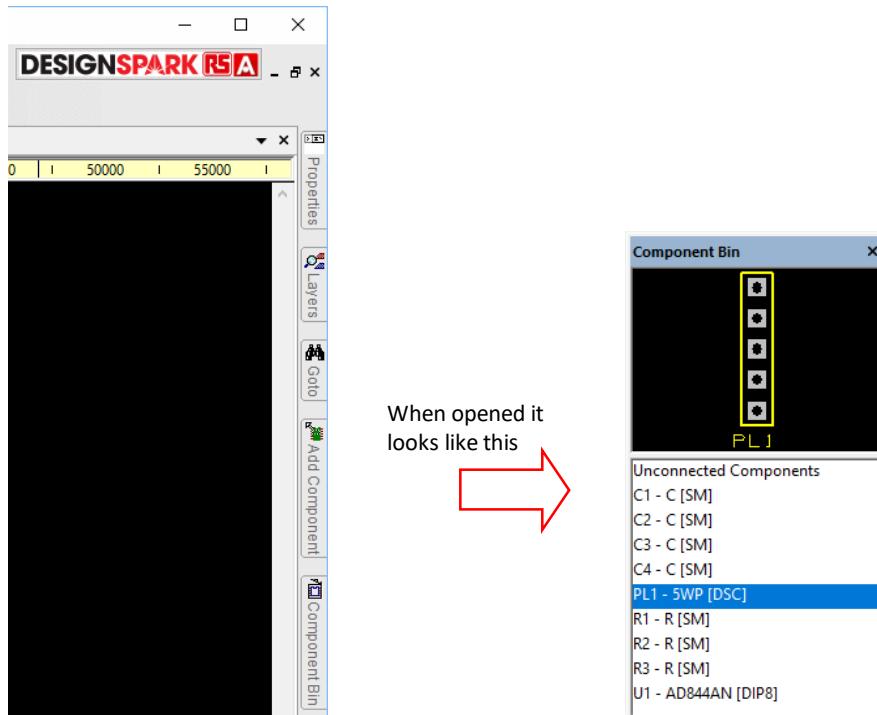


We now need to place the components in our design.

Using the Component Bin

While we've been starting the design with the board outline and preparing the technology, our components have been sat in an off-design location waiting to be placed. During the Translate To PCB stage, we checked the box on the dialog to place the components in the **Component Bin**. There are other ways to work, such as placing unplaced components around the board outline or placing them in a 'pile'. We now need to open the Component Bin and place the components in the design.

From the right side of the DesignSpark PRO design edge (called the framework), you can hover your cursor over the tab **Component Bin**



This will display the components that were sent to the bin.

► To drag Components from the bin

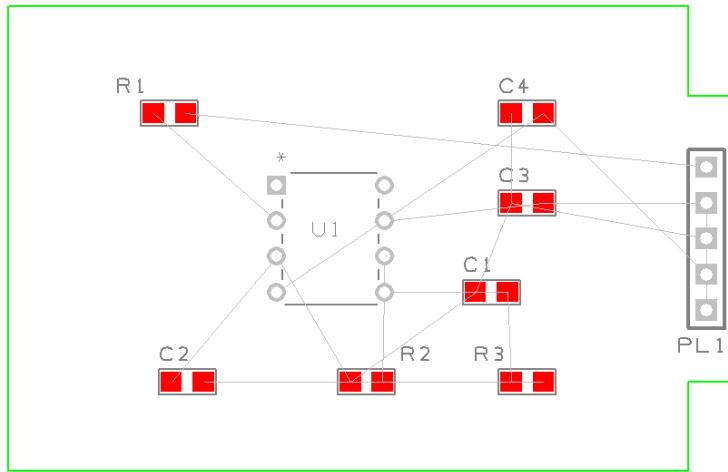
You can select a component from the **Bin** or the **Preview** window. To use a component, ‘drag and drop’ it into the design (pick it and while picking, drag it into place).

Note: during place the drag may be cancelled by dropping the component back into the Bin, by pressing the <ESC> key, or once released by using Undo (**Ctrl-Z**).

Once in the design the component can be moved until released, again by picking and dragging.

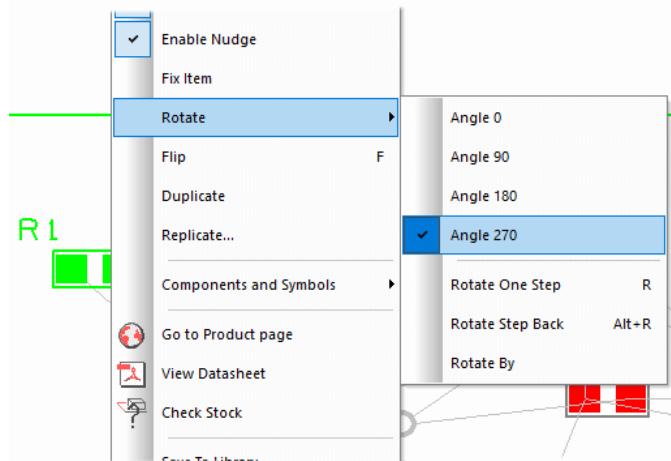
The following PCB images have now had their backgrounds changed to be white to make them stand out more clearly on paper. You are free to use a black background or any other colour of your choosing.

Place all the components to look like the picture below. It is suggested you place **PL1** and **U1** first and place the other components around these. Placement on this design isn’t critical but if you can improve on it, do it.



Moving Components

Once the Components have been dragged from the bin you can place them in the design. While placing components, use the shortcut key **R** to Rotate them and **F** to Flip (mirror) surface mount components, these options (and more) are available on the shortcut menu during Place by clicking the right mouse button.



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

Once fully placed, the design should be routed.

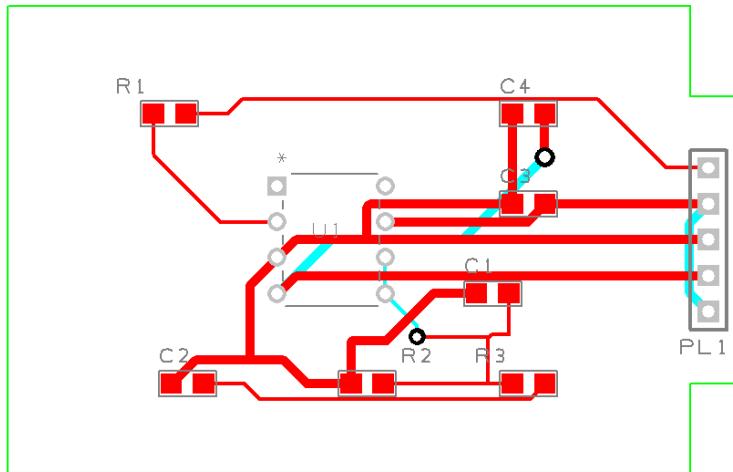
Save the design at this point using the **File** menu | **Save** option.

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’.

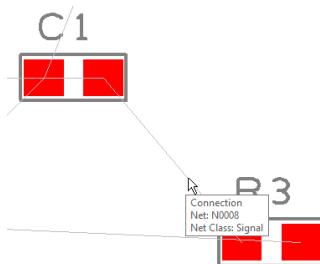
Initially we will route the design manually. DesignSpark PRO has a set of powerful manual routing tools to aid this process. These tools allow you to swap layers, automatically add vias on layer swaps and edit track thickness.

The routed board might look something like this (we’ve changed the colours to make the design easier to view in this tutorial):



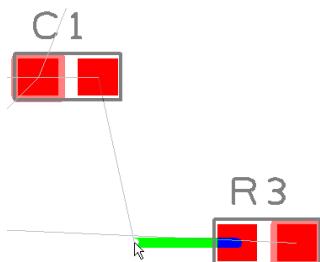
► To start routing tracks manually

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

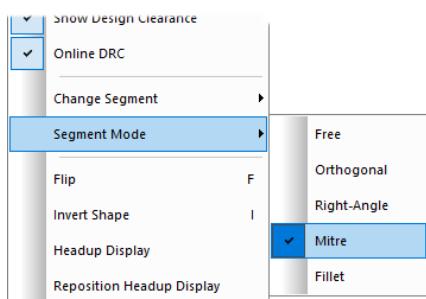


Hover the cursor over the connection to reveal the **design tooltip**. This will tell you that you have selected the correct connection.

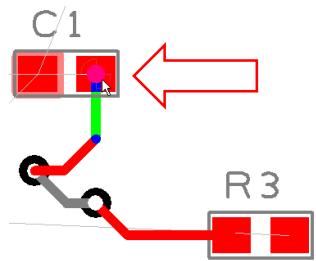
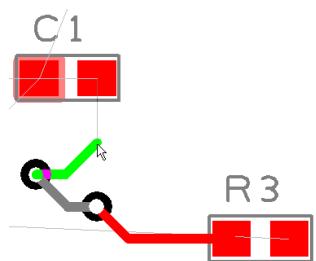
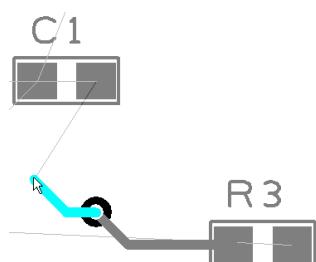
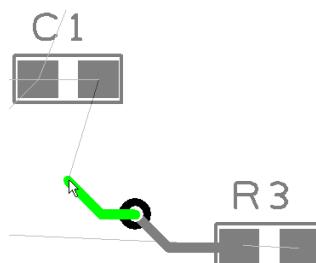
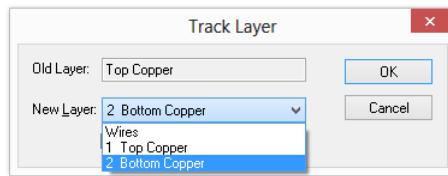
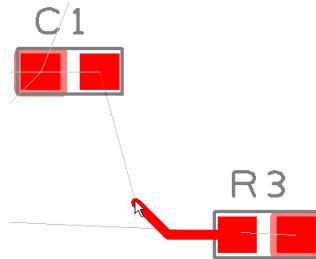
If you followed our design placement loosely, **double-click** the connection between R3 and C1.



Once started, move the cursor to the left slightly. You’ll see the track starting to take shape and a connection between the end of the track and the ‘target’ pad (on C1).



If you right click once routing has started, you can select Segment Mode> from the menu. From the submenu, you can choose how you wish to route the board. It is normal to use Orthogonal mode (two floating segments at 90-degrees) or Mitre mode. Mitre mode is our preference and used for this tutorial. This will route with a 45-degree angle on the second floating segment.



Click the mouse once, this places the first track segment, then move the cursor up slightly. You'll see the 45-degree track segment, click the mouse again.

If at any point you make a mistake, use the **<Backspace>** key to delete the previous segment back to the last corner. You can do this while still in 'edit' mode.

Now we will add a track layer change.

Press the '**L**' key on the keyboard and the **Change 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 and is shown in a different colour (blue).

Move the cursor left and up slightly. This will either be the next corner or the next location for a via for a layer change. Click the mouse to release a corner.

Obviously, this is a bit unrealistic because we could just route it on the same layer.

Press '**L**' and **<Enter>** again to change layers back to the Top side. Another via is added automatically. The via and track sizes are taken from the net class, this is set up as part of your Technology file.

Notice how the tracks and pads on the top layer have changed colour to indicate you are routing on the other side.

Move the cursor up towards the target C1. Notice how it is on the Top layer again and shown in Red.

Move the cursor (with track) over the target pad. When a legal landing site is available, the cursor changes to a finish marker, a highlighted 'blob'.

Click once to finish the track editing for this connection. After the track has been added, it can always be modified afterwards by picking and dragging the track or by double-clicking on the track to edit it.

Move onto the next connection for editing.

Summary of the basic modes of routing

During track editing, the basic commands for use are:

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

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 PRO 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 keys used for the commands **Select Next** and **Select Previous** are set up as **N** for **Next** and **P** for **Previous**, these can be changed if you wish to.

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.

Unrouting Tracks

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

Try using the **Unroute Nets> All Nets** option to unroute the whole design removing all the tracks already done. For the next section we will use the autorouter to re-route the design. Don't forget, once unroute, you can always restore the routes using **Undo (Ctrl-Z)**.

Using the Autoroute Option

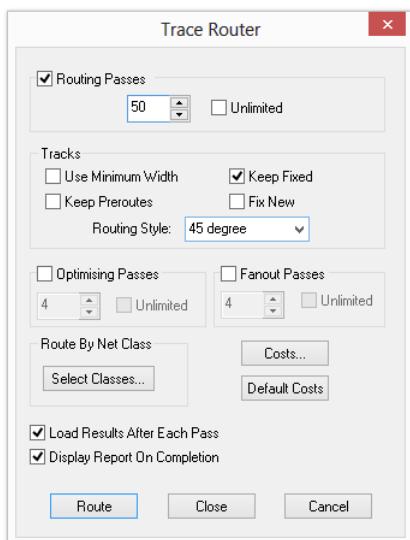
The Trace-Router autorouter option is available in the demonstration mode or if you have purchased it. This feature is used to convert your nets into electrical tracks as with manual routing discussed previously. However, on dense designs it can save you many hours or days of work. The autorouter can take an unroute 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**.

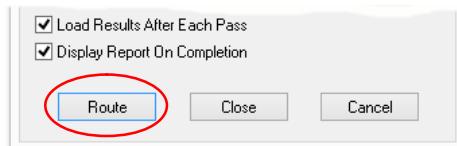
► To use the autorouter

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

We will leave the default settings as they are:

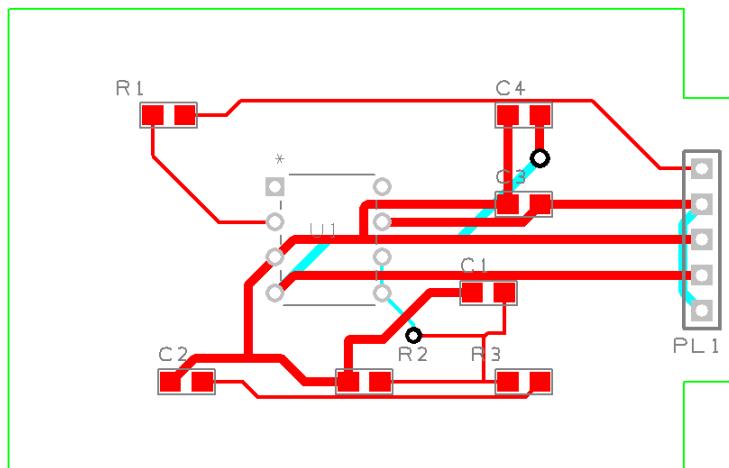


Simply click the **Route** button at the bottom of the dialog to start the router. This small design will easily route 100%.



The results are displayed in a report, click **OK** to close the dialog and view the completed routes.

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



► Save the design

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

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

Pouring Copper into areas

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

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

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

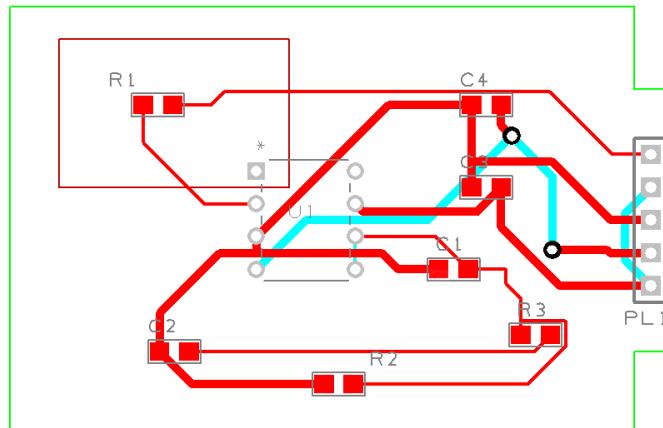
The principle of this facility is to insert a Copper Pour Area and ‘pour’ the copper into this area.

► To create a Copper Pour Area

From the **Add** menu, select **Add 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 PRO will pour the copper using the Spacing rules and rules defined for the Pour area.

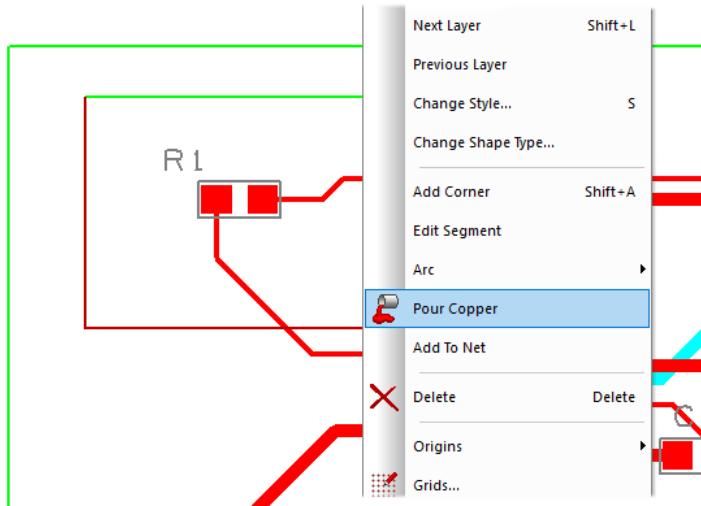
Following the picture below of our example, drawn in a basic rectangle pour area shape, this is shown in black in the top left hand side of the design:



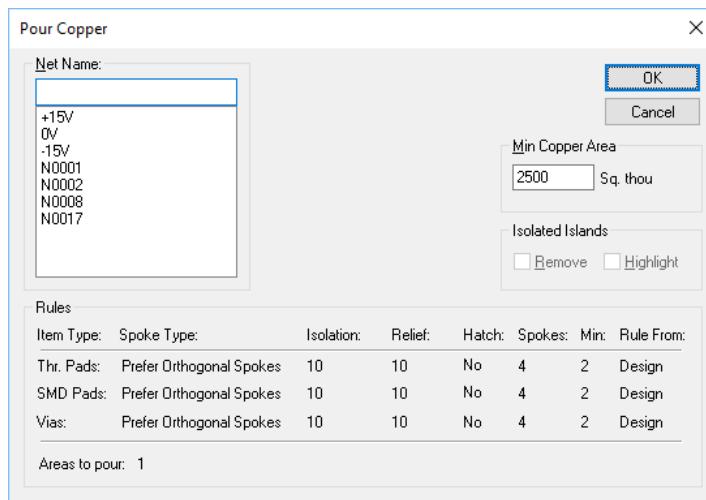
For this example, you will see how easy it is to add a poured copper area.

Right click on the pour area.

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

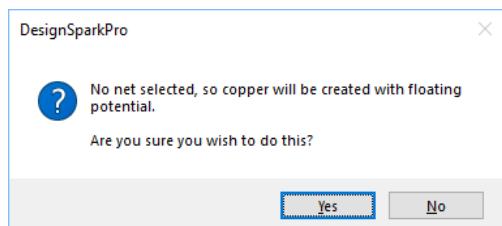


A dialog is displayed from where you can select parameters for the poured area, such as the net name association and thermal connections etc.

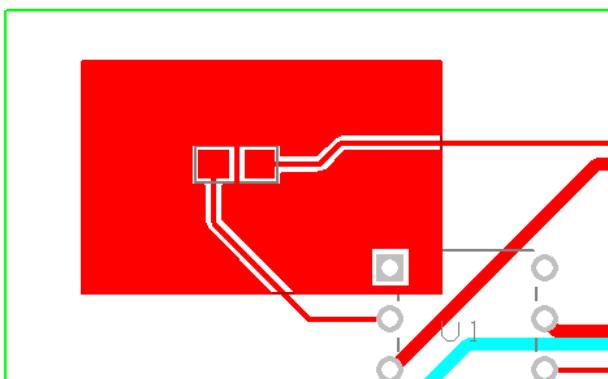


For this exercise, leave the **Net Name** blank and simply click **OK**.

A warning dialog is displayed. Normally, you won't see this as you would associate the copper with a net, such as GND.

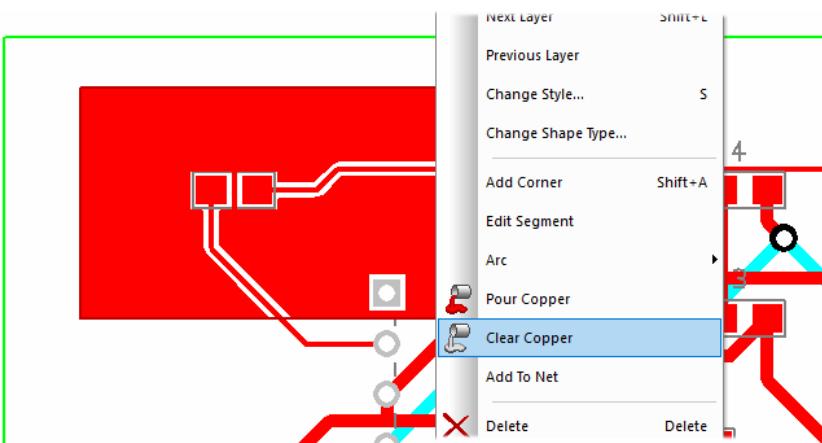


Notice that the poured copper has obeyed the **Spacing** rules of all defined items to copper including Board to Shape rules.



► To remove poured copper from a copper pour area

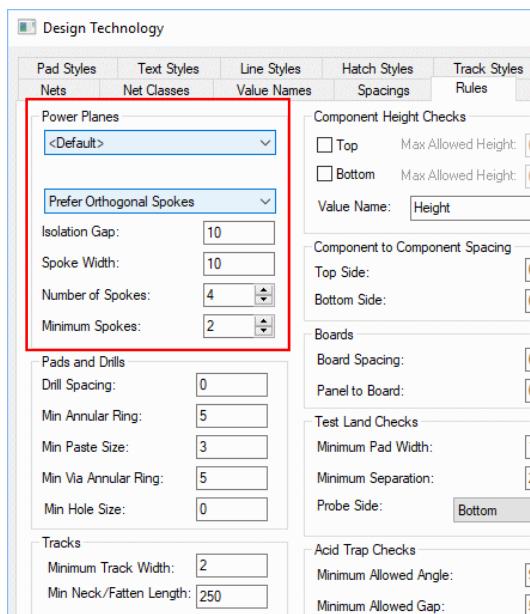
To remove a poured copper area (including any thermal spokes), simply select the area outline and select **Clear Copper** from the shortcut menu. This removes the copper but not the Copper Pour area.



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

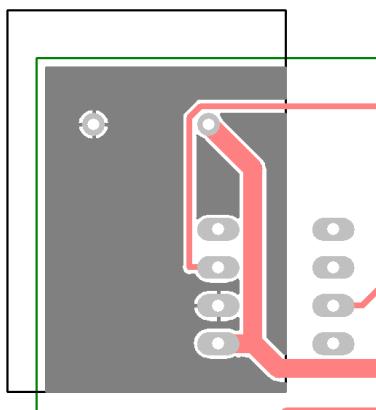
There are more facets to the poured copper than we have discussed above.



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 OV as the net name and click **OK**.

From the **Settings** menu, choose **Design Technology** and **Rules** tab. This allows you set up your copper pouring rules.

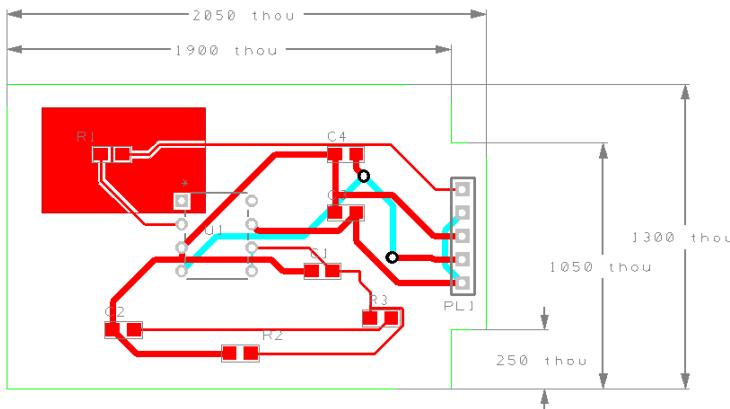
Use the **Isolation Gap**, **Spoke Width** and **Number Of Spokes** settings, this defines how the Thermal spokes will be presented in your design.



Copper Pour Area outlines are not plotted and will not appear on your plots, they provide the system with an intelligent area into which the copper can be poured.

Adding Dimensions

Dimensions used for documenting the physical measurements of the PCB and can be added using the **Add Dimension** option from the **Add** menu. This is used to display (in the form of a dimension) lengths of items, radii of arcs and angles between lines.



► To insert dimensions

Select **Add Vertical Dimension (Ctrl-J)** from the **Add** menu and **Dimension >**. You can also select **Horizontal** and other **dimension types**.

Select the first point to start the leader line (the board outline for example), then the second snap point which will be the outer dimension limit.

Move your cursor away from the design so that the dimension measurement text and leader lines can be placed. Click the mouse to place the text. You can move the lines afterwards to make adjustments. Aspects of the dimensions (text, values, lines, arrows etc.) can be customised to your own requirements using **Defaults** and **Properties** of the dimension.

Checking the Design Integrity

The **Integrity Check** option is 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.

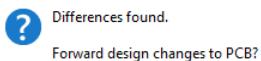
► To check the designs

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

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



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



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

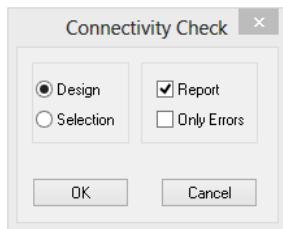
► To back annotate name changes to the Schematic

Back annotation changes can be made in the PCB, these can also be passed back to the schematic 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’.

► To validate the electrical connectivity of the design

Another essential aspect to the overall design integrity check is the validation of full electrical connectivity in the PCB design. You need to be sure that all connections have full electrical paths between each point. This would include electrical paths through copper planes, vias, plated and non-plated through-holes and through tracking.

Run the **Connectivity Check** from the **Tools** menu. This will display a dialog from which to select what to check.



A report of all the connectivity is displayed. Any warnings or errors are reported and should always be investigated. The full report above the summary explicitly lists these warnings and errors so you can locate them.

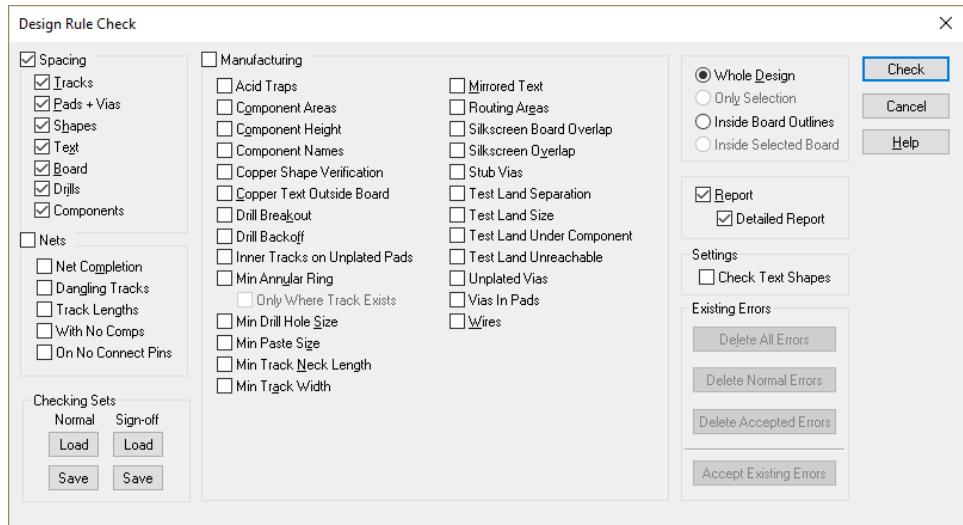
Design Rules Checking

Before outputting 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 **Tools** menu.



A dialog is displayed that allows you to control which items are included in the check.

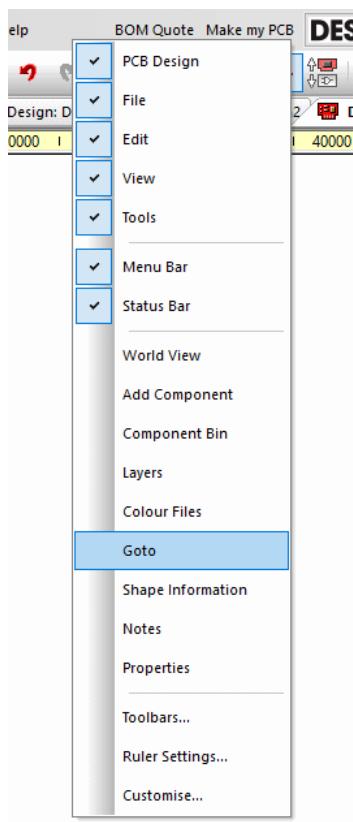
A minimum check would be for **Spacings** but other checks can be made as required.

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 by right-clicking the mouse with the cursor positioned over the DesignSpark PRO framework (the area around the design or over one of the menu items or toolbar icons).

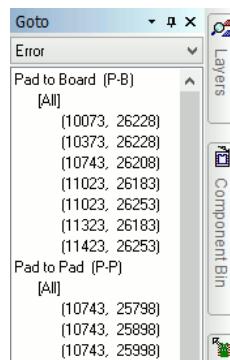


Select the **Goto** option from the menu list.

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



This list is used to view all the design error markers as a sorted list. It is displayed in an 'active' browser. You can use a double click of the mouse to select an error by name in the browser and then the error marker in the design. The error marker will be brought into the centre of the display so it can be seen, checked and cleared.



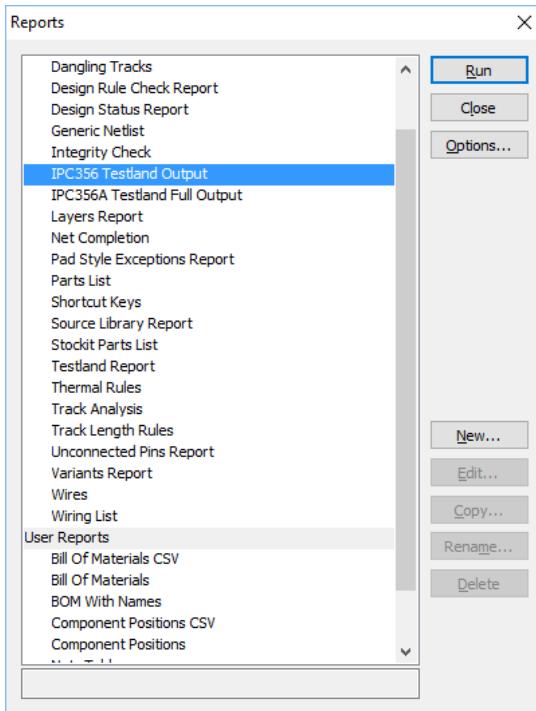
Reports, Part Lists and Net Lists

DesignSpark PRO can generate reports where all vital aspects of the design can be reported in detail. The reports also include a set of ‘standard’ reports like Parts lists and bill of materials (BOMs).

User defined Parts lists and Netlists can also be generated using the **Custom Reports** option.

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

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



You'll see on the list plenty of ‘general’ reports plus **User Reports** that can be created and customized by you to your own requirements.

Plotting the Design

DesignSpark PRO provides you with an output mechanism for generating professional PCB layout plots for manufacturing. All the plot and drilling outputs are located in one easy-to-use dialog.

From the **Manufacturing Plots** option on the **Output** menu, you can output all types of plots. You can select the layer or combinations of layers to plot. You can also set the scale, rotation, the position on the plot and the plotter type. All plots and outputs can be delivered in one button press once set up.

Plotter types available are:

- Gerber 274-D, 274-X (extended Gerber format) and Gerber X2 format photo-plots
- Numerically controlled drilling (NC Drill) information
- Windows printer plots using any installed Windows driver
- Active PDF files
- HPGL Pen Plots
- Standard and customised reports
- ODB++ format files

Plot types

Plots are available for all aspects of the design but because DesignSpark PRO uses a configurable output mechanism, it means that almost any plot required can be generated:

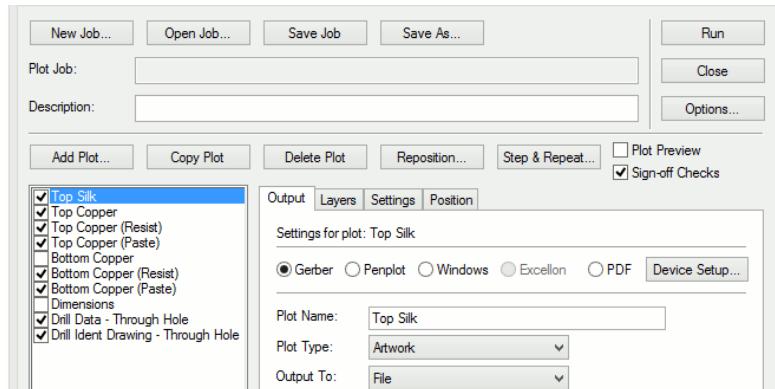
- Top, bottom and inner electrical signal layers
- Silk screen layers
- Solder resist plots
- Solder paste masks
- Copper and power plane (full or split) layers
- Assembly and manufacturing drawings
- Drill ident/symbol drawings
- NC Drilling files

DesignSpark PRO can also output to DXF and IDF formats but these are not generated as part of the Printing & Plotting mechanism.

► To output plots

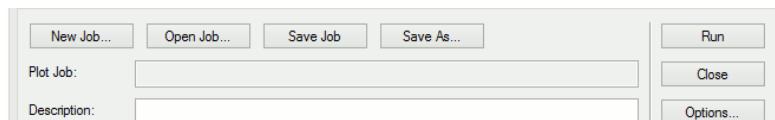
Click on the **Manufacturing Plots** option on the **Output** menu.

When you first enter this dialog, DesignSpark PRO automatically creates you a set of plots based on the layers in the design. These can be edited and amended at any time. Plots defined and customised can be saved for reuse using the **Save Job** button.

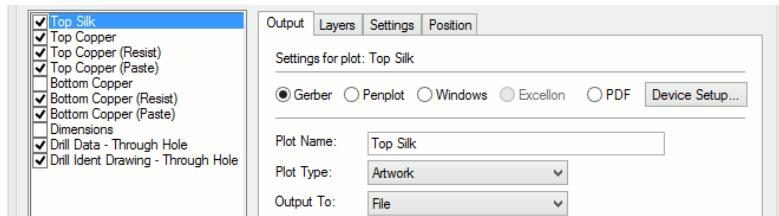


The dialog itself is split into four main sections for saving the plot settings, defining the rules of the plot, setting up the plotter type and device options.

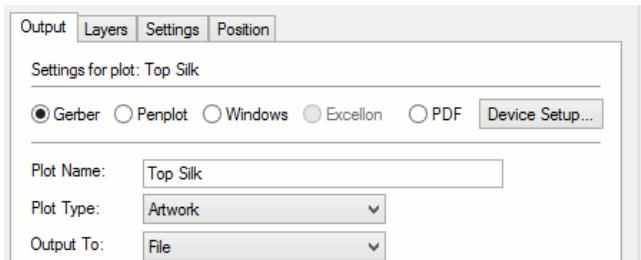
The top part of the dialog is used to save plot jobs if you customise your plots (like plotting Top Silk and Assembly together for example). You can use this to recall existing plot jobs to other designs:



The left side of the dialog is the plot to be output. This is automatically named the same as the layer but can be user-defined and edited by you. The check box indicates whether that plot/layer will be plotted when Run is pressed.

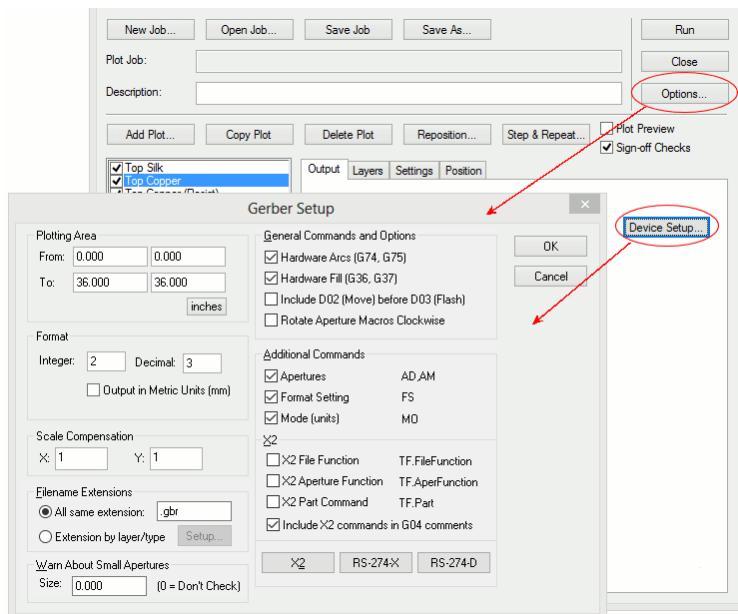


The tabbed portion of this dialog defines the plotter type used for the selected plot/layer. Each tab defines different properties of the plot, like the type, the layers included in that plot, the settings (powerplane settings, thermal sizes, pad under/oversize for solder masks etc.) and the position of the plot on the paper/film.



The **Options** button and **Device Setup** button allow you to change the settings of the selected plotter type. For example, changing from RS-274-D to 274-X format for Gerber.

The **Options** button also allows you to define which folder the plots will be plotted to.



Click the **Run** button to output all the selected plots.

Concluding The Tutorial

This concludes the tutorial. As we said above, it's a quick run through to give you an idea of the flow of the DesignSpark PRO product and how easy it is to use.

If you would like more information about DesignSpark PRO, please contact our sales office or visit our web site
<https://www.rs-online.com/designspark/pcb-software>