RS UNIVERSITY

Teach. Learn. Share.

LABORATORY MANUAL TO ACCOMPANY DESIGNSPARK PCB ONE DAY WORKSHOP

20th October 2013

Compiled by:

EDU PREP CENTRE

South-Africa

www.eduprep.co.za

TABLE OF CONTENTS

TITLE **PAGE**

LIST OF FIGURES

TITLE **PAGE**

LIST OF TABLES

TITLE **PAGE**

LIST OF ABBREVIATIONS

A		
AC	:	Alternating Current
AMP	:	Ampere
C		
CTRL	:	Control
CAD	:	Computer Aided Design
CML	:	Component Library
D		
DC	:	Direct Current
DS	:	DesignSpark PCB
F		
FPGA	:	Field Programmable Gate Array
I		
IC	:	Integrated Circuit
L		
LED	:	Light Emitting Diode
M		
mA	:	Milli-Amp (Amp/1000)
mm	:	Millimetre (meter/1000)
O		
ODW	:	One Day Workshop
P		
P	:	Pad (width)
PCB	:	Printed Circuit Board
PSL	:	PCB Symbol Library
PTF	:	PCB Technology File
R		
R	:	Reference Origin
S		
S	:	Symbol Origin
SPICE	:	Simulation Program for Integrated Circuits Emphasis
SSL	:	Schematic Symbol Library
STF	:	Schematic Technology File
T		
T	:	Track (width)
Thou	:	Thousands of an inch (inch/1000)

U

UF	:	Micro Farad (Farad/1000 000)
UV	:	Ultra Violet

V

V	:	Volt

W

W	:	Watt

1 INTRODUCTION

1.1 Scope

This Laboratory Manual documents the theory and practical exercises that accompanies the DesignSpark (DS) PCB One-Day-Workshop (ODW).

The purpose of the ODW is to provide all attendees with enough background and exposure to the various DesignSpark PCB features in the process of PCB design. This manual can either be used in conjunction with the ODW Power Point presentation (DesignSpark_ODW.ppt), or on its own as self-study material.

1.2 Applicable Item

The latest versions of DesignSpark PCB obtainable from:
www.designspark.com/pcb

1.3 Overview

DesignSpark PCB is a free professional circuit schematics and PCB layout design tool provided by RS. Among the main features such as generating schematic circuit diagrams and translating it into a PCB layout, DesignSpark PCB also allows the user to generate user defined schematic symbols as well as PCB footprints.

In the ODW a top-down approach is followed which will allow us to explore various DesignSpark features.

2 LIST OF SOFWARE REQUIRED

Table 2-1 : Software Required

ITEM No.	DESCRIPTION	DETAIL
1.	DesignSpark PCB latest version	www.designspark.com/pcb
2.	LT Spice IV	http://www.linear.com/designtools/software/#LTspice
3.	Viewmate (Gerber Viewer)	http://www.pentalogix.com/viewmate.php

3 PCB STRUCTURE

The process of manufacturing the Printed Circuit Board laminate is normally handled by a subcontractor.

The PCB CAD files (or Gerber files) are sent to the manufacturer by email. The process starts with bare laminate material which comprise woven glass (Figure 3-1), reinforced epoxy resin, with copper on both sides. The holes are drilled and chemically coated to improve the electroplating process. The laminate is placed in a copper plating bath and all the holes are electroplated, which will connect pads on both sides of the board.

The laminate is then coated with UV-sensitive photo-resist, after which the tracks are imaged onto each side using the photo-plots and exposure to UV light. The laminates are put in acid to etch away the unrequired copper, thereby forming the track pattern. The bare copper PCB, with tracks and pads, is cleaned and is now ready for the solder masking and tinning. The bare copper is then solder masked (which is typically a green colour, but could also be white, blue or red) by means of either silk-screening, photo-imaging or dry film. The exposed pads are then tinned or plated with solder, silver or gold. The bumps in the solder are made flat by hot air, which is called hot-air-levelling. The white painted component identification and any board lettering and annotations are finally added by means of a silkscreen process.

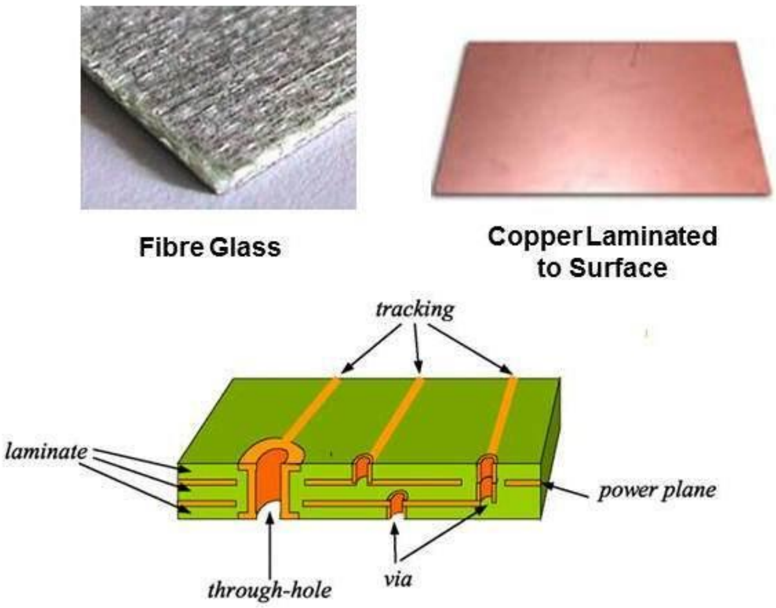


Figure 3-1: Layer Construction of a PCB

The main parts of a PCB can be seen in the annotated picture Figure 3-2. The mounting holes are plated through, but isolated from the circuitry. The Pads are the copper surface that the components will be soldered to, and is either surface mount or Through-hole.

The solder mask is a thin polymer coating on the board, which surrounds the pads to help prevent solder from bridging between the pins. The size of track to be used depends on the electrical requirements of the design, the routing space and clearance available, and also personal preference (note - the bigger the track width, the better).

Vias and through-holes are holes in the laminate with electroplated copper on the surface, forming a bridge between two (or more) conducting layers. Line art and text indicating board identification, component designation & outline, pin identification, test points, etc. are printed onto the outer surfaces of a PCB by silkscreen, which is also called screen printing.

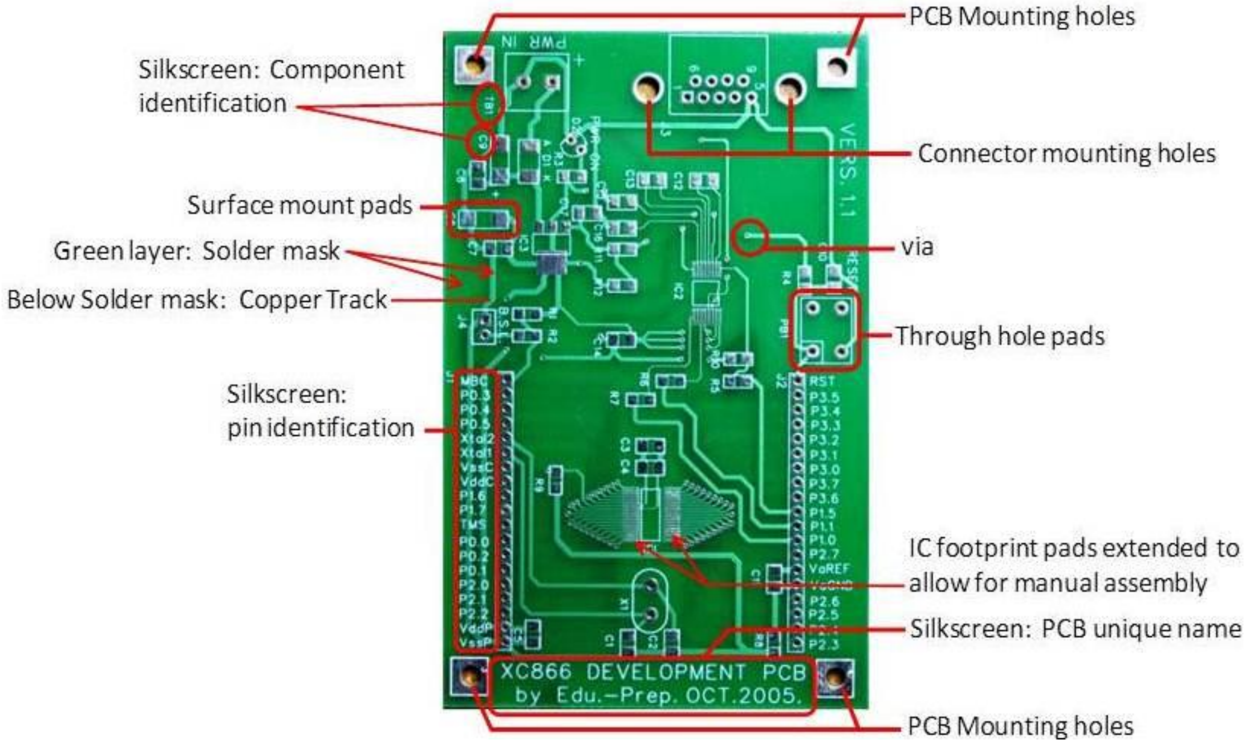


Figure 3-2: PCB Structure

The datasheet of a component will normally specify all its physical dimensions, but could sometimes only refer to a standard package (i.e. DO-41 see Figure 3-3). The dimensions are then specified in either mm, inches or both - so we must be able to work with both dimensions.

To get a feel for the relation between inches and mm, we refer to Figure 3-4 showing that 1 inch = 25.4mm. Because tracks are merely fractions of an inch, we are introduced to a new dimension which is 1/1000th of an inch and is known as "thou". This therefore means that a "thou" is equal to 25.4mm/1000 or more commonly 0.0254mm.

As a general rule we use "thou" when working with track width, pad dimensions, track and pad spacing, and we use "mm" when working with board dimensions and hole sizes.

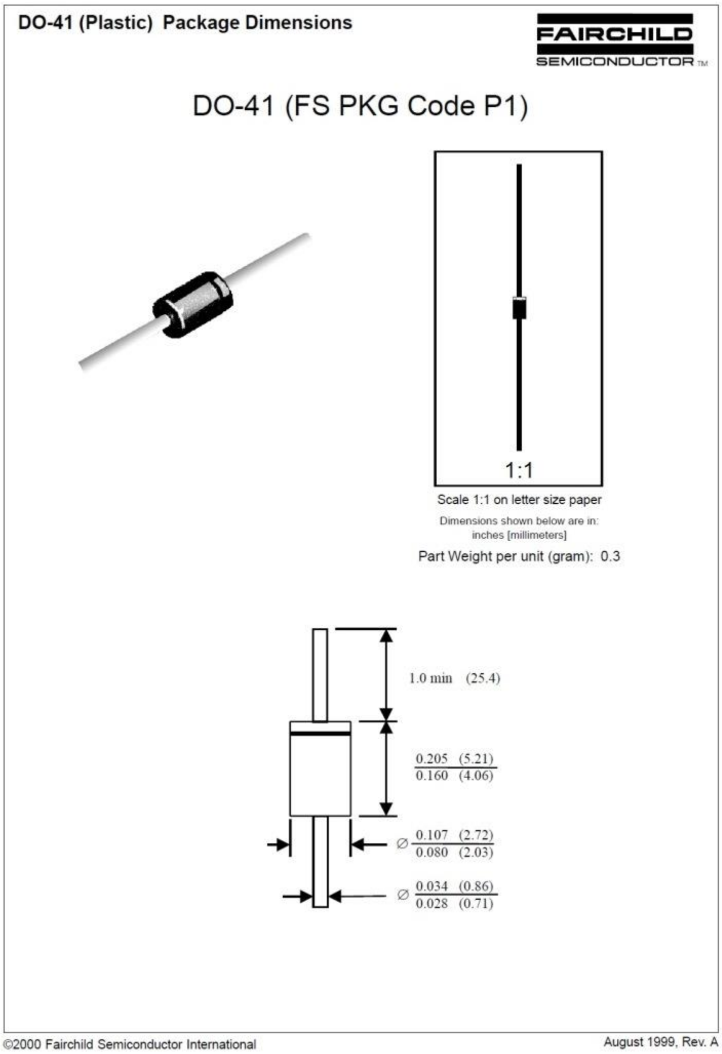


Figure 3-3: Standard DO-41 package dimensions

Figure 3-4: Important dimensions

4 SCHEMATIC DESIGN

A schematic is a presentation of the element-by-element relationship of all parts of a system. To ensure a consistent and good general look for a schematic diagram, the following items need to be considered:

- Relative Symbol Size
- Component Reference (e.g. U1)
- Component Value (e.g. 100uF)
- Component Power or Voltage Rating
- Relative Text Font & Type
- Line size (consistent with component Pin size)
- Inputs/Outputs on either side or on same side
- A4 Layout, Name, Number, Revision
- Pull-up Resistors
- Decoupling Capacitors
- Circuit partitioned into logical/functional blocks
- Good Grounding

Figure 4-1 is a simple schematic diagram showing a few of the points described above. What can be seen is the following:

- Relative Symbol Size,
- Reference (e.g. U1),
- Component Value (e.g. 4700uF),
- Component Power or Voltage Rating (e.g. Cap 35V / Res 0.5W),
- Relative Text Font, Line size consistent with component Pin size,
- Input on the left & Output on the right,
- A4 Drawing Layout,
- Name,
- Number,
- Revision,
- Circuit partitioned into logical/functional blocks, and

- Ground.

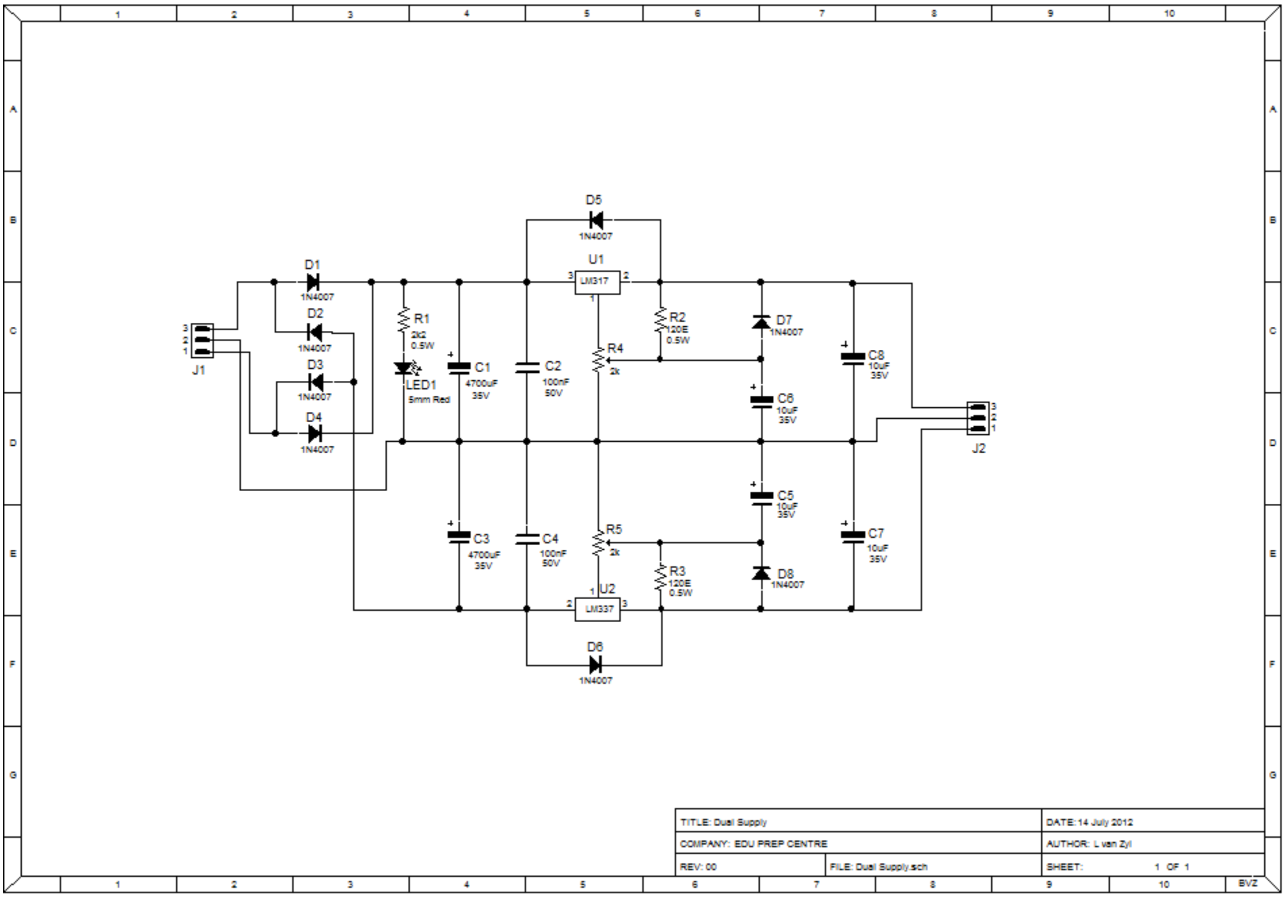


Figure 4-1: Schematic Design example

5 PCB DESIGN

The PCB contains the electrical interconnections between the components. Always attempt to keep the tracks as big as possible because they have lower DC resistance, as well as lower inductance, and is easier to manufacture (therefore cheaper). Limitations on the lower limit of the track sizes are due to manufacturing constraints. The manufacturer will specify the track-to-track as well as pad-to-pad and pad-to-track limits. Typically 8 thou or 0.2mm would be the preferred minimum for both, although some manufacturers can produce boards with spacings of 6 thou and less.

Therefore, keep the tracks big and only reduce the size to achieve the required manufacturable clearances. For certain applications though, the current carrying capability will require bigger track widths.

Always work on a grid setting, that would assist in connecting the different components (especially ICs with a high-pitch pin spacing). DesignSpark PCB "Snap Mode" is ideal for fractions of a Working Grid Step Size. Working on grids will assist in achieving a professional look. When selecting "Snap Mode", a grid setting of either 100 thou, "Half Grid" (50 thou), "Quarter Grid" (25 thou) etc. may be selected.

A via is similar to a through-hole pad but is much smaller. It is used to connect two or more points in the circuit situated on different layers.

A via that bridges two layers inside the PCB, which is not be visible at the surface of the PCB, is called a *blind* via. A via may have very little copper around it – called a *landless* via. Remember not to cover the vias with the solder mask to allow for easy access when debugging a circuit. It is also a handy connection point when a PCB needs to be modified.

To ensure a consistent and a good general look for a PCB Layout, the following items need to be considered:

- Footprint dimensions
- Component outline (silkscreen)
- Component Placement (symmetrical / access to components)
- Component References
- Annotations and Guidelines
- Unique Identification of PCB

- Mounting Holes
- Track / Pad size relevant to current carrying requirements
- Miter Tracks (good looks)
- Test Point if required
- Pin Numbers on Silkscreen
- Layout Partitioning
- Thermal Considerations
- Copper Pour

Figure 5-1 is a simple PCB layout showing a few of the points described above, such as:

- Footprint dimensions,
- Component outline (silkscreen),
- Component Placement (symmetrical / access to components),
- Component References,
- Annotations and Guidelines,
- Unique Identification of PCB,
- Mounting Holes,
- Track / Pad size relevant to current carrying requirements,
- Miter Tracks (good looks),
- Pin Numbers on Silkscreen,
- Layout Partitioning,
- Thermal Considerations, and
- Copper Pour.

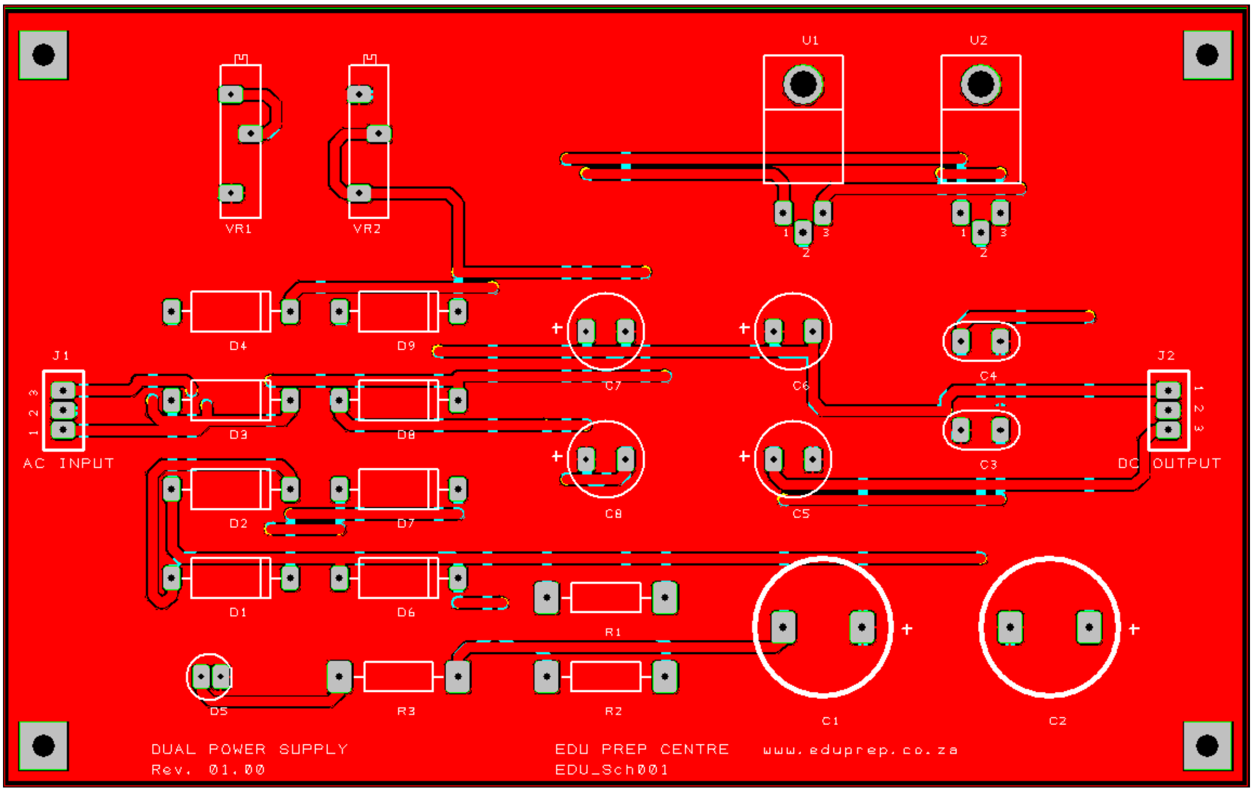


Figure 5-1: PCB Layout example

6 EXERCISES

6.1 Exercise No. 1 - Simulation

6.1.1 Background and Discussion

In general, "Simulation Program for Integrated Circuits Emphasis" (SPICE) allows for circuit analysis, using:

- Non-linear DC analysis,
- Non-linear transient analysis,
- Linear AC Analysis,
- Noise analysis,
- Sensitivity analysis,
- Distortion analysis,
- Fourier analysis, and
- Monte Carlo Analysis.

DesignSpark PCB has built-in SPICE parameters that can be assigned to components, so that a number of circuit elements could be output to a number of standard SPICE simulators, such as:

- LsSpice,
- LTspice,
- B2spice and
- TINA.

As an example to show the DesignSpark SPICE output capabilities, we will explore the results using LTspice. A number of components come standard with DesignSpark PCB in the component library "ltspice.cml", which is what we will use in our exercise.

Mesh current calculations form part of most first year network theory courses.

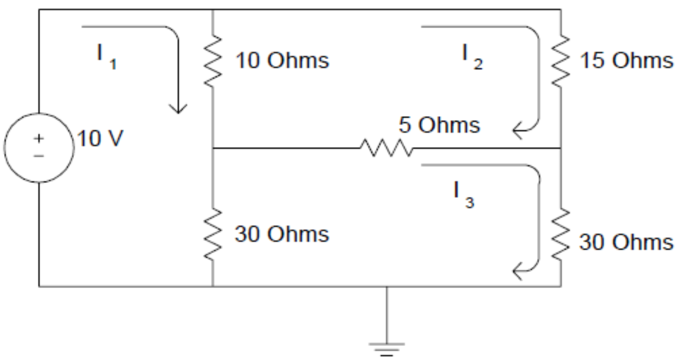
For the Figure 6-1, the first loop I_1 starts from the top of the 10Ω resistor, through the 30Ω resistor, then through the 10V supply voltage back to the starting point.

The second loop I_2 starts at the junction of the 10Ω, 5Ω and 30Ω resistors, after which it flows up through the 10Ω, then through the 15Ω and finally through the 5Ω stopping at the junction again.

The third loop I_3 starts at the grounded side of the 30Ω resistor, flowing up through the 30Ω resistor, then turns direction right through the 5Ω resistor and finally through the 30Ω again, stopping at the grounded side of the 30Ω resistor.

The example matrix (AI=B) is for the three loop network which results in the following, by solving for the current matrix I, i.e. I = inv(A) * B:

I1=475mA, I2=198mA and I3 = 235mA (Current through 10Ω = I1-I2 = 277mA, through 5Ω = I2-I3 = 37mA, through 30Ω = I1-I3 = 240mA)



The loop equations are

Loop 1,

$$10(I_1 - I_2) + 30(I_1 - I_3) - 10 = 0$$

Therefore $40I_1 - 10I_2 - 30I_3 = 10$

Loop 2,

$$10(I_2 - I_1) + 15I_2 + 5(I_2 - I_3) = 0$$

Therefore $-10I_1 + 30I_2 - 5I_3 = 0$

Loop 3,

$$30(I_3 - I_1) + 5(I_3 - I_2) + 30I_3 = 0$$

Therefore $-30I_1 - 5I_2 + 65I_3 = 0$

A I = B

$$\begin{bmatrix} 40 & -10 & -30 \\ -10 & 30 & -5 \\ -30 & -5 & 65 \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \\ I_3 \end{bmatrix} = \begin{bmatrix} 10 \\ 0 \\ 0 \end{bmatrix}$$

Figure 6-1: Mesh current Calculations

6.1.2 Requirements

Table 6-1 : Exercise 1 Files Required

ITEM NO.	FILE NAME	FILE SIZE
1.	Mesh_Example.sch	14KB

6.1.3 Exercise 1 Procedure

1. We want to simulate the above circuit (Figure 6-1), therefore open DesignSpark PCB (right click on the desktop icon and select open, or double-click on the icon) and open the file “Mesh_Example.sch” as shown in Figure 6-2.

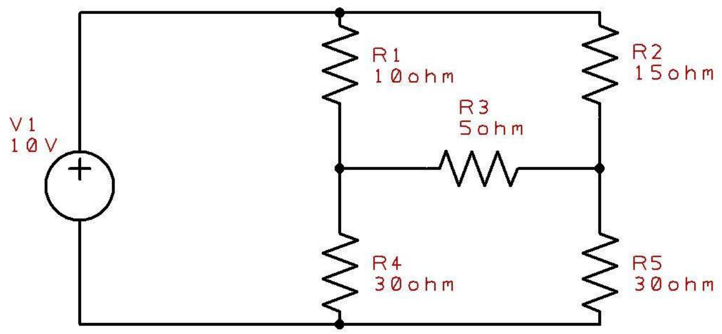


Figure 6-2: Mesh Current Simulation

2. Select the “Output” menu and “Spice Simulation Output” as shown in Figure 6-3 image 1.
3. Select “LTspice” from the “Simulator Format”.
4. In the “Program” field, browse to where the program is installed, as shown in Figure 6-3 image 2.
5. Selecting the “OK” button brings us to the “LTspice Simulation Output” selection screen as per Figure 6-3 image 3.
6. This screen allows for the different types of analysis as previously mentioned. We will be examining the plot outputs, so simply select the “OK” button.
7. LTspice will be opened with a blank plot screen. In LTspice on this plot screen, right click anywhere and select “Visible Traces” to choose either individual current values, or more than one, by holding in the <CTRL> key while selecting.
8. Now compare the obtained simulated results with the expected values.

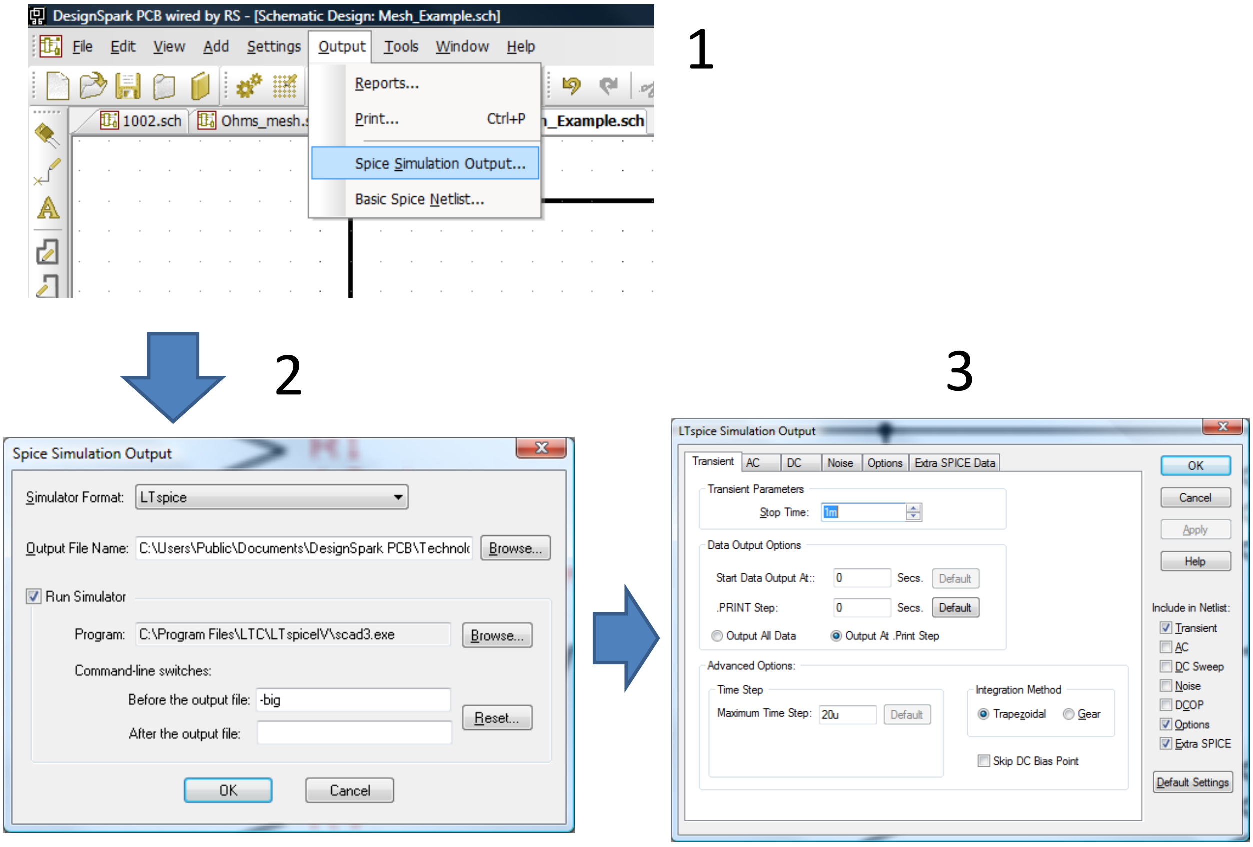


Figure 6-3: Simulation

6.1.4 Exercise 1 Summary / Conclusion

DesignSpark PCB includes many LTspice components for SPICE simulation, under the library “ltspice.cml”.

DesignSpark PCB outputs the simulation files required by LTspice to run simulations, and the “LTspice Simulation Output” window within DS PCB allows the user to select between the different circuit analysis options to be output.

The library components can either be used as is, or the user can also define user-defined components by generating new components and saving them as SPICE devices (Package = SPICE).

6.2 Exercise No. 2 – Manufacturing Files

6.2.1 Background and Discussion

Now that we have seen a typical DesignSpark PCB Schematic, it is time to quickly look at some of the main features, before we attempt our first design.

There are three parts to a component namely:

- a Schematic Symbol,
- a PCB Symbol (footprint) and
- a Component, which is a combination of the two as depicted in Figure 6-4.

There are therefore three libraries containing the detail for each of these parts as shown in Figure 6-4.

Technology files can be used to provide the technology data such as units, grids and styles when creating library symbols. Technology files provide a means of setting most of the important parameters when starting a new design, or when first performing a translate operation from schematic to PCB.

We want to take a look at the detail contained in the technology files, but first let's see how we access the Schematic symbol, PCB symbol and the Component in DesignSpark PCB.

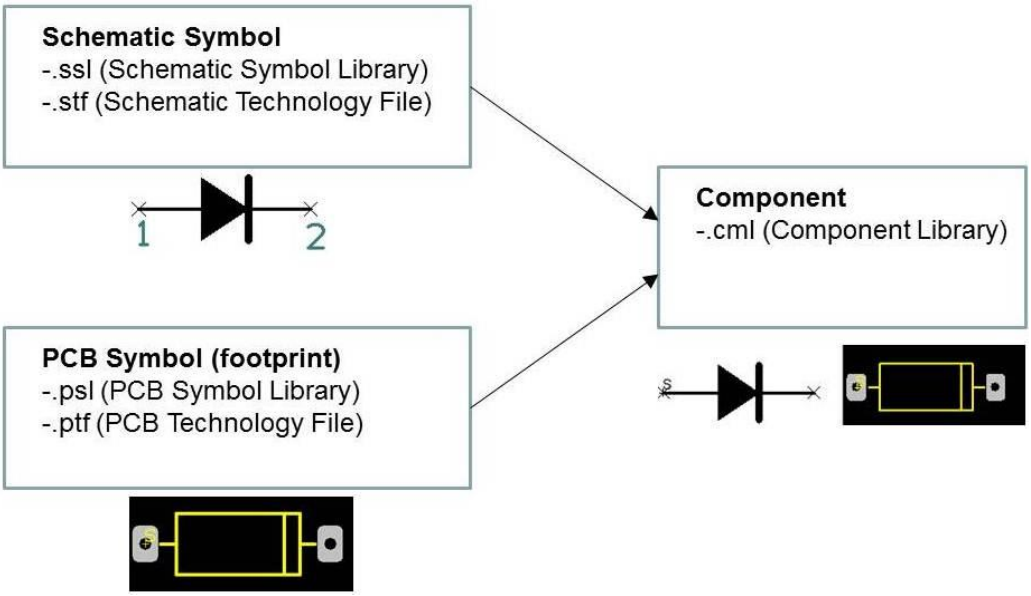


Figure 6-4: DesignSpark PCB Libraries

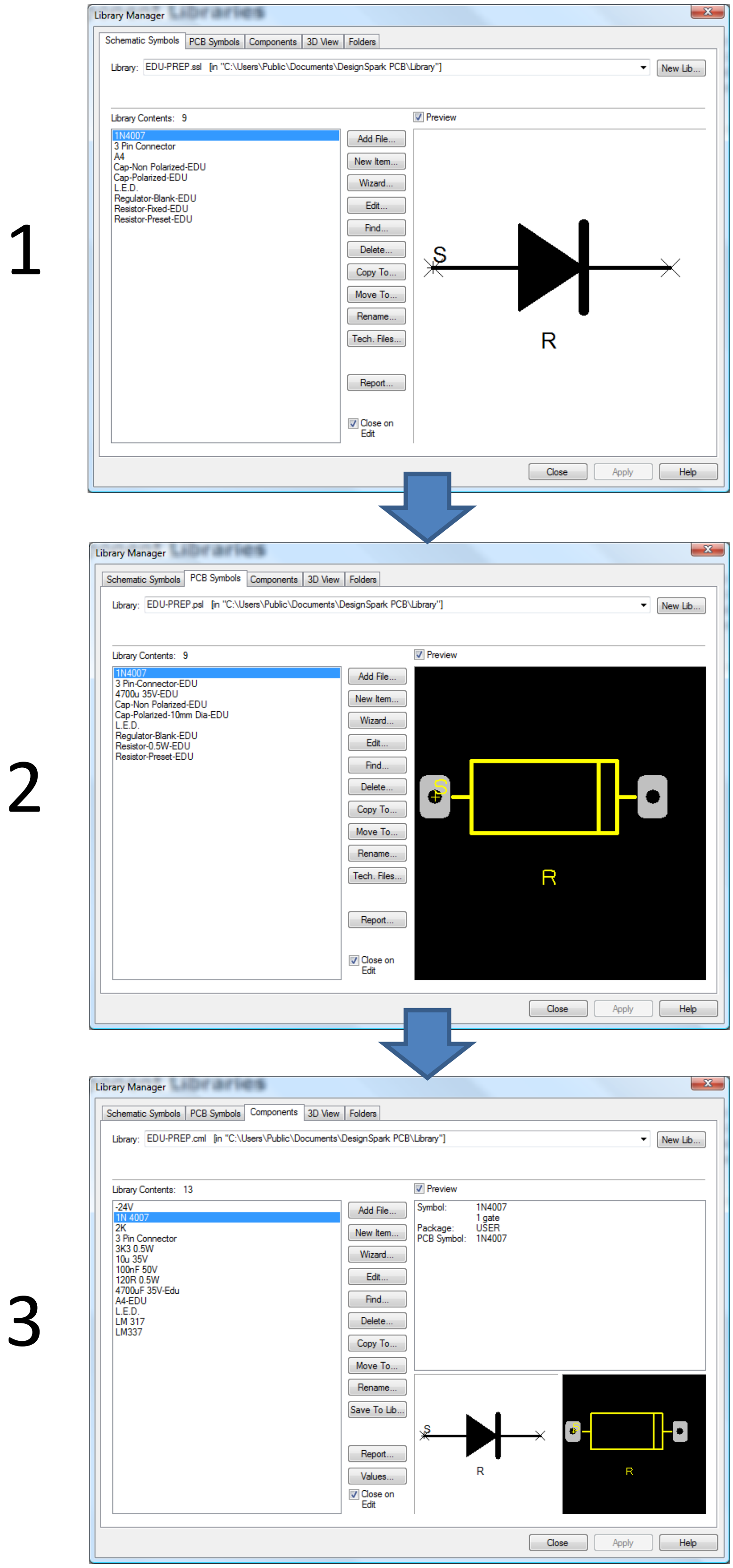


Figure 6-5: DesignSpark PCB Library Manager

Figure 6-5 above shows the Library Manager with the three different views of a common through-hole diode, when selecting:

- Schematic Symbols (image 1),
- PCB Symbols (image 2) and
- Components (image 3).

DesignSpark PCB general settings are stored in the Technology files.

Everything related to the Schematic is stored in the Schematic Technology File (.stf), and similarly all PCB settings are stored in the PCB Technology File (.ptf), as shown in Figure 6-6.

Matching Schematic and PCB technology files can be created to enable you to define your own Net classes, and to enable them to be transferred from Schematic to PCB.

They also provide a means of defining complete setups for particular design processes, and include everything from:

- display colours,
- screen and working grids,
- standard net names (optional),
- standard dimensions for all drawing parameters, such as tracks, connections, component pads and text, Design Rule clearances, Board outlines and Board layers used, and even the standard setups for the optional range of auto-routers.

In Figure 6-6, Image 1 shows the Design Technology related to the Schematic Design, and Image 2 shows the PCB Design Technology detail.

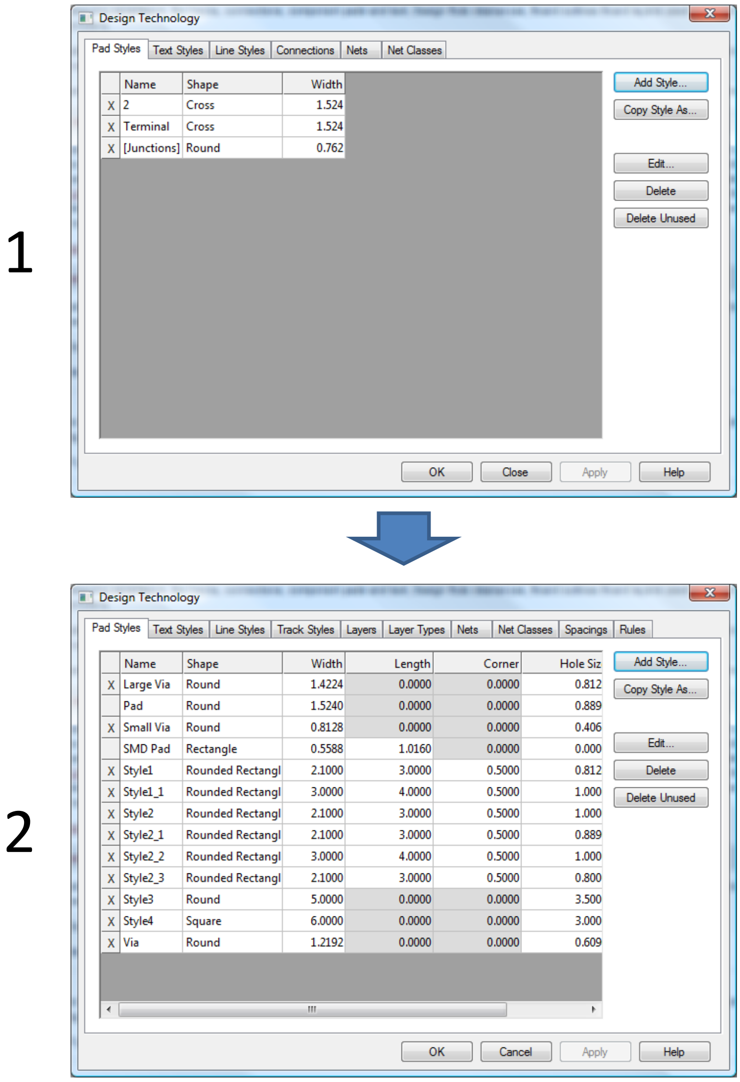


Figure 6-6: Technology Files – PCB Design Technology

Grids, Units and Coordinates (as shown in Figure 6-7) are also stored in the relevant Technology file, and can be accessed and changed in the Settings tab.

Remember to save the Technology file each time you make changes in the event that you want to keep it for future use.

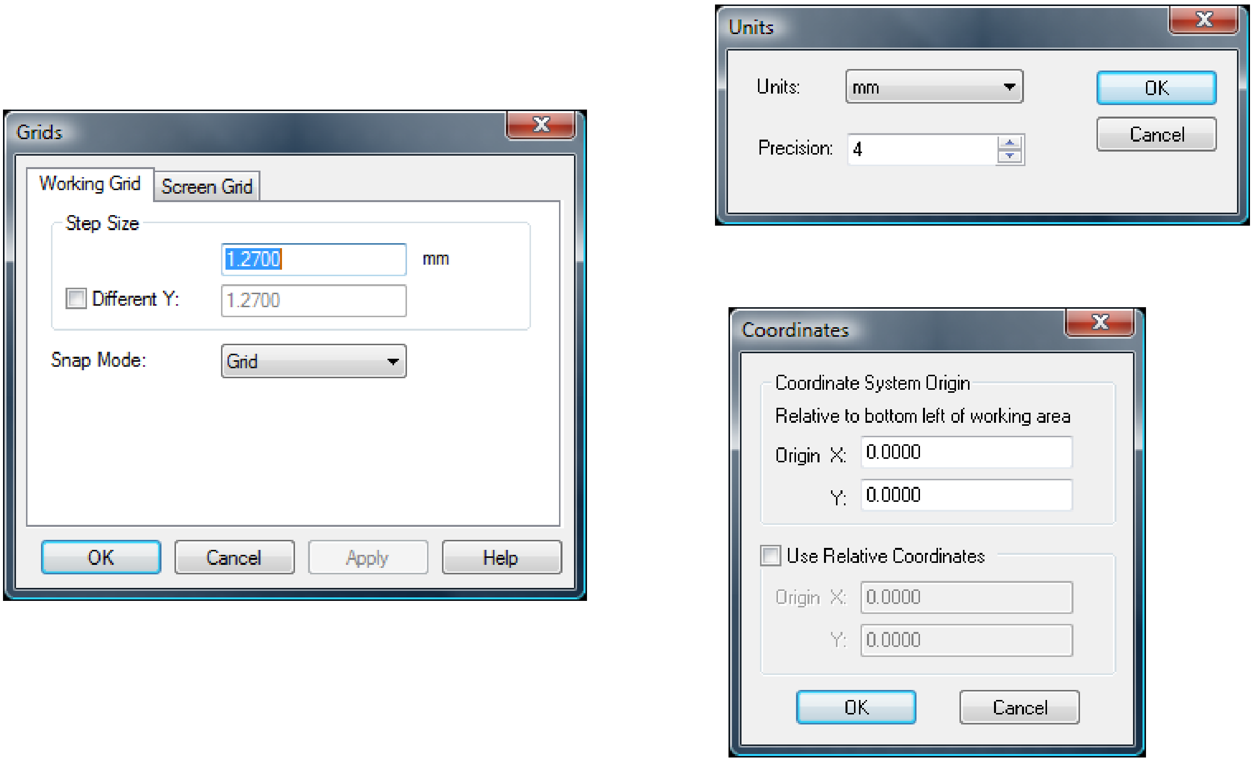


Figure 6-7: Examples of Settings

6.2.2 Requirements

Table 6-2 : Exercise 2 Files Required

ITEM NO.	FILE NAME	FILE SIZE
1.	riaa amp.pcb	23KB

6.2.3 Exercise 2 Procedure

We will start at the top level by generating the Gerber files from an existing design.

1. First open the file “riaa amp.pcb”.
2. To output the Gerber files, select “Output” - “Manufacturing plots” or press <Shift-P> as shown in Figure 6-8 (image 1).
3. Select the “Auto-Gen Plots” (image 2).
4. Select “Gerber” Device Type (image 3) and select the “OK” button (image 3).

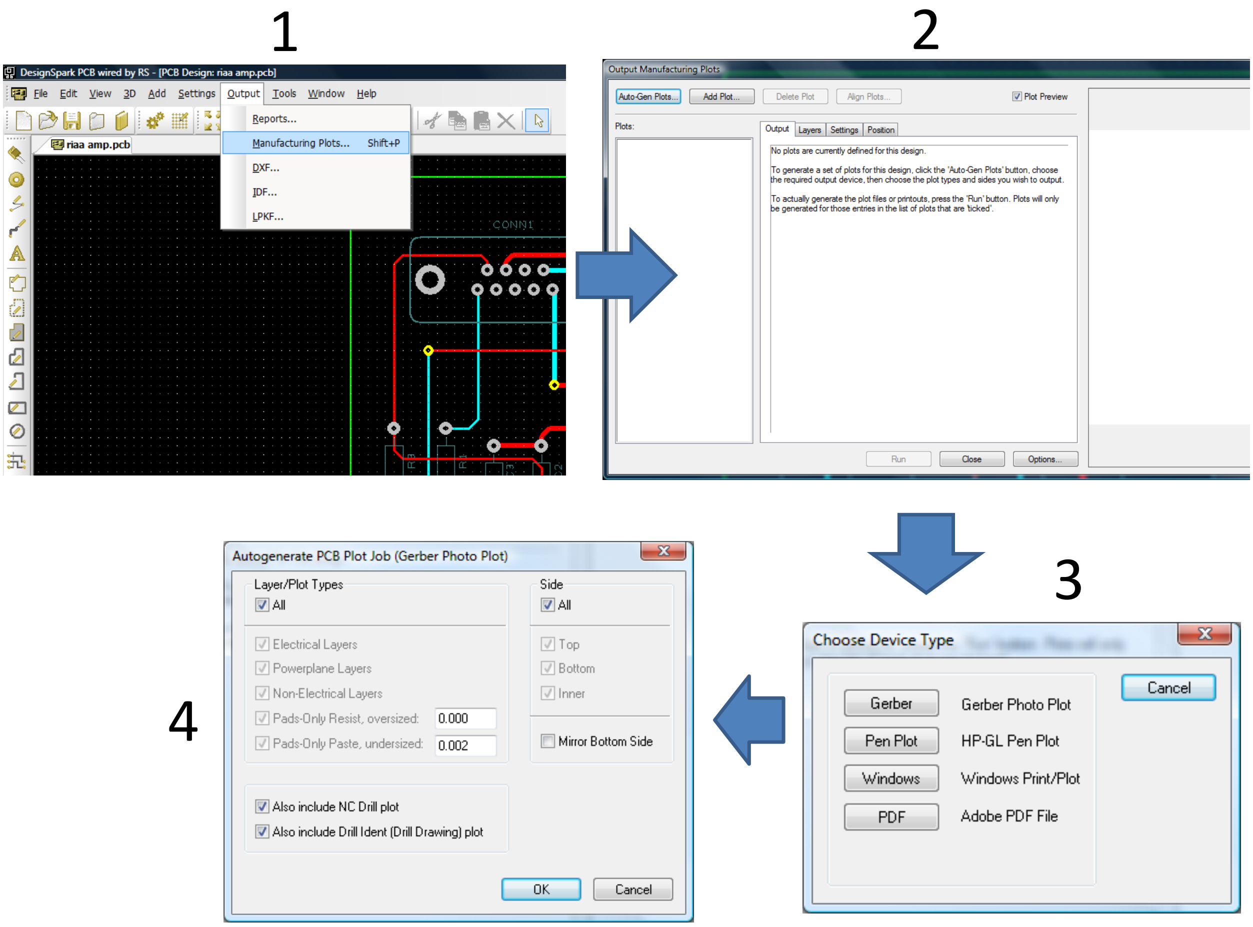


Figure 6-8: Manufacturing Files 1

5. Figure 6-9 (image 1) shows the different plot outputs as well as the formats, for which we require the Gerber format.
6. Image 2 indicates that clicking on each layer on the left will display the relevant layer that will be output.
7. Under the “Settings” tab (image 3), all the settings related to each layer can be viewed by selecting the layer on the left.
8. Remember to select the “Plated Board Outline” and “Unplated Board Outline” options as shown in image 4.
9. Select the “Run” button which produces all the plot files. A free viewer such as “Viewmate” can be used to look at the plots as can be seen in the Figure 6-10.

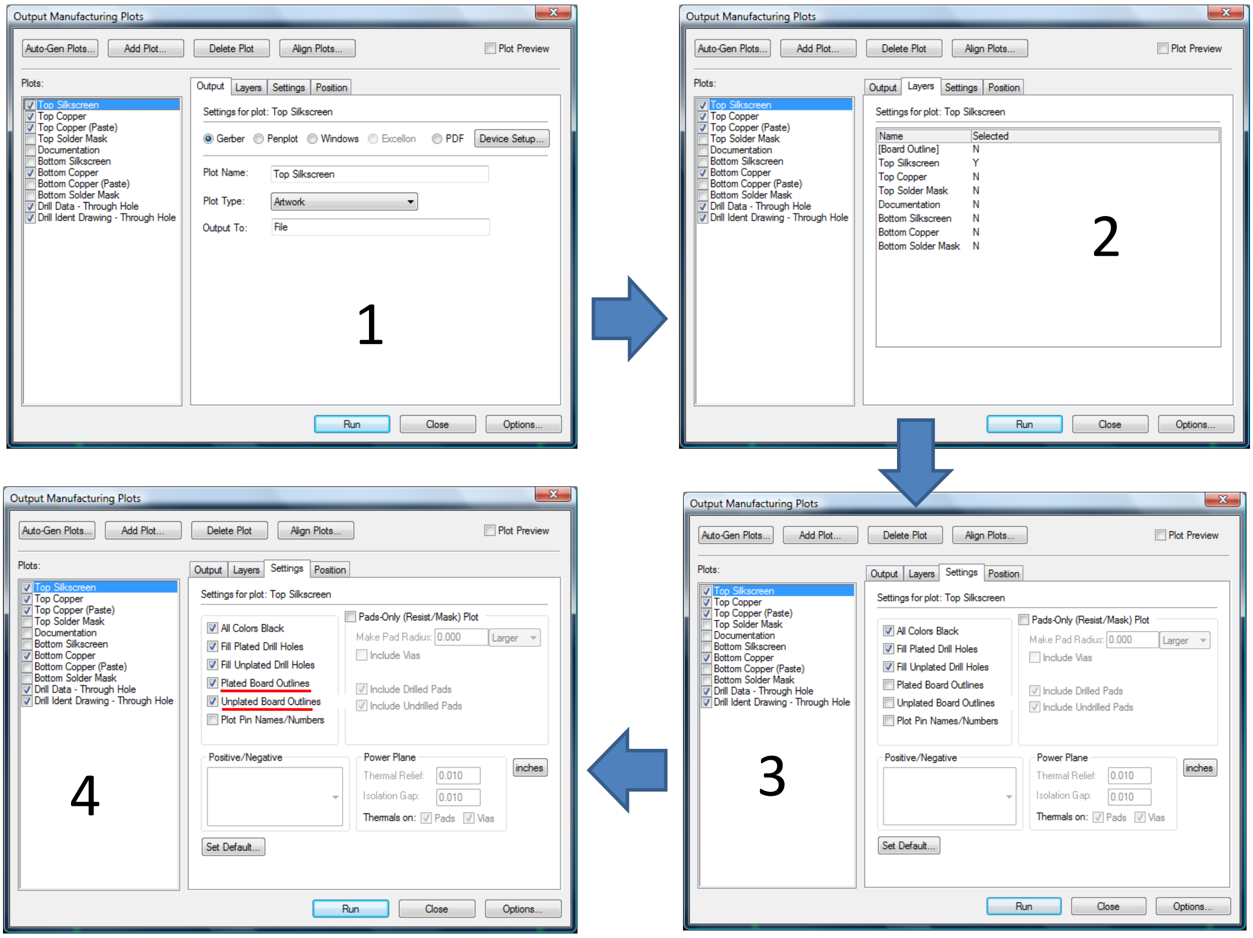


Figure 6-9: Manufacturing Files 2

10. Open “PentaLogix Viewmate” and select “File” – “Import” – “Gerber” or press <F2>.
11. Select all the files previously produced by keeping in <Ctrl> while selecting.
12. Click on “Import” to open all files.
13. Now we can select and de-select the different layers on the left, by clicking on the tick boxes as shown in Figure 6-10. These are the files that are required by the PCB subcontractor for manufacturing.

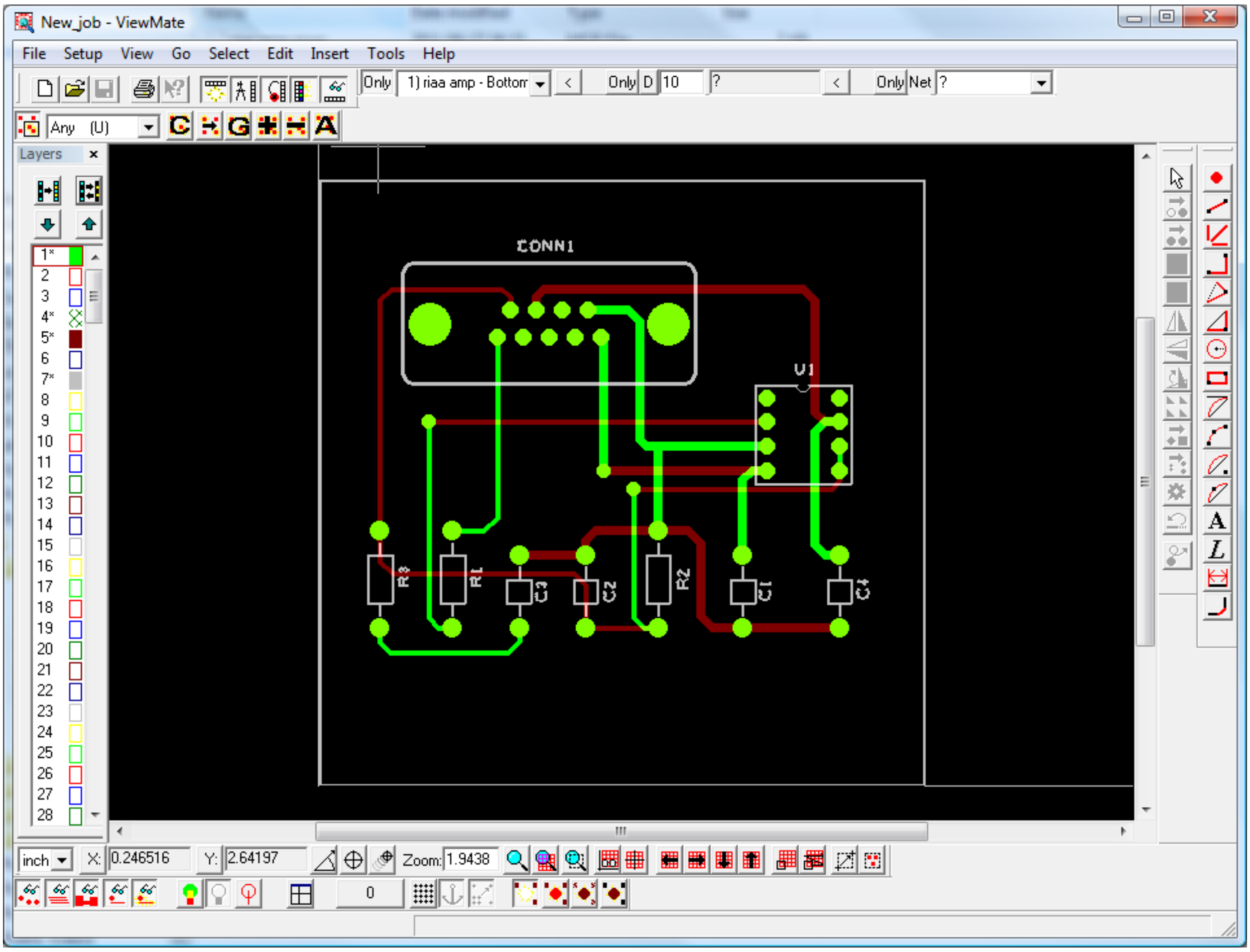


Figure 6-10: Manufacturing Files 3

6.2.4 Exercise 2 Summary / Conclusion

Once the PCB design is completed, the process for generating manufacturing plots will always be the same and is really straight forward.

The one thing to remember is to ensure that the “Plated Board Outlines” and “Unplated Board Outlines” are selected, so that the board outline dimensions are available to the manufacturer.

The files generated can be zipped into a folder and can be sent to the manufacturer by means of an email.

6.3 Exercise No. 3 – Generating PCB Files

6.3.1 Background and Discussion

Moving down one level, we will now be translating a schematic to a PCB and route the layout.

We will be using an existing completed schematic design which we want to translate to the PCB design, and then auto-route the tracks.

Sometimes the PCB or part of the PCB must be hand-routed, and DesignSpark PCB allows for either of the two or a combination of both.

In this exercise we will only be using the auto-routing capability.

6.3.2 Requirements

Table 6-3 : Exercise 3 Files Required

ITEM NO.	FILE NAME	FILE SIZE
1.	riaa amp.sch	28KB

6.3.3 Exercise 3 Procedure

1. Open the schematic file “riaa amp.sch”.
2. As shown in Figure 6-11 (image 1), select “Tools” and “Translate to PCB” which will open the “Schematic To PCB Wizard”.
3. Select the “Next” button, and choose the default technology file, and select “Units” in mm with “precision” 4. Precision is used to set the number of decimal places reported for units.
4. If we wanted to design a two layer PCB, we would select the settings as in image 2. Remember not to cover the vias with solder mask.
5. We can now define the board dimensions as in image 3.
6. It is good design practice to position all the components outside of the board area as shown at the top of image 4, to allow the designer to first place the components with mechanical constraints, followed by the bulky components like microcontrollers and FPGAs.
7. For this example we do not want to save or overwrite the existing PCB layout, so de-select the “Save the PCB design to the file” check-box as in image 5.

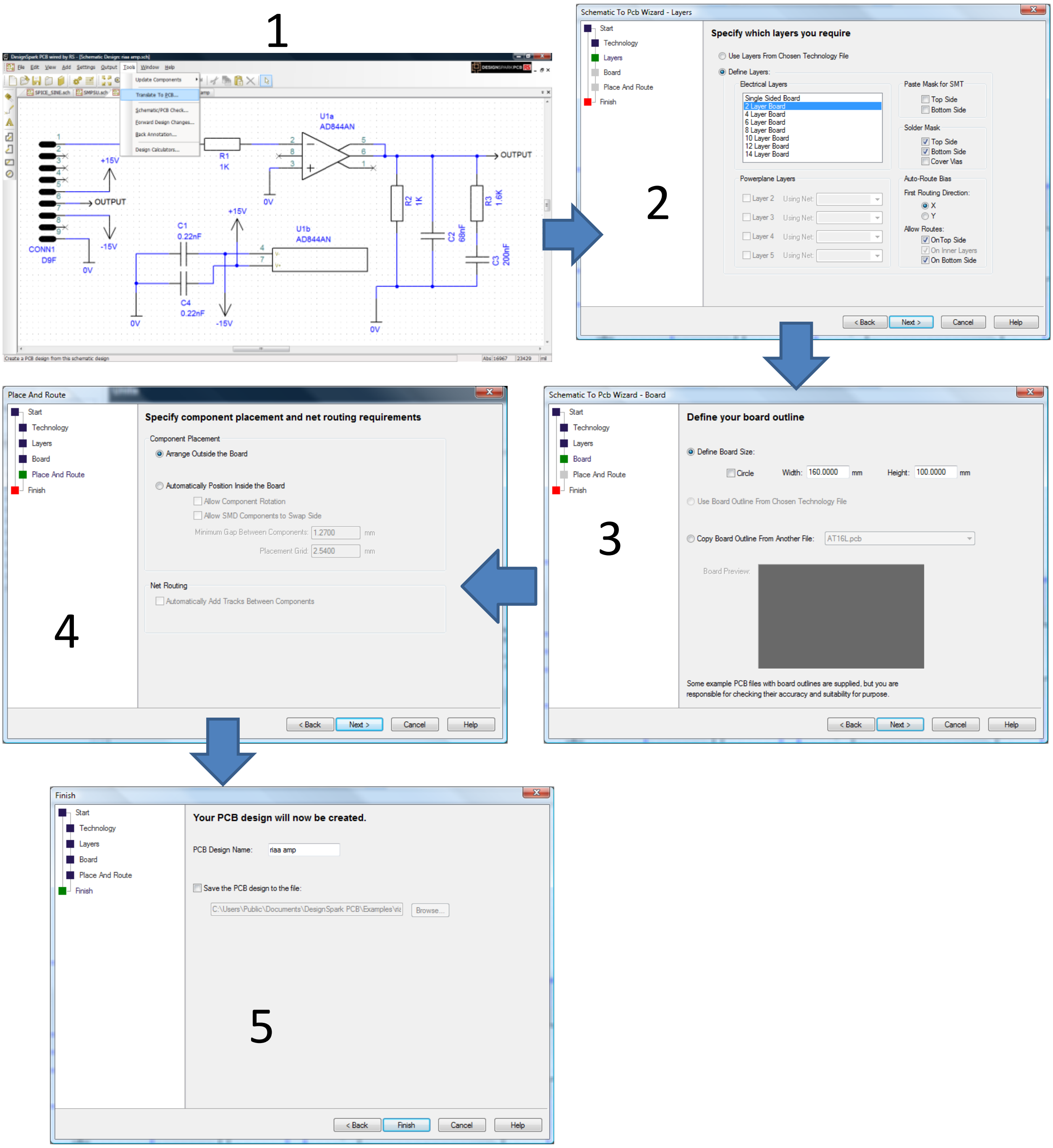


Figure 6-11: Generating PCB Procedure

8. The result is that the board as previously specified, is shown with all the components placed on the outside of the board as shown in Figure 6-12.
9. The connections between the components are often referred to as the rubber bands or the rats nest. The components can now be selected and moved onto the board.

10. After you are pleased with the component placement, we are ready to route.

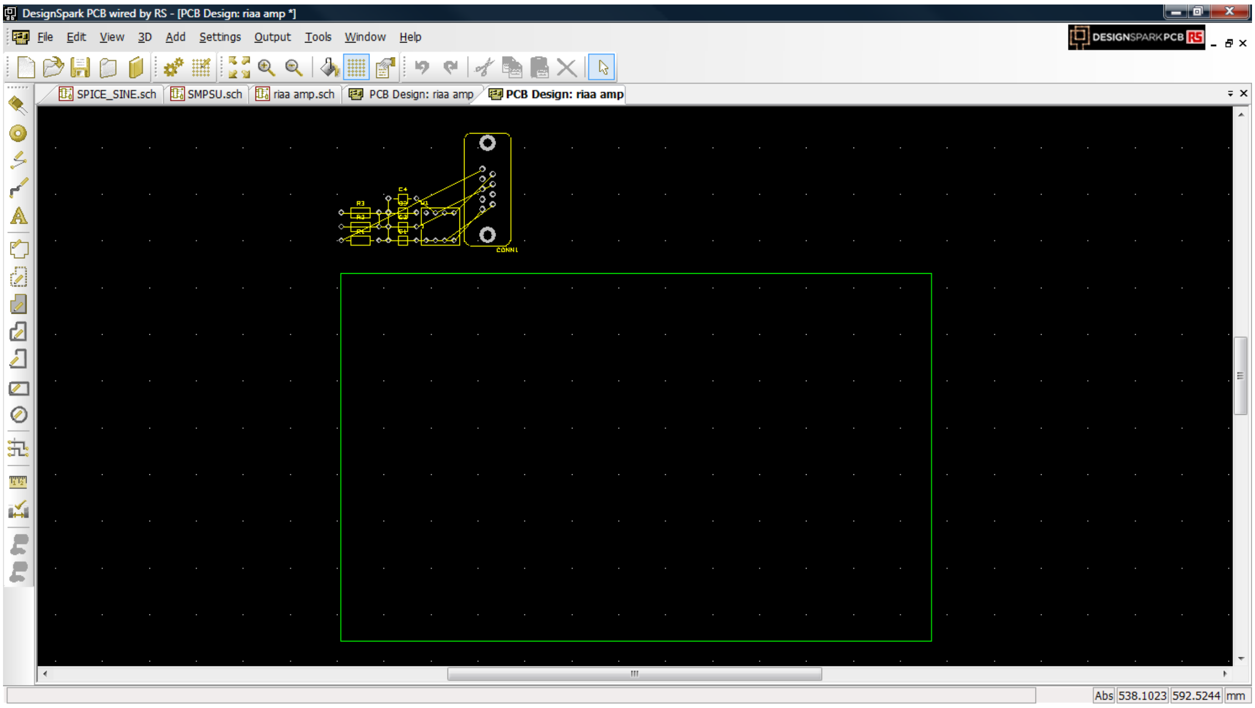


Figure 6-12: PCB Component Placement

11. Select the “Route All Nets” icon as shown in Figure 6-13, which will result in the pop-up screen as shown on the right.
12. Remember to select “Miter Track” (meaning bevelling each of two points to be joined at a 45° angle) and select the “Route” button.
13. Once you are pleased with the layout, it can be saved and Gerber files can be generated as done in Exercise 2.

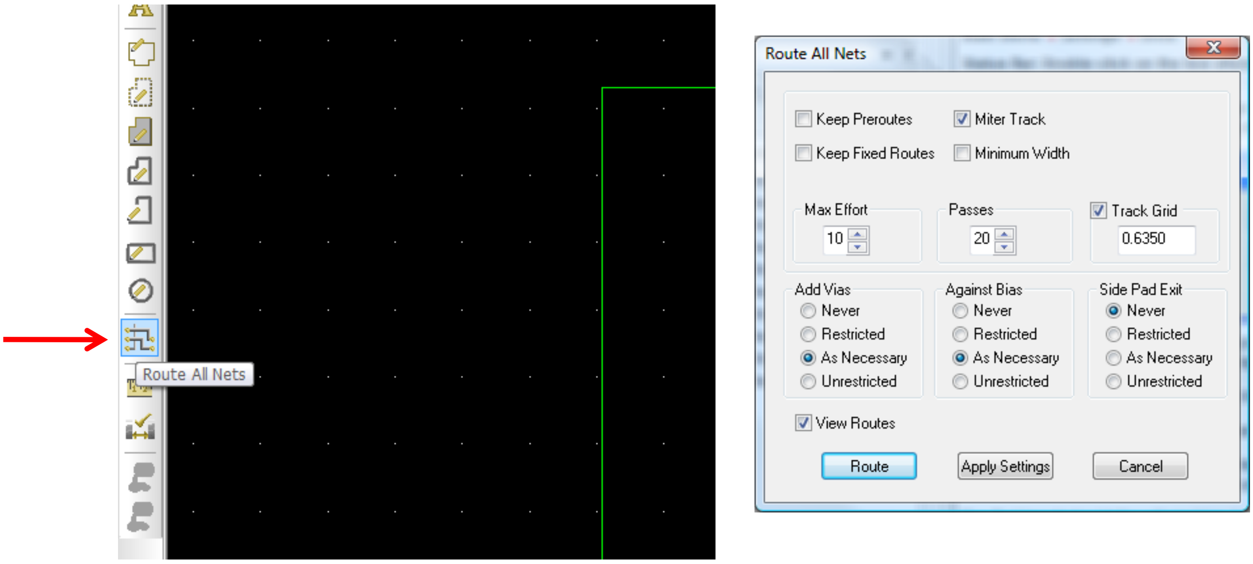


Figure 6-13: Auto-Routing the PCB

6.3.4 Exercise 3 Summary / Conclusion

Once a schematic circuit diagram needs to be translated to a PCB, the Schematic to DS PCB Wizard allows the user to select the various parameters such as the number of board layers, board size and filename.

Once the parameters are specified, the user can place the components (connected with rubber-bands) and auto-route the PCB.

6.4 Exercise No. 4 – Generating Schematics

6.4.1 Background and Discussion

In this exercise we will generate a schematic diagram from existing components and explore some of the DS PCB settings.

A schematic is a diagram showing the interconnections between components, which include the component identification, its reference as well as voltage and power ratings.

To generate a schematic circuit diagram, the user selects the “Schematic Design” option as shown in Figure 6-14 (image 1) from the “File”-“New” drop-down menu, which will create a new file with a blank display (image 2).

To change the colour appearance, select “View” from the top menu bar and then select “Colours” (image 3).

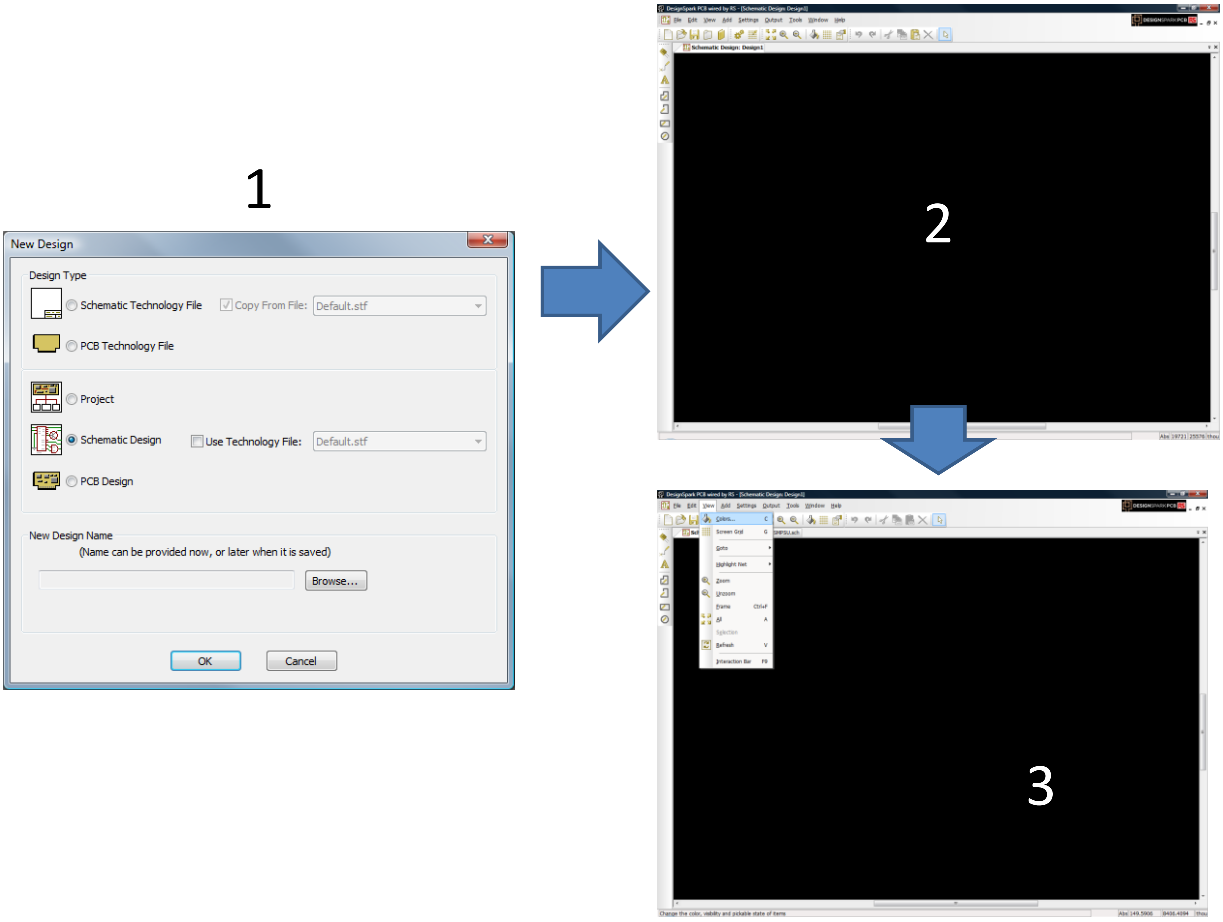


Figure 6-14: Generating Schematics

The colour scheme can arbitrarily be chosen. As a guide, choosing the colours as shown in Figure 6-15 will produce an easily-readable schematic as can be seen in Figure 6-16.

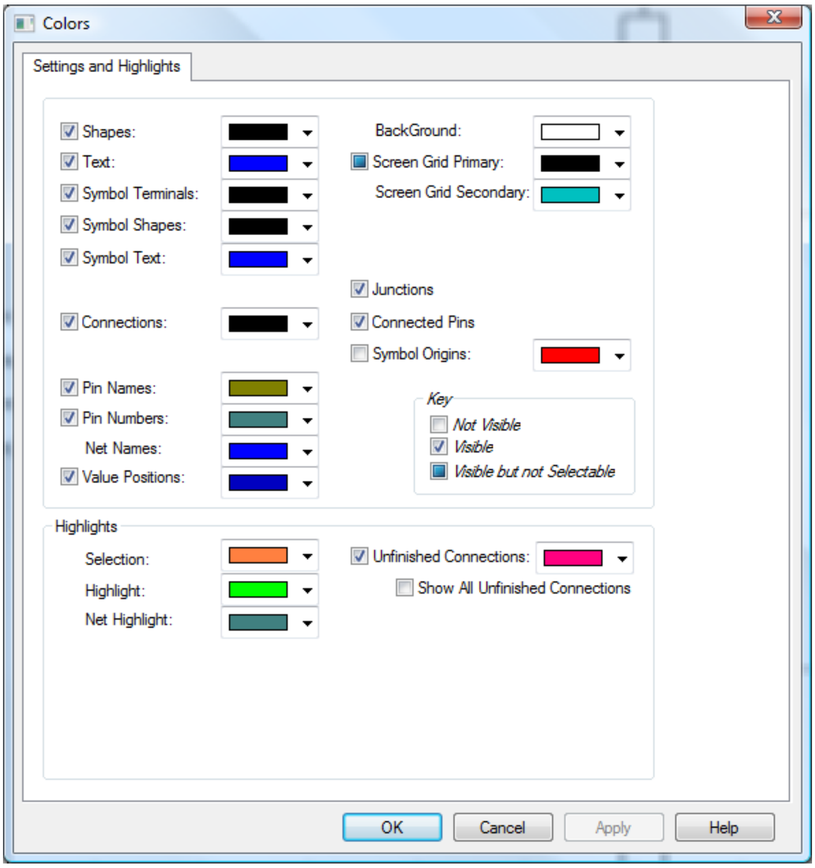


Figure 6-15: Generating Schematics – Colors Settings

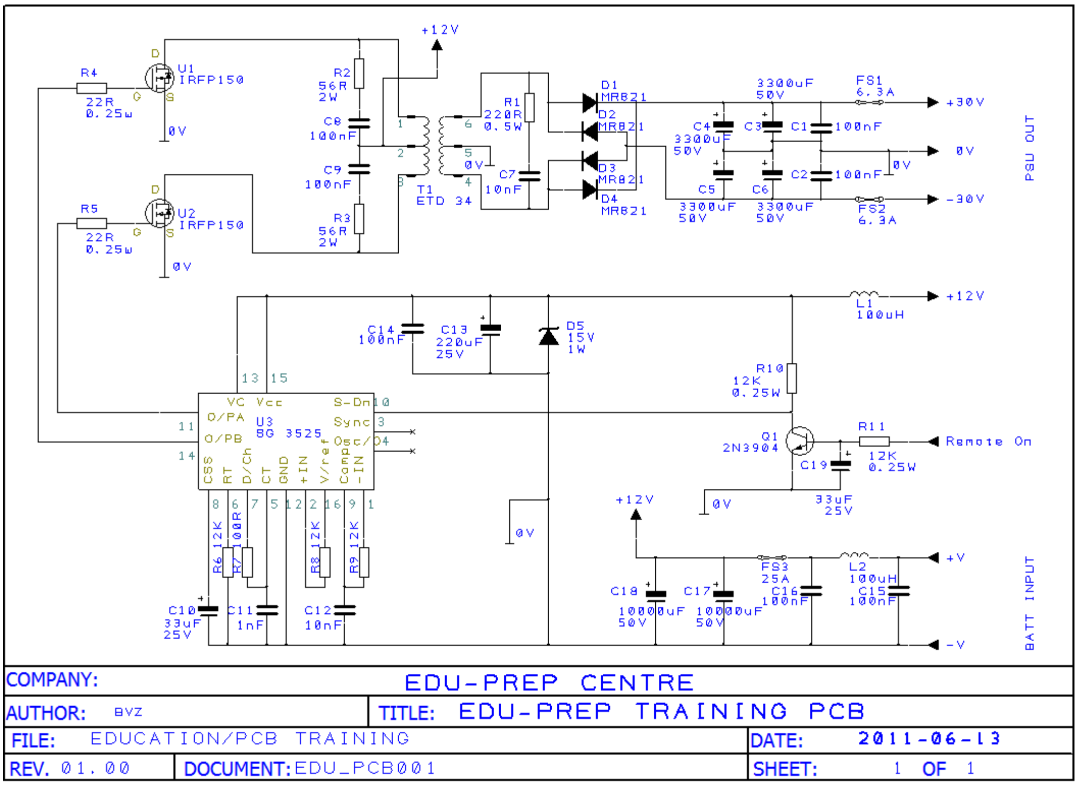


Figure 6-16: Schematic Example

6.4.2 Requirements

Table 6-4 : Exercise 4 Files Required

ITEM NO.	FILE NAME	FILE SIZE
1.	N/A	N/A

6.4.3 Exercise 4 Procedure

Next we are going to draw a schematic diagram using components in the existing libraries.

Nets that represent similar sets of signals can be grouped by assigning the same Net Class to them. A Net Class contains a set of properties that the group of nets referring to it, share.

Note that every net must have a net class, whether it has its own or shares with a group of nets. When designing a Schematic, you assign each net a Net Class name. When the Schematic is translated to the PCB, for each net the matching net class is found in the PCB technology file (matched by name) and, if found, the PCB net class properties are applied to that net.

The Net Class in the PCB design contains track thicknesses and via styles, that will be used once routing is started. We therefore need to generate Net Classes as shown in Figure 6-17.

1. Select “Settings” - “Design Technology” (as shown in Figure 6-17 image 1) to open the “Design Technology” screen.
2. We start by generating these three simple Net Classes by selecting the “Add” button under the “Net Classes” tab (as shown in Figure 6-17 image 2).

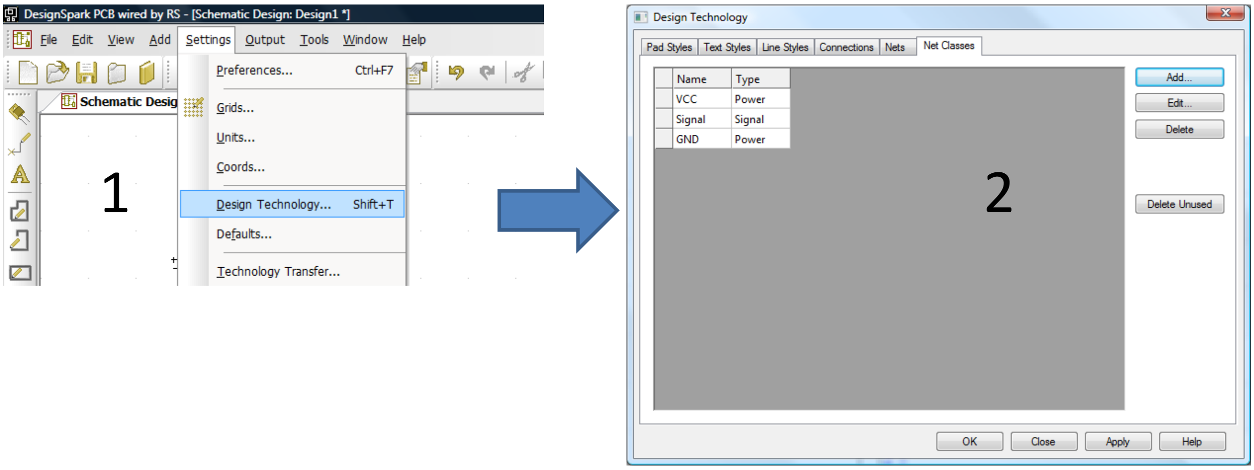


Figure 6-17: Generating Schematics – Design Technology

3. We will draw the Power Amp Circuit shown in Figure 6-18, and use it to explore some of the DesignSpark PCB Features. The components we require can be found in the following component libraries:
   - transistors.cml,
   - discrete.cml,
   - connector.cml and
   - schema.cml.

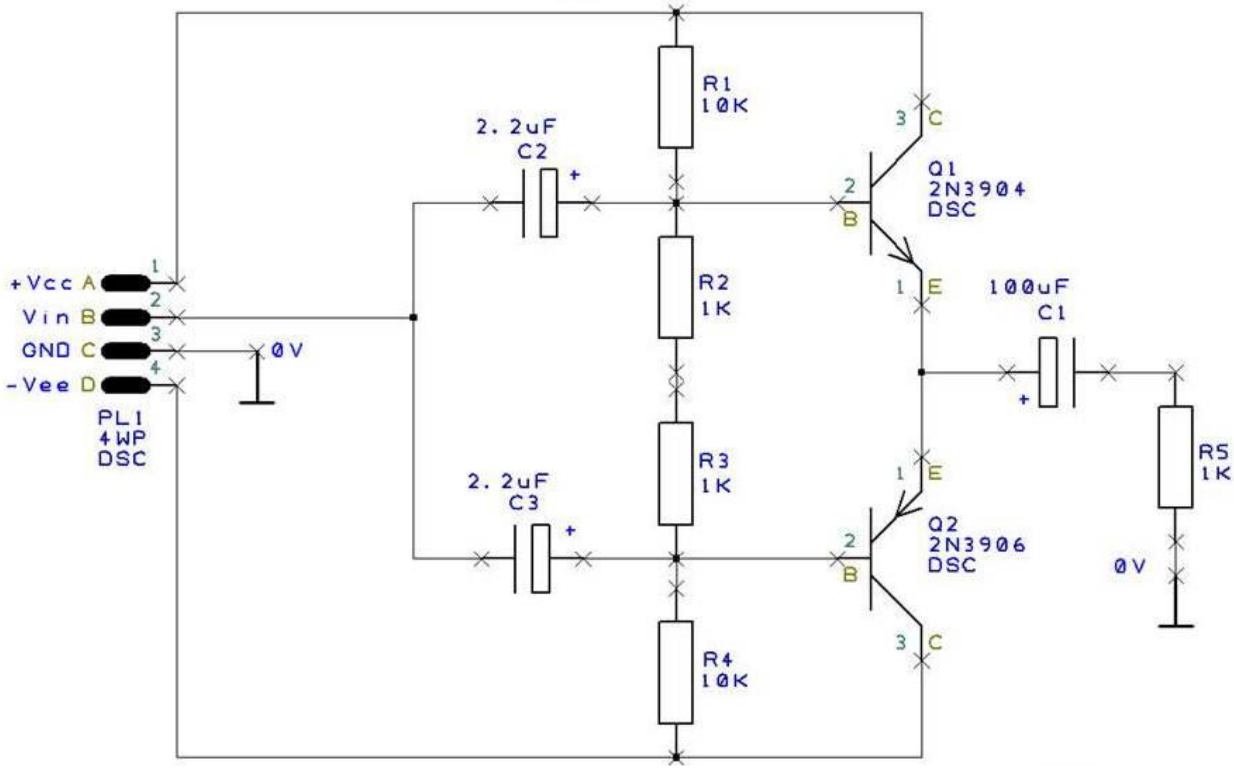


Figure 6-18: Generating Schematics Design

4. Text can be rotated by selecting the relevant annotation (must be highlighted) and pressing “r”.
5. After connecting the components, you will see that the connections are shown with a cross. It is good practice to use the cross to indicate unconnected pins, so we will switch off the “Connected Pins” option as shown in the Figure 6-19 image 1.
6. Select “View” - “Colours” from the top menu bar.
7. It is a good idea to have the “Symbol Terminals” switch on, and the “Connected Pins” switch off in the “Colours” screen (Figure 6-19 image 1). This ensures you only see the cross on unconnected pins (image 2). This makes it easier when adding connections to the schematic, to see which pins are left to be connected.
8. Next we want to ensure that the line styles match those of the components, to ensure that the schematic has a uniform line width. Select “Settings” - “Design Technology” (as shown in Figure 6-17 image 1) to open the “Design Technology” screen.

9. In the “Design Technology” screen, select the “Connections” tab, and select the “Add Style” button (Figure 6-19 image 3).
10. Add a Connections Style with a width of 10 thou and name it as “Normal”.
11. Now we can go back to the connection lines and right click to edit its property and change it to Style “Normal”.
12. While holding in the <Ctrl> button, multiple connections can be highlighted and changed at the same time as shown in image 4.
13. This schematic can now be translated to a PCB (by selecting “Tools” - “Translate to PCB” from the top menu bar) and routed, which is also a good exercise to complete.
14. Remember that the PCB environment will not know the Net Classes, so you can either create them and save the PCB Technology file, or DesignSpark PCB will automatically create them and allocate “Normal” track and vias to the Net Classes.

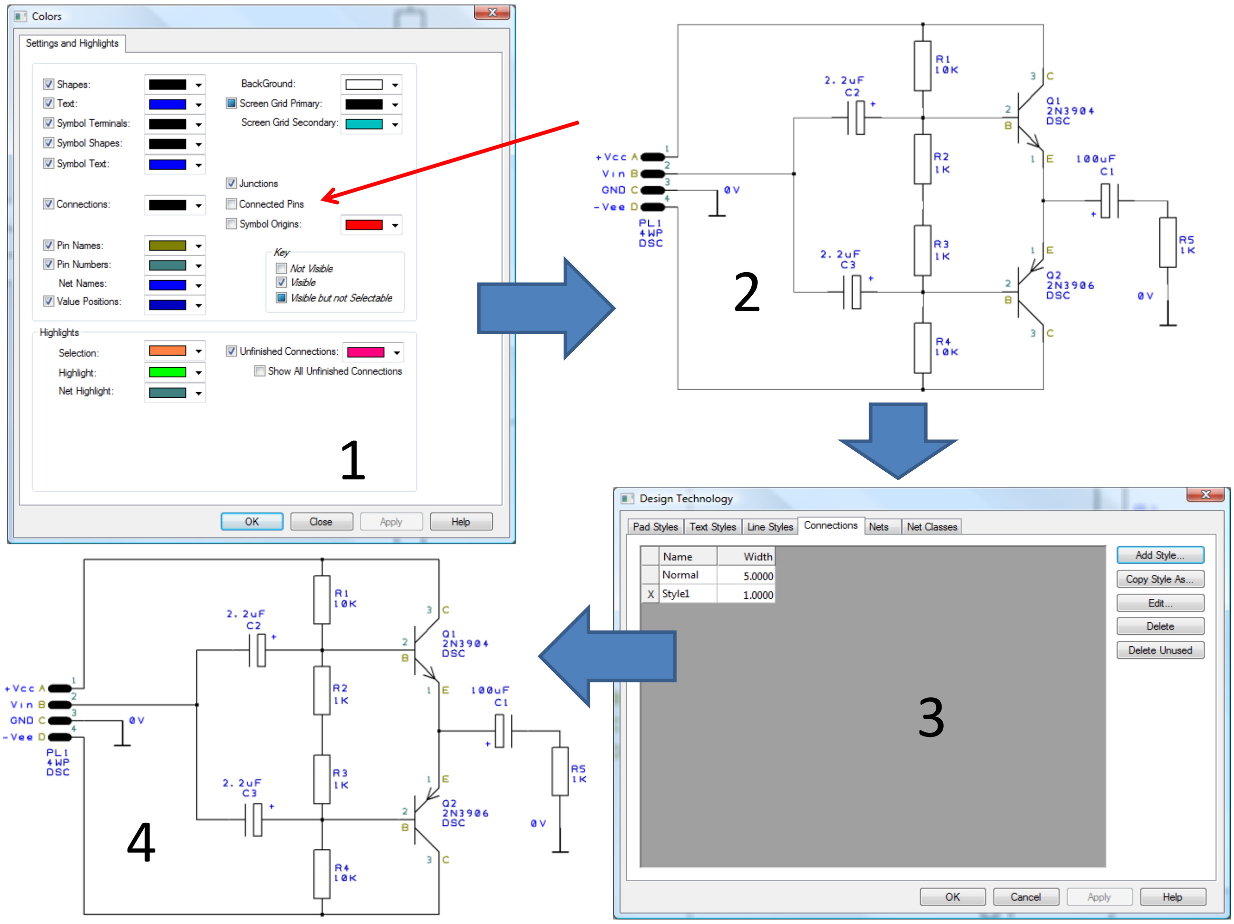


Figure 6-19: Generating Schematics Styles

6.4.4 Exercise 4 Summary / Conclusion

We have seen that Net Classes are important for specifying attributes related to a group of signals (e.g. Power, GND, Signal, etc.).

All colour settings can be changed to user preferences and will be saved in the technology file.

The components in the libraries are easy to access and interconnections can be made using line widths also saved within the technology file.

It is therefore very important to remember to save the technology file for re-use once the user has defined the desired settings.

6.5 Exercise No. 5 – Library Creation

6.5.1 Background and Discussion

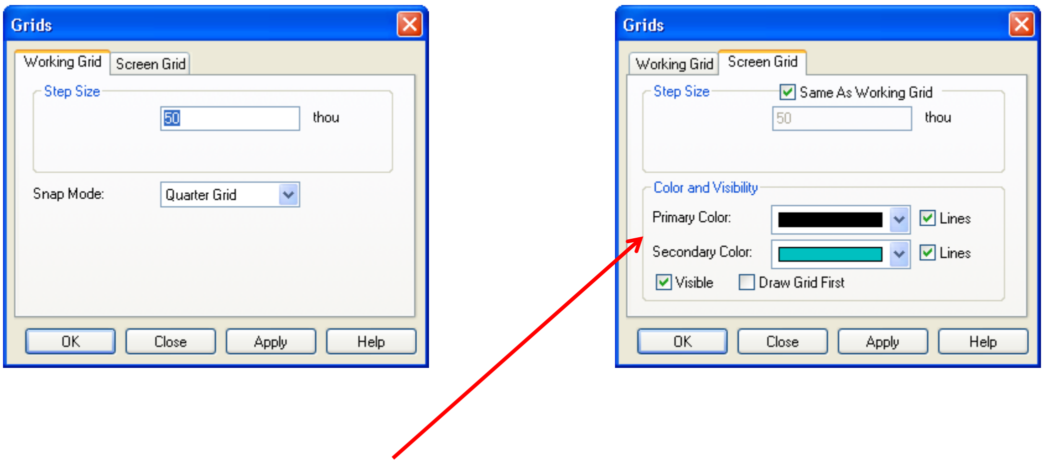
The Library Manager is used to access, amend or add any library symbols or components in the Schematics Symbols, PCB Symbols or Component Libraries.

We are going to create a through-hole diode component by creating both schematic and PCB symbols and then linking them as a component.

The Library manager is accessed from “File” - “Libraries” on the top menu bar or <Ctrl-L>, or from the Toolbar icon (book), and is used to manage all libraries.

We will start by creating our own Schematic symbol and PCB footprint and link them into a component.

If we set up our grids as shown in Figure 6-20, the screen actually appears as graph paper. We do this by selecting “Settings” – “Grids” from the top menu bar, which we finally save in our own Technology File.



Working area looks like Graph Paper

Figure 6-20: Grids Settings

6.5.2 Requirements

Table 6-5 : Exercise 5 Files Required

ITEM NO.	FILE NAME	FILE SIZE
1.	N/A	N/A

6.5.3 Exercise 5 Procedure

1. We start a new Schematic symbol by opening the Library Manager (selecting "File" - "Libraries" on the top menu bar) and selecting the "Schematic Symbols" tab.
2. We want to load our previous Grid settings which we do by selecting the "Tech. Files" button, and selecting our previously saved Technology File.
3. Select the "New Item" button (Figure 6-21) to open the symbol editor.

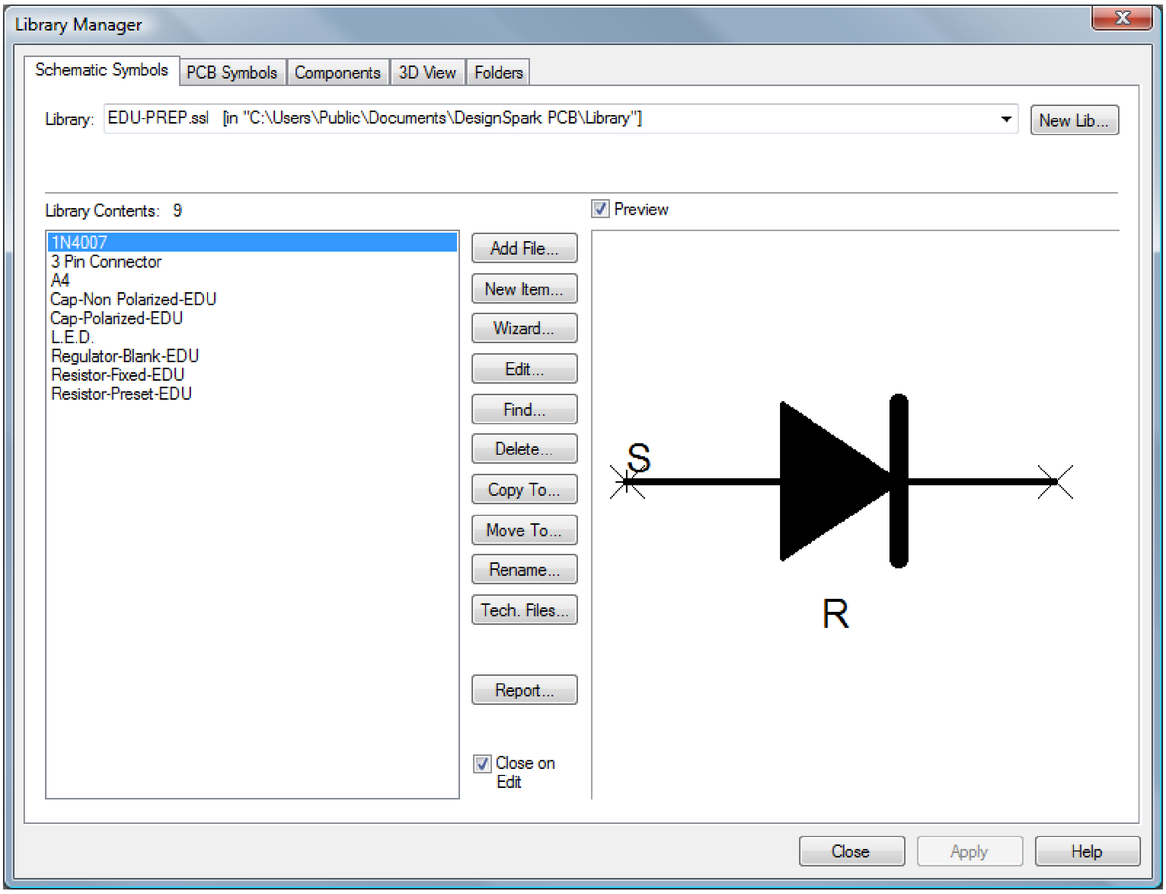


Figure 6-21: Library Creation – Library Manager – New Item

4. For this exercise we will draw a diode symbol. We use the "schematic symbol" toolbar on the left to draw.
5. While drawing, you can right-click the mouse and change both the line style to something defined in the Technology File, or the Segment Mode e.g. Free-hand, Orthogonal, Right-Angle, Miter (45degree) or Fillet.
6. While drawing the horizontal line, right-click and select "Change Style".

7. Type a style name “Line 10” and change the “Width” field to 10 and select the “OK” button.
8. Select “Yes” to the query “Add new line style to the design?”.
9. Right-click at the end of the shape to select “Finish Here”.
10. When drawing the vertical line (diode cathode), right-click and select “Change Style”.
11. Type a style name “Line 25” and type 25 in the “Width” field, and select the “OK” button.
12. Select “Yes” to the query “Add new line style to the design?”.
13. To draw the solid triangle shape, choose “Add” – “Shape” – “Triangle” from the top menu bar, and while drawing, right-click and choose “Filled Shape”.
14. DesignSpark PCB automatically places the Symbol Origin S. You can place the References by selecting “Add” – “Reference Origin” from the top menu bar, and place the required references, such as:
    - Reference Name,
    - Component Name,
    - Package Name,
    - Symbol Name,
    - Description and
    - Value.

    It can all be included in the one Reference or added separately as individual references.
15. To add the pads/pins to the component, select “Add” - Pad” from the top menu bar or click on the “Add Pad” icon on the left (or press <F4>).
16. Place a pad on the left (called “N1”) and one on the right (called “N2”).
17. In the event that we would want to add Terminal Names, we would have to move the “N1” & “N2” to locations suitable for adding text away from the symbol.
18. Save the symbol in the required schematic symbol library selecting “File” – “Save” from the top menu bar.
19. Next we will generate the PCB footprint symbol.

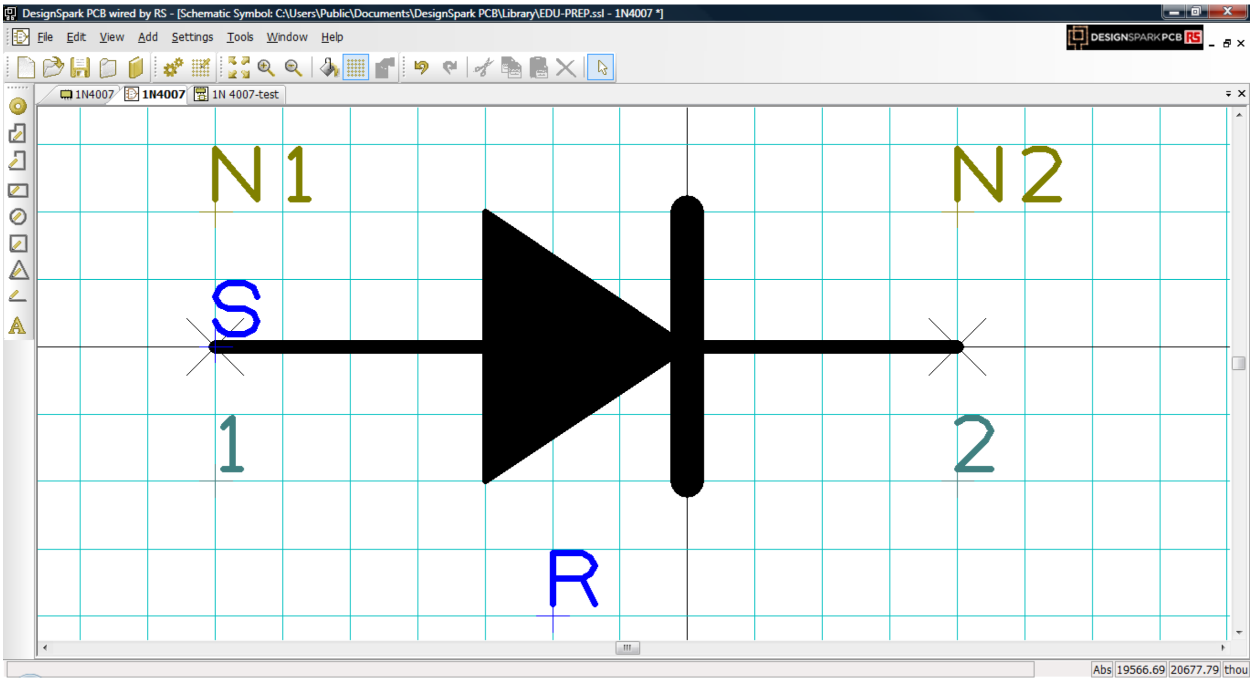


Figure 6-22: Library Creation – Symbol

20. Because most through-hole components have a standard spacing of 2.54mm, we set our Grid to this dimension as shown in Figure 6-23 (image 1), by selecting “Settings” – “Grids” from the top menu bar.
21. As shown in image 2, standard vero-board and proto-board dimensions are 2.54mm (100thou) to accommodate these components.

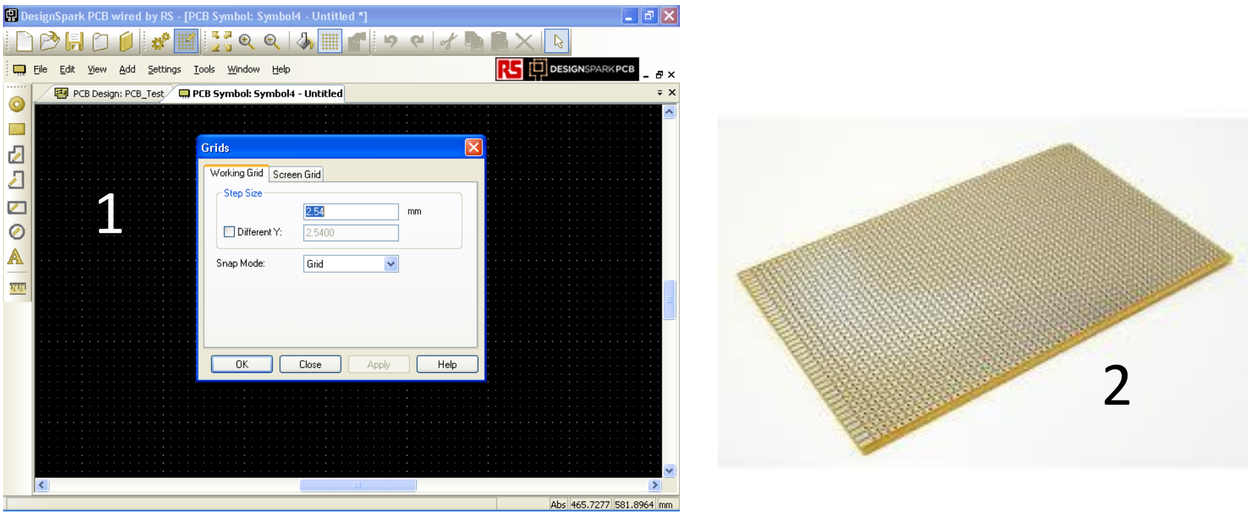


Figure 6-23: Library Creation – Grid setting

22. Similar to the schematic symbol, the PCB footprint symbol is created through the Library Manager. The Library manager is accessed by selecting “File” –

"Libraries" from the top menu bar, or pressing <Ctrl-L>, or from the Toolbar icon ("book"), and is used to manage all libraries.

23. Select the "PCB Symbols" tab.
24. Select the "Tech. Files" button to load previously saved Technology Files with Units and Grid preferences, before selecting the "New Item" button as shown in Figure 6-24.

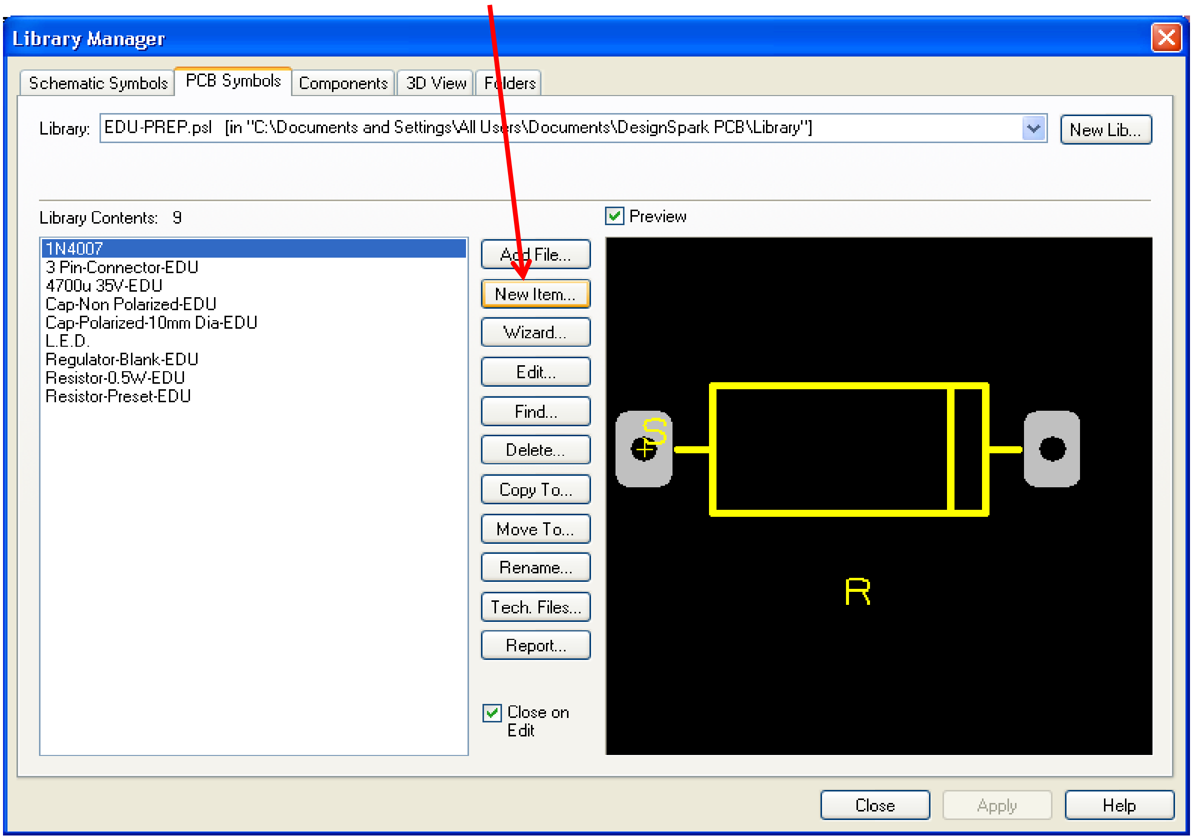


Figure 6-24: Library Creation – New item

25. We start by adding a though-hole pad from the "Add" – "Pad" selection from the top menu bar (or by pressing <F4>), as shown in Figure 6-25 (image 1).
26. After placing the pad, right click and select "Properties".
27. Change the pad to have the dimensions as shown on image 2.
28. For hand-assembly it is always better to have a Rounded Rectangular pad shape to ensure sufficient pad surface to allow for solder flow. This pad can be saved as a new style by adding a name into the "Style" field (indicated with the red arrow on image 3).

29. You will be prompted to Add the new style in the design, which means that it will be saved as a Pad Style in the Technology File for future use. Save this style as “Style 1”.
30. Selecting these pad dimensions results in the shape with a 2.1mm x 3mm rectangular pad as shown in image 4.
31. An important design rule to always remember is the relation between the track and the pad, as shown in Figure 6-26.

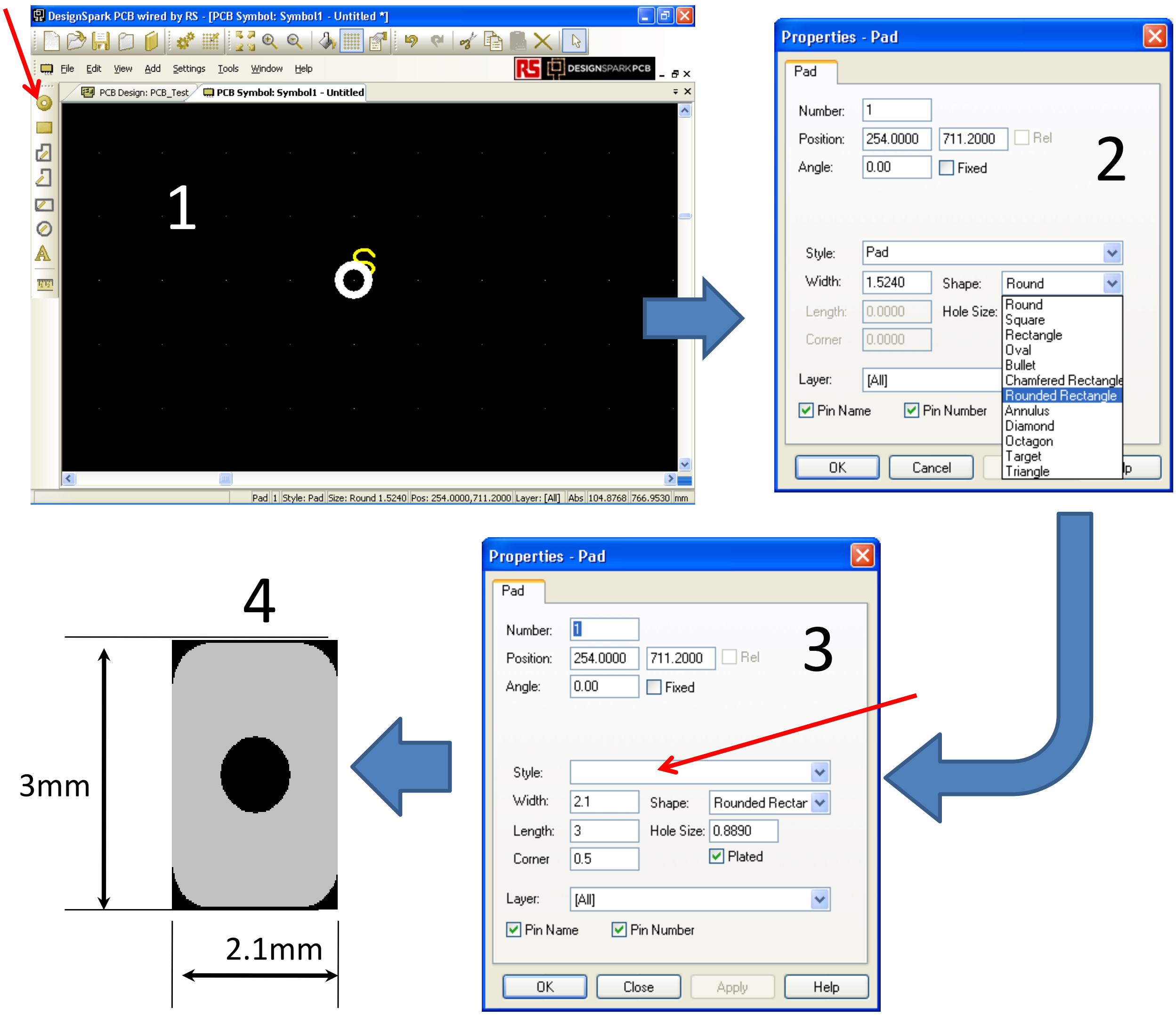


Figure 6-25: Library Creation – Add Pad

Track / Trace Width Guidelines:
Signal 10 thou (0.254mm)
Power 40 thou (1mm)
Ground 40 thou (1mm)

Track / Pad / Through Hole Relationship must be $\frac{T}{2} < \frac{P-D}{2}$

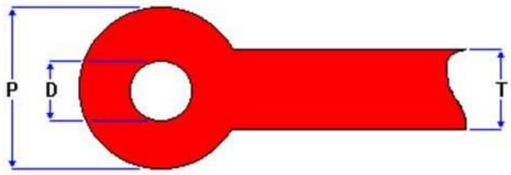


Figure 6-26: Library Creation – Track/Pad/Through Hole relationship

32. Our 1N4007 diode device has a pin spacing of 500thou or 12.7mm. We therefore place the pads 12.7mm apart, which is 5 grid positions (2.54 x 5) as shown in Figure 6-27 (image 1).
33. We then select our rounded rectangular pads, which we can either individually specify or select from your previously saved style (“Style 1”) as shown on image 2.
34. We add any References by selecting “Add” – “Reference Origin” from the top menu bar.
35. Our final layout shows the Symbol Origin (S), pin numbers (1 & 2), the Pin Names (N1 & N2) and the Reference Origin (R) as shown in image 3.

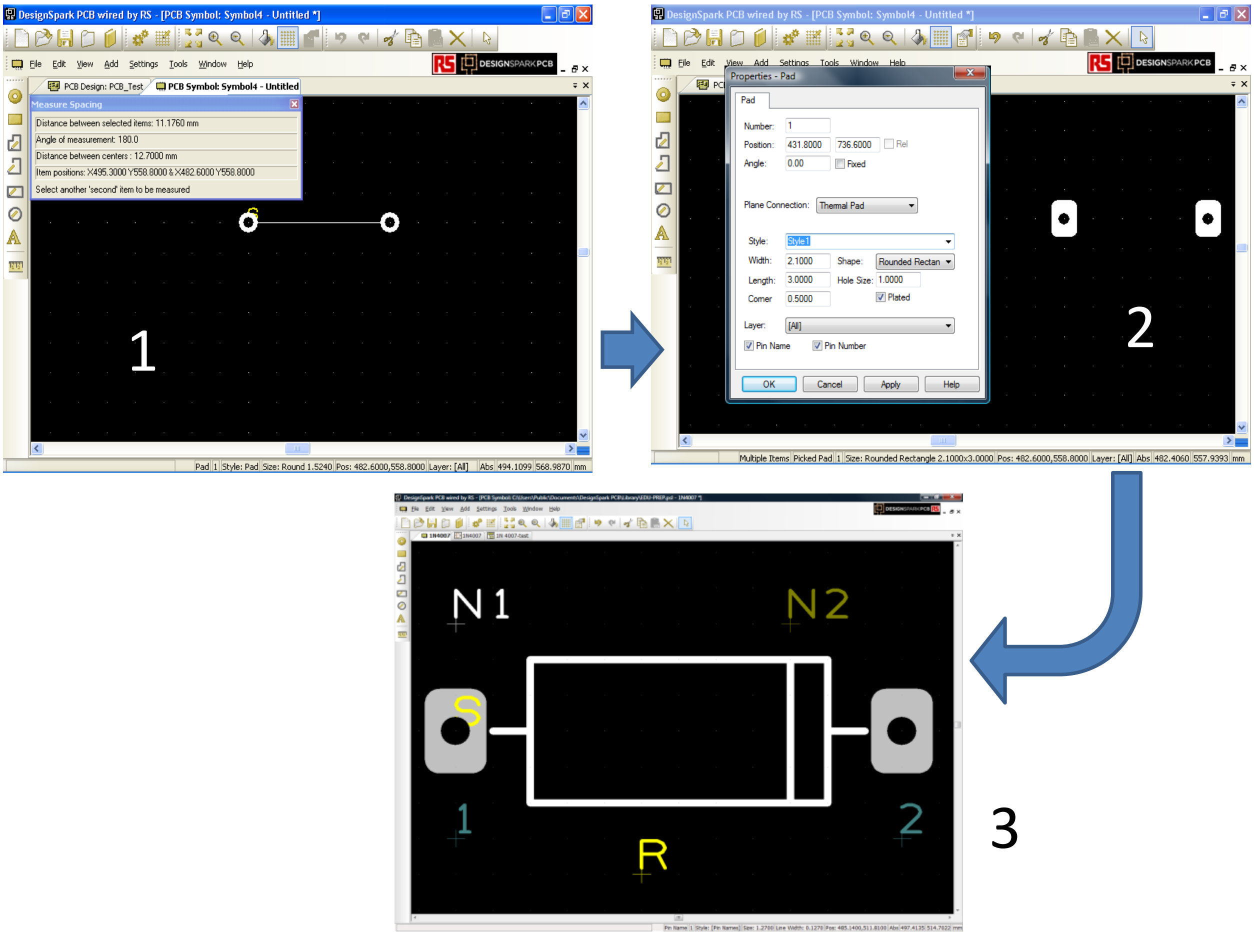


Figure 6-27: PCB Footprint Library Creation – Diode

36. Before the diode body outline can be added, the working grid must first be changed to a finer 1.27mm (by selecting "Settings" – "Grids" from the top menu bar, and change to 1.27mm).

37. The outline must be added on the Top silk layer as shown in Figure 6-28. For adding the body outline, while drawing, right-click and select "Change Layer" (or press <L>) and select "Top Silk".

38. The PCB footprint symbol can now be saved to the appropriate library by selecting "File" – "Save" from the top menu bar.

39. We now have a schematic symbol and a PCB footprint symbol which we can link to form a component.

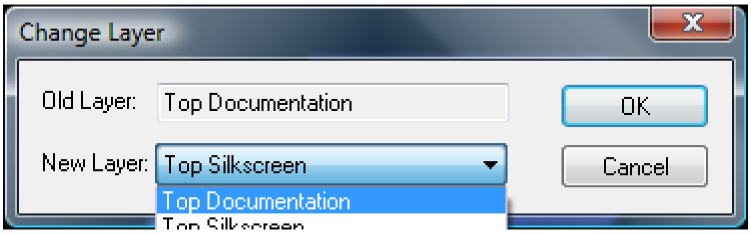


Figure 6-28: Library Creation – Change Layer

40. To create our new component we select “File” – “Libraries” from the top menu bar, or select the “Library Manager” icon, or press <Ctrl-L>.
41. On the “Components” tab, we select the “New Item” button as shown in Figure 6-29 (image 1).
42. The “New Component” screen (image 2) allows us to link a schematic symbol to a PCB footprint symbol.
43. Select the two symbols (previously created), using the “Find Symbol” buttons for both the Schematic and the PCB fields.
44. Enter an appropriate “Component” name, choose a “Package” and “Default Reference” and save the new component in a suitable library by selecting the “OK” button.
45. Link the Schematic, PCB and Component pin numbers by completing the table in the Top left hand corner of image 3.
46. Save by selecting “File” – “Save” from the top menu bar.
47. In the table displayed in image 3, terminal names can be added to the component under the “Sch Symbol” “Terminal Name”, which will now be visible only if it was selected in the properties when the symbol was created (as per steps 16, 17 & 35 above).
48. The component as shown in image 3 is now available in both Schematic and PCB environments as described in exercises 3 and 4.

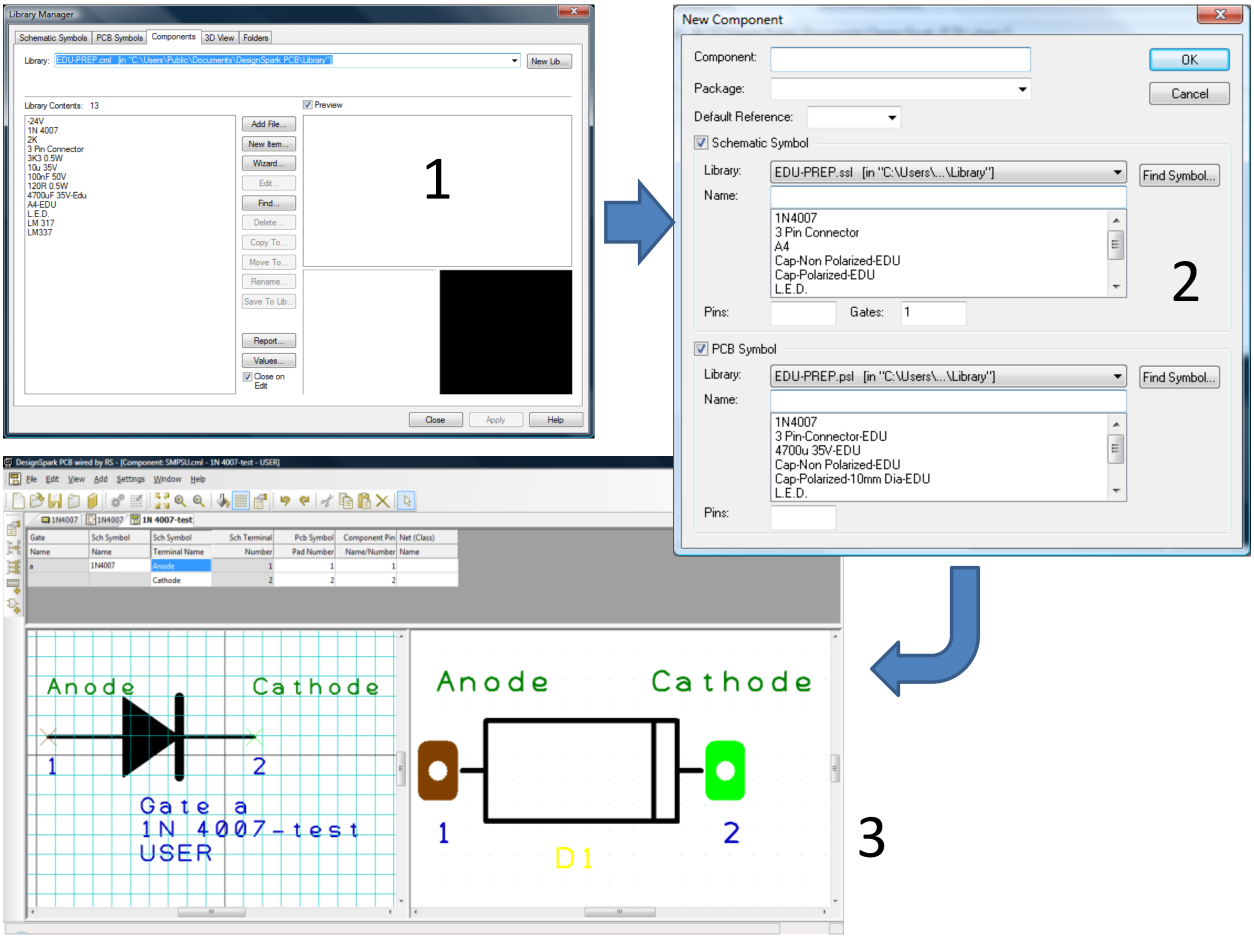


Figure 6-29: Library Creation – New Component

6.5.4 Exercise 5 Summary / Conclusion

All Schematic, PCB and Component libraries are accessed and amended through the Library Manager.

Before a Schematic or PCB devices can be used, it must be a component.

A component is created by adding at least a Schematic symbol, but could also include a PCB Footprint, although not necessary.

To be able to translate a Schematic design to a PCB design, the component must have both Schematic and PCB symbols linked.

6.6 Exercise No. 6 – Symbol Wizard

6.6.1 Background and Discussion

There is a Symbol Wizard capability in both the Schematic and PCB within the Library Manager to assist in generating components.

Pre-defined symbols can be selected and by providing a few parameters, components can be created similar to the manual process as described in exercise 5.

The PCB footprint symbols contain standard packages and the wizard requires a few simple parameters as available on the component datasheet to create the footprint.

Keeping the schematic, PCB and component as separate entities allows the user to link multiple schematic symbols to any footprint, and saving this combination as different components.

Figure 6-30 shows an example of the Schematic Symbol Wizard and Figure 6-31 shows the PCB Footprint Wizard.

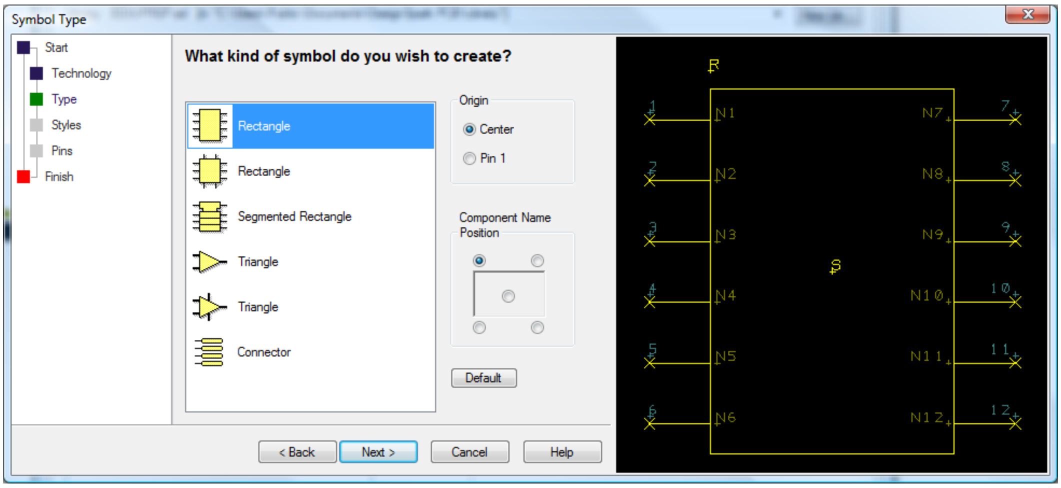


Figure 6-30: Library Creation – Schematic Symbol Wizard

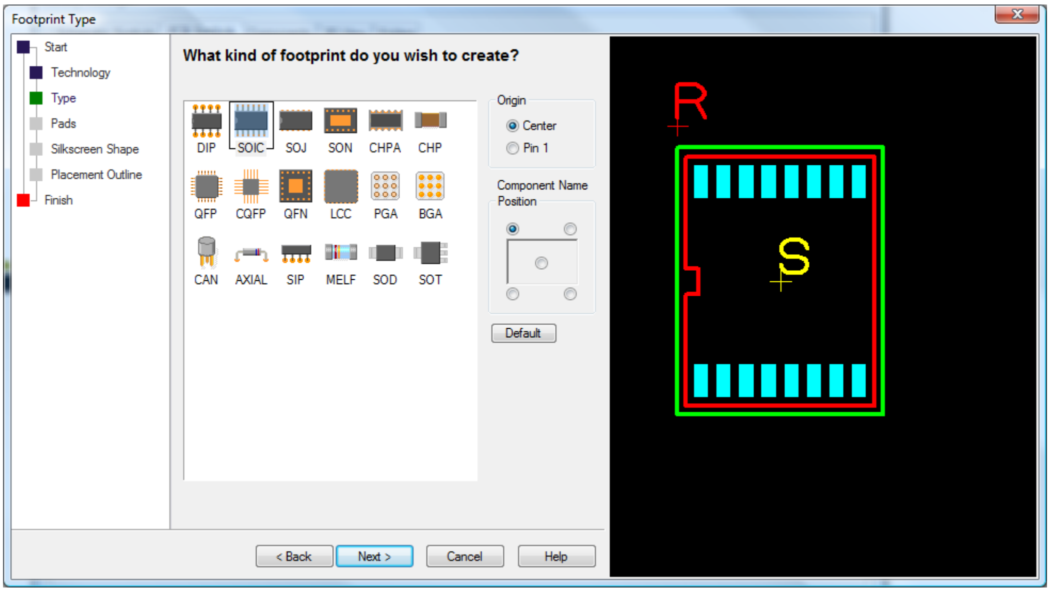


Figure 6-31: Library Creation – PCB Footprint Wizard

6.6.2 Requirements

Table 6-6 : Exercise 6 Files Required

ITEM NO.	FILE NAME	FILE SIZE
1.	N/A	N/A

6.6.3 Exercise 6 Procedure

First we will generate the schematic symbol for a 555 timer 8 pin using the schematic wizard.

1. Select "File" – "Libraries" from the top menu bar, or select the "Library Manager" icon or press <Ctrl-L>.
2. Select the "Schematic Symbols" tab and select the "Wizard" button.
3. Select the "Next" button.
4. Choose the "Units" in thou and select the "Next" button.
5. Select the Rectangular shape, with "Origin" in Centre, and the "Component Name Position" in the top left, and select the "Next" button.
6. Select the "Next" button again (thereby keeping the defaults).
7. Now input 4 pins on the left and 4 pins on the right.

8. Select in the "Pin Numbering" field, the "Top Left to Bottom Right" option, and select the "Next" button.
9. Click on the "Save the symbol to the library" tick-box and choose an appropriate library.
10. Select the "Finish" button when done, which will produce the schematic as shown in Figure 6-32.
11. The symbol can be edited as required, but just remember to save before exit.

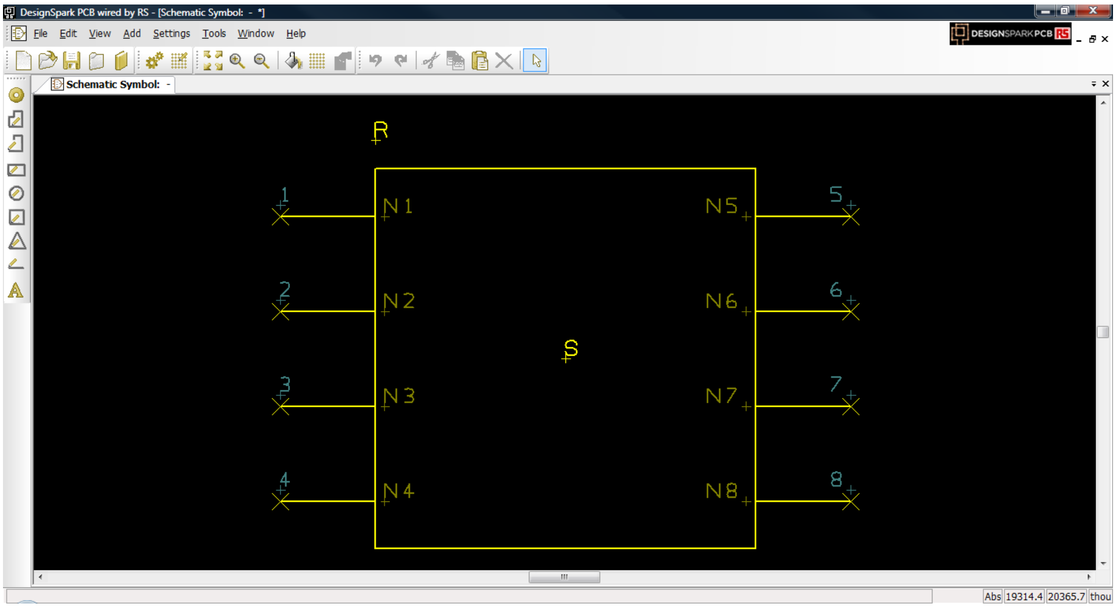


Figure 6-32: Wizard – Schematic Symbol

Now we will generate the 8 pin DIP PCB footprint for the 555 timer using the PCB wizard.

12. Select "File" – "Libraries" from the top menu bar, or select the "Library Manager" icon or press <Ctrl-L>.
13. Select the "PCB Symbols" tab and select the "Wizard" button.
14. Select the "Next" button.
15. Choose the "Units" in mm, "Precision" to 4, and select the "Next" button.
16. Select the "DIP" layout, with "Origin" in Centre, and the "Component Name Position" in the top left, and select the "Next" button.
17. Change "Pad Counts" to 8 for the 8 pins.
18. Select the "Pad Style" as "Rounded Rectangle" and input in the "Measurements" field the following dimensions as per the datasheet:

- pin spacing (e = 2.54mm),
- pin top to pin bottom (E = 7.62mm),
- pad width (PW=1.524mm) and
- Hole Dimension (HD=0.7620mm).

19. Select the "Next" button again (thereby keeping the defaults).
20. Select the "No" option for placement shape and select the "Next" button.
21. Click on the "Save the footprint to the library" tick-box and choose an appropriate library.
22. Select the "Finish" button when done, to create the PCB footprint as shown in Figure 6-33.
23. The footprint can be edited as required, but just remember to save before exit.
24. The schematic symbol and PCB footprint can now be linked into a component as done in exercise 5.

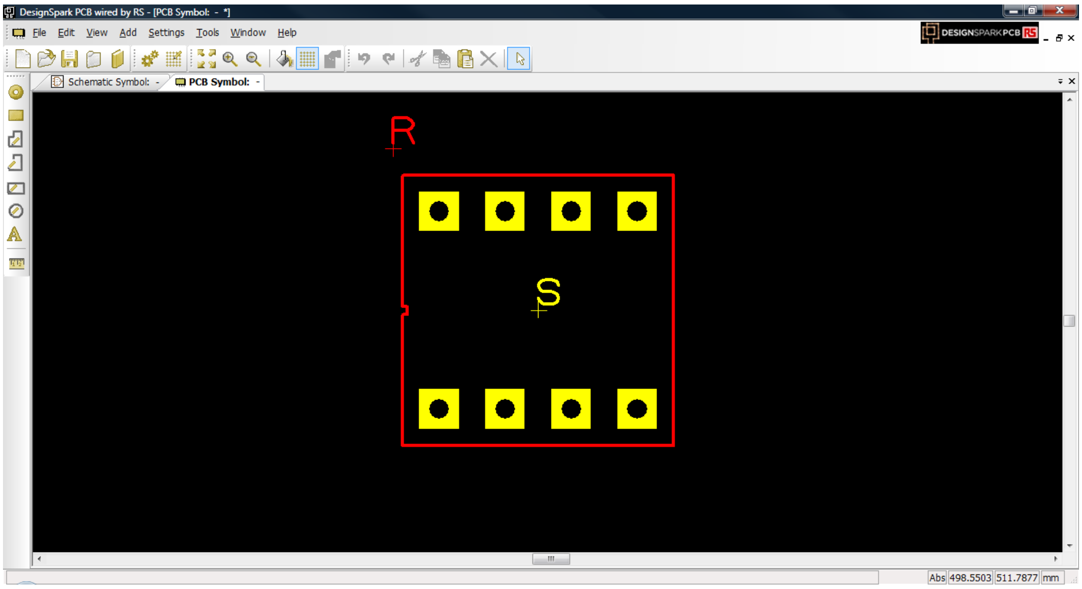


Figure 6-33: Wizard – PCB Footprint

6.6.4 Exercise 6 Summary / Conclusion

Using the Schematic and PCB Symbol Wizards, devices may be created that can subsequently be linked within a component.