PRINTED CIRCUIT BOARD DESIGN: HANDBOOK

PRINTED CIRCUIT BOARD DESIGN: HANDBOOK
Using Eagle Cadsoft

Puneet Jain
IIIT, Delhi (India)

Vandana Mittal
IIIT, Delhi (India)

Manoj Gulati
IIIT, Delhi (India)

CONTENTS

INTRODUCTION

This tutorial provides a basic introduction to the EAGLE PCB-Design CAD tool Version 5.11.0. along with support for earlier versions too.

It covers the use of the EAGLE Schematic Editor, Layout Editor, and Auto router. This guide will lead you through the program in the natural order, starting with the Schematic Editor module and working through to board design and auto routing. EAGLE is a powerful graphics editor for designing PC-board layouts and schematics. It is compatible with both Windows and Linux OS.

CHAPTER 1

FEATURES OF EAGLE

1.1 Professional Version

1.1.1 General

1. Maximum drawing area 64 x 64 inches (about 1600 x 1600 mm)
2. Resolution 1/10.000 mm (0.1 microns)
3. mm or inch grid
4. Up to 255 layers, user definable colours
5. Command files (Script files)
6. C-like User Language for data import and export
7. Simple library editing
8. Composition of user-defined libraries with already existing elements by Drag & Drop
9. Easy generation of new package variants from any library by Drag & Drop
10. Free rotation of package variants (0.1 degree steps)
11. Library browser with powerful search function
12. Support of technology feature (e.g. 74L00, 74LS00..)
13. Generation of graphics output as well as manufacturing and testing output with the CAM processor or the help the User Language
14. Printouts via the OS's printer drivers
15. User-definable, free programmable User Language to generate data for mounting machines, test equipment, milling machines or any other data format
16. Part-list generation with database support (bom.ulp)

17. Drag & Drop in the Control Panel
18. Automatic backup function

1.1.2 Layout Editor

1. Full SMD support
2. Support of blind and buried vias
3. Rotation of elements in arbitrary angles (0.1-degree steps)
4. Texts can be placed in any orientation
5. Dynamic calculation of signal lines while routing the layout
6. Tracks can be layed out with rounded corners in any radius
7. Miltering to smooth wire joints
8. Design Rule Check for board layouts (checks e.g. overlaps, measures of pads or tracks)
9. Copper pouring (ground plains)
10. Package variants support

1.1.3 Schematic Module

1. Up to 99 sheets per schematic
2. Simple copying of parts
3. Online-Forward & Back Annotation between schematic and board
4. Automatic board generation
5. Automatic generation of supply signals
6. Electrical Rule Check (error check in the schematic and consistency check between schematic and layout)

1.1.4 Auto-Router Module

1. Fully integrated into basic program
2. Uses the layout's Design Rules
3. Change between manual and automatic routing at any time
4. Ripup & retry algorithm
5. User-definable strategy by cost factors
6. Routing grid down to 0.02 mm (about 0.8 mil)
7. No placement restrictions
8. Up to 16 signal layers (with user definable preferred directions)
9. Up to 14 supply layers
10. Full support of blind and buried vias
11. Takes into consideration various signal classes

1.2 Standard Edition

Restrictions to Standard Edition in Layout Editor:
1. The layout area is restricted to a maximum of 160 x 100 mm (about 6.3 x 3.9 inches).
Outside this area it is not possible to place packages and draw signals.
2. A maximum number of 4 signal layers are allowed (top, bottom, and 2 inner layers).

1.3 Light Edition(Freeware)

Restrictions to the EAGLE Light Version, which is available as Freeware (for testing and evaluation):
1. The board area is restricted to 100 x 80 mm (about 3.9 x 3.2 inches).
2. Outside this area it is not possible to place packages and draw signals.
3. Only two signal layers can be used (no inner layers).
4. A schematic can consist of only one single sheet.
5. Larger layouts and schematics can be printed with the smaller editions.
6. The CAM processor can generate manufacturing data as well.

CHAPTER 2

CONTROL PANEL

After starting EAGLE, the Control Panel will be opened. It allows you to load and save projects as well as to setup certain program parameters. Right mouse click to an entry in the Projects branch of the tree view opens a context menu that allows to start a new project.
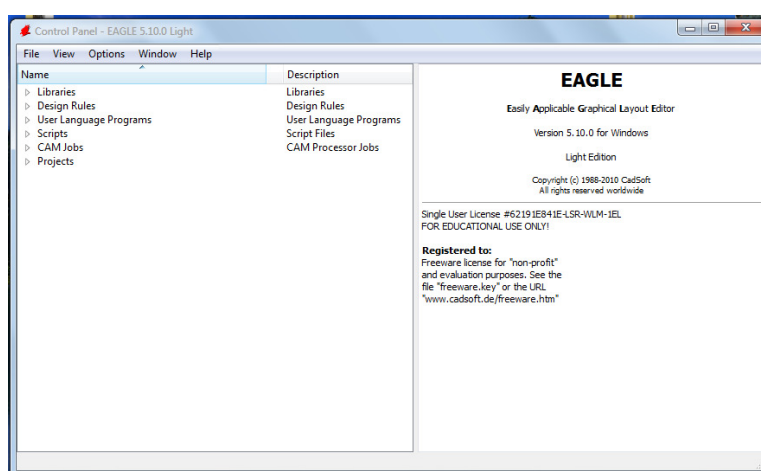

Figure 2.1   Eagle Control Panel

The tree view allows a quick survey of EAGLE's libraries. Double-click an entry in the Libraries branch. Now the contents of the library are displayed.

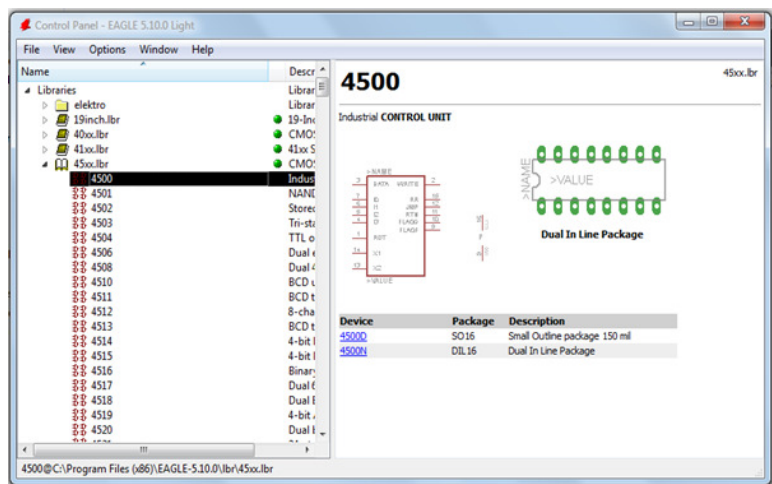Selecting an object shows a short descriptive text on the right.

Figure 2.2 Eagle Libraries

2.1 Eagle Files

The following table lists the most important file types that can be edited with EAGLE:

Type	Window	Name
Board	Layout Editor	*.brd
Schematic	Schematic Editor	*.sch
Library	Library Editor	*.lbr
Script File	Text Editor	*.scr
User Language Program	Text Editor	*.ulp
Any text file	Text Editor	*.*

Figure 2.3 Eagle Files and their Extensions

2.2 Backup Files

EAGLE creates backup data of schematic, board, and library files. They will be saved with modified file extensions: .brd becomes .b#1, .sch becomes .s#1 and .lbr becomes .l#1.There can be a maximum number of 9 backup files. It is also possible to have EAGLE files saved in a certain time-interval. In this case the files get the extension b##, s## or l##. The files can be used again after renaming them with the original file extension.

2.3 Create EAGLE Projects

To create a new project first. After starting the program, first double-click the + character of the Projects path, then double-click the + character of the entries examples and tutorial in the tree view. The contents of the tutorial directory appear. Double-click tutorial with the right mouse button. Select the option New Project in the popup menu. Name the new project My Project.

CHAPTER 3

SELECTING LAYERS FOR DISPLAY

EAGLE-Drawings contain objects in different drawing layers. In order to obtain a useful result several layers are combined for the output. For example, the combination of Top, Pad, and Vias layers is used to generate a film for etching the component side of the printed-circuit board. Consequently the combination of Bottom, Pad, and Vias layers is used to generate the film for the solder side of the board. The Pad layer contains the through-holes for the component connections and the Vias layer contains the via-holes which are needed when a signal track changes to another layer. Load the board demo2.brd using the menu File/Open/Board than try for path $eagle5.11.0\backslash projects\backslash examples\backslash tutorial$ and click in the command toolbar on the icon for the DISPLAY command (look at the toolbar layout on the previous pages). The marked layers are currently displayed. By clicking on the layer number the display of each layer can be switched on or off. The All and None buttons switch on or off all layers.

3.1   Layer Colours

The layer colours are freely definable. In the Options/Set, Colour tab, you can define colour values. You always have to define a pair of colours: The normal colour of the layer and the highlight colour, which is used to emphasize an object while using the SHOW or MOVE, command. Use the DISPLAY menu, Change button, Colour item to assign colours to layers.
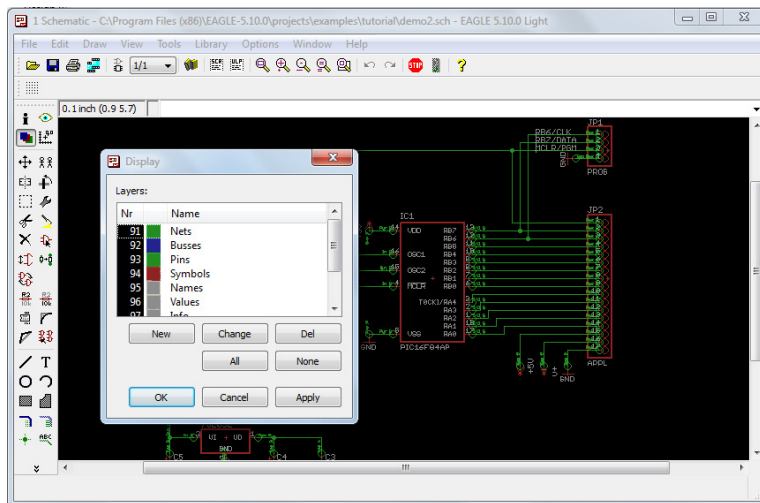

Figure 3.1   Layer Window

CHAPTER 4

SETTING UP GRID AND UNITS

Schematics should always be drawn on a grid of 0.1 inches (2.54 mm) since the libraries are defined in this way. The grid for boards is determined by the components used and by the complexity of the board. Grid and unit are setup with the GRID command by clicking on the GRID icon in the parameter toolbar. All values are given in the currently selected unit.

Figure 4.1   GRID Icon

For all settings in the Design Rules window (Double click Edit/Design Rules...) one can use values in mil or in millimetres (1 mil = 1/1000 inch). The default unit is mil. If you prefer to work with millimetres simply add the unit to the value, for example: 0.2mm.

NOTE:-Suppose if component is not allowed to be placed at a location where you want to place it i.e it slides forward or backward than you must reduce grid to half of previous e.g from .01 inch to .005 inch than the component can be placed with better precision.

<u>Inch - Mil - Millimeter Table for the Most Usual Values:</u>

inch	mil	mm
0,008	8	0,2032
0,010	10	0,2540
0,012	12	0,3048
0,016	16	0,4064
0,024	24	0,6096
0,032	32	0,8128
0,040	40	1,0160
0,050	50	1,2700
0,100	100	2,5400

Figure 4.2 Table for Reference

CHAPTER 5

DRAWING A SCHEMATIC

In this section you will learn how to add components to any circuit, connect nets and buses used in a circuit design. You will then be able to create a schematic (Schematic is actually a free hand circuit diagram built using prebuilt or self-build components in libraries provided by any schematic capture tool). To create an empty schematic, open $New \rightarrow Schematic$, and enlarge the editor window.

Grid   Note:-The standard grid for schematics is 0.1 inches. Symbols should be placed on this grid or a multiple of it, since otherwise it can happen that nets cannot be connected to the pins.

Set the alternative grid to 0.25 inch. This would allow to place components in a better manner. Now as the pointer is moved on graphical area it will show the current coordinates of mouse pointer in terms of (x, y). Using this we can calculate exact position of any component on this graphical area.
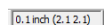
0.1 inch (2.1 2.1)

Figure 5.1   Coordinates are shown like this

5.1   Using Libraries

Libraries are entities clubbing together similar type of electronics components.

5.1.1   Using Add Command

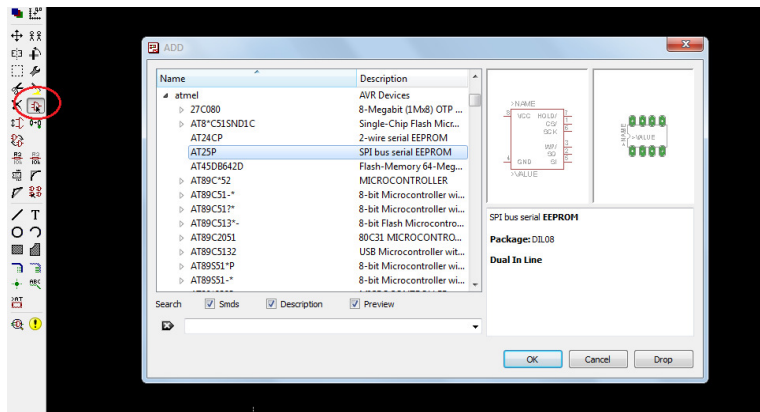This command used to add the component to the schematic from the suitable library.

Figure 5.2   Add Button and Component Window

After selecting the component click on OK adds that component to mouse pointer. A number of similar components can be added using left click of the mouse over various graphical locations.

5.1.2   The USE Command

This command is used to add/use a new library to the pre-existing set of libraries using a drop down menu on top.

Figure 5.3   Use Button

5.1.3   The INVOKE Command

The INVOKE command can be used to allow the connection of active components to a power source other than VCC and GND. To demonstrate its use: Click on INVOKE and left click on the any IC package. A popup menu appears displaying number of gates used in that IC package.

5.2   Adding a frame to the Schematic

Frame is generally preferred to be added to schematic in order to give a systematic view to circuit layout. But there is no such restriction that a frame must always be present in a casual schematic.
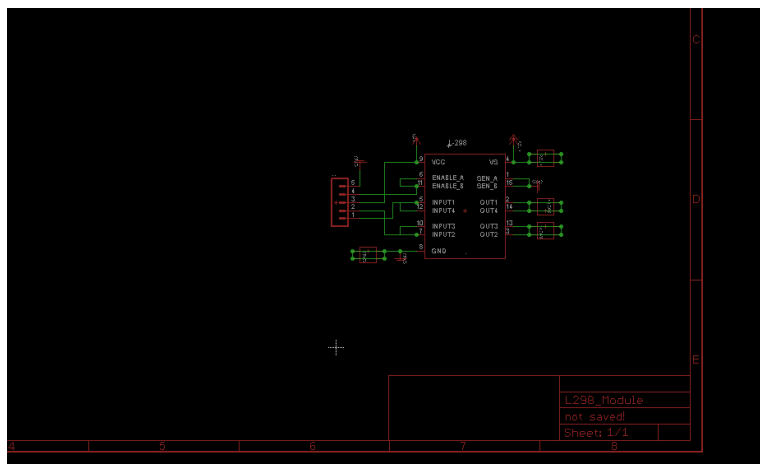

Figure 5.4   Frame added to Schematic

5.3   The WIRE Command

The wire command has dual existence on the schematic part it is used to connect component with other components and is defined under Display as NET layer and it can be modified to various bending angle to suit the interconnection by suitable right click. Note:
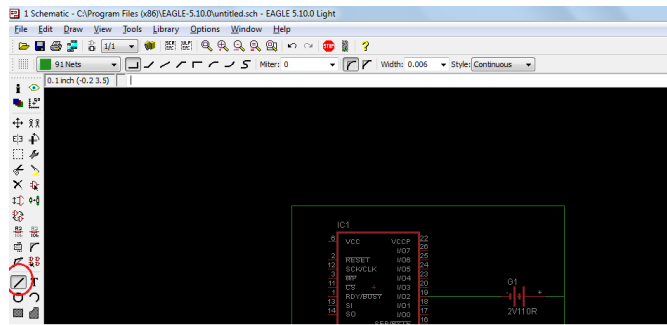

Figure 5.5   Wire Command

-Another purpose of wire command is on board part it is used to draw copper tracks over any location on Board.

5.4   The TEXT Command

This command is used to add certain text to different locations on schematic as well as Board layout. This text can be added at various layers of PCB but it is preferred to be present at top dimension layer, top name (t-name) layer. Note: - Text command is useful
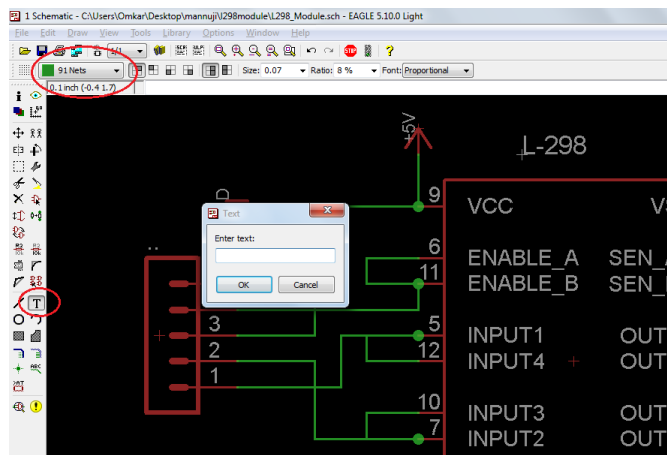

Figure 5.6   Text Command(will write on the 91Nets Layer)

in specifying any name, dimensions, value of any electrical component with variable Font and sizes but preferred font is Vector.

5.5 Further in Schematic Editor

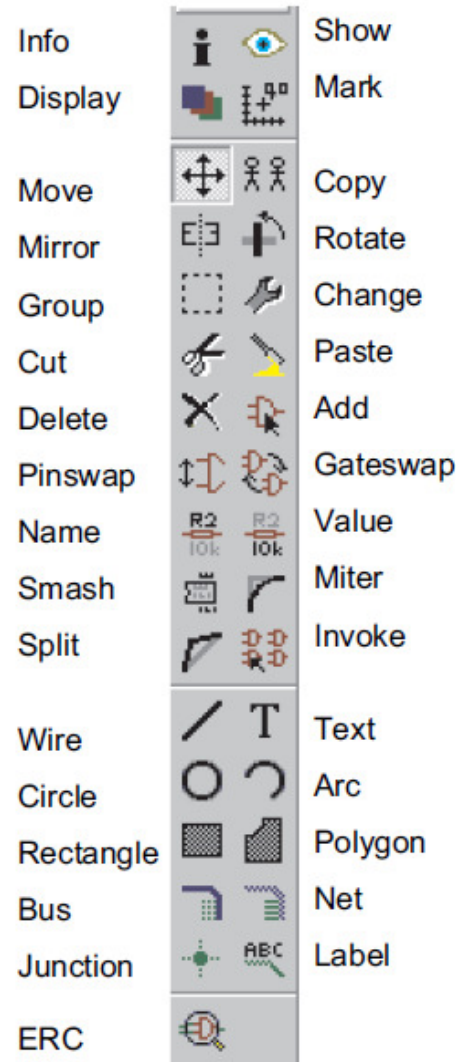The tool box on left hand side has following tools:-

Info		Show
Display		Mark
Move		Copy
Mirror		Rotate
Group		Change
Cut		Paste
Delete		Add
Pinswap		Gateswap
Name		Value
Smash		Miter
Split		Invoke
Wire		Text
Circle		Arc
Rectangle		Polygon
Bus		Net
Junction		Label
ERC		

Figure 5.7   Tool Box

1. $Info$ is used to give information about the component selected. First click on info tab than click on component whose information is required.

2. $Show$ is used to highlight any particular connection and track the actual interconnection of current segment Just by selecting show tab than a left click on required track.

3. $Display$ is used to select in between the various layers.

4. $Mark$ is used to place a reference symbol or mark to identify a point and to make reference measurements.

5. $Move$ is used to move the component on graphical area of schematic as well as layout

editor. It is done by a left click on + mark in mid of component than place it where required. We can also rotate the component by suitable no. of Right click.

6. $Copy$ command is used to copy an electrical component from existing schematic or layout.

7. $Rotate$ is used to rotate any component to any among four predefined rotations.

8. $Group$ is used to select a bunch of electronic components. A combination of Move + Group will allow to move a group or bunch of electronic components.

9. $Change$ is used to change any parameters or attributes of a component.

10. Cut is used to remove a block which can be placed elsewhere using paste command.

11. $Delete$ is used to remove any thing from graphical area.

12. ADD is very important tool in any schematic editor used to Pick n Place any component in schematic part.

13. $Gateswap$ is used to swap the connections done to the IC Package.

14. $Name$ is used to assign name to any component, wire or pin. If we name any two entities with same name on Schematic or Board Part that will be automatically connected in Board Layout.

15. $Value$ is used to define value of any component.

16. $Miter$ is used to provide a bending angle to the wire.

17. $Split$ is used to split a straight wire into a bended line.

18. $Wire$ is used to connect components it is defined as NET under display tab.

19. $Circle, Rectangle, Arc$ are used for adding any symbol.

20. Bus command is used to interconnect adjacent ports.

21. Net is used to draw an electrical interconnection using net wire.

22. $Junction$ is used to connect more than two overlapping but non touching wires.

23. $ERC - Electrical rule check$ command to use checks any electrical discontinuity in the circuit.

24. $Paste$ is used to put data in the paste buffer to any specific location.

25. $Value$ is used to define the value of an electrical component.

26. $Label$ is used to define / print the value of any pin, wire or electrical interconnection.

5.6 Entering a Schematic

1. Use the ADD command to place the devices.

   Please keep in mind:
You really should not change the default grid of 100 mil or .01 inch (= 2.54 mm) in the Schematic Editor. Only this way you can be sure that nets will be connected to the elements pins.

   2. You can toggle the grid on and off by clicking the GRID icon or more easily by using F6, to help you locating the parts. 3. Once you have placed the parts you can relocate them with the MOVE command. Activate the MOVE command by clicking the appropriate icon in the command toolbar, and then move the cursor to the part you want to move, click on a symbol [+]. EAGLE will highlight the part, to let you know that it is attached to the cursor and ready to be relocated.

4. Relocate the part, and again make a left click to place it in its new location. The MOVE command is still active and ready to move the next part. Press the right mouse button if you want to rotate a part.

5. For duplicating parts you may use the COPY command (for example, two similar capacitors). Thus you don't have to fetch each part with the ADD command.

6. When you have located the parts, start connecting them using the NET command.

Attention : *Use the NET command, not WIRE*! A net is only connected to a pin if it is placed on the connection point of the pin. Display the layer 93, Pins, with the DISPLAY command to locate these connection points. They are marked with a green circle. EAGLE automatically names electrical connections (nets). While the NET command is active, the status bar below shows properties of the selected net. → Nets with the same name define an electrical connection.

5.7 Power Connections

Instead of connecting power terminals of each part separately to single power output from Source better choice is to use Symbols +5v,+12v,GND from SUPPLY1 library. e.g. Just put a symbol of +5v to any number of points in schematic this will reduce no. of connections on your schematic part also will automatically connect the all points to source which is marked with +5v on Layout (Board) part.

5.8 Example of a Schematic

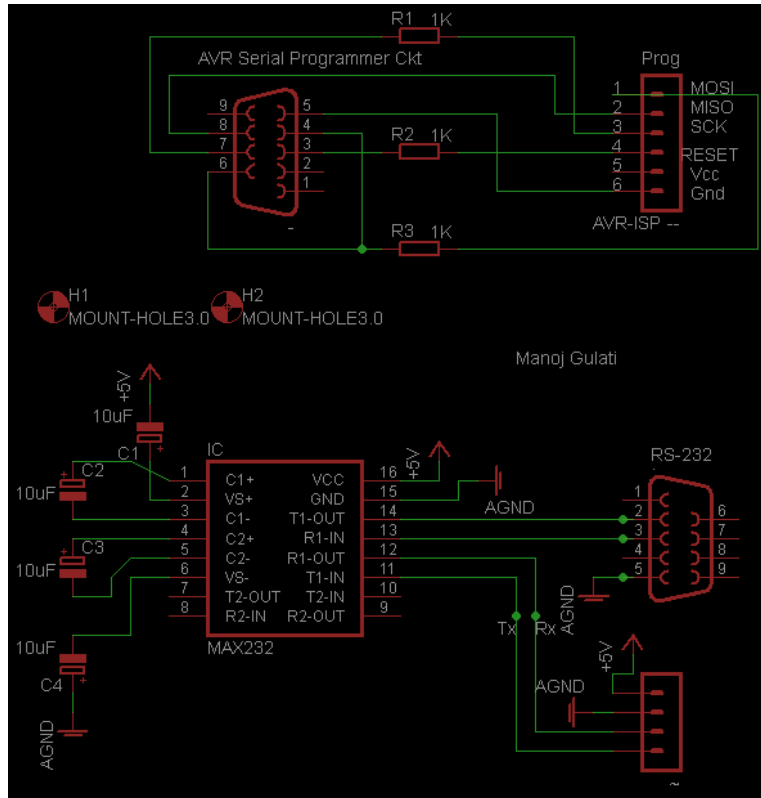A Schematic and its correspoding layout of the board is shown on the next page:-

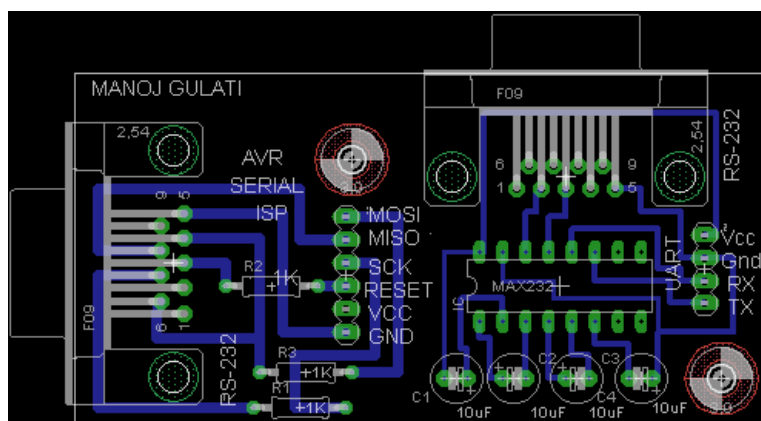**Figure 5.8**   Schematic example

Figure 5.9   Board layout corresponding to the above schematic

CHAPTER 6

GENERATING A BOARD FROM A SCHEMATIC

After loading a schematic from which you would like to design a board, click on the BOARD icon in the action toolbar:

Figure 6.1   BOARD button.

A board file will be generated in which the packages are positioned next to an empty board as shown below:

6.1   Defining Board Shape

The first thing we will do is define the shape of the board. Before defining the shape, we must establish the unit of measurement we will be using to draw the board outline. We want to use the default grid which can be chosen by clicking the GRID icon in the parameter toolbar. Then click the Default button and then OK.
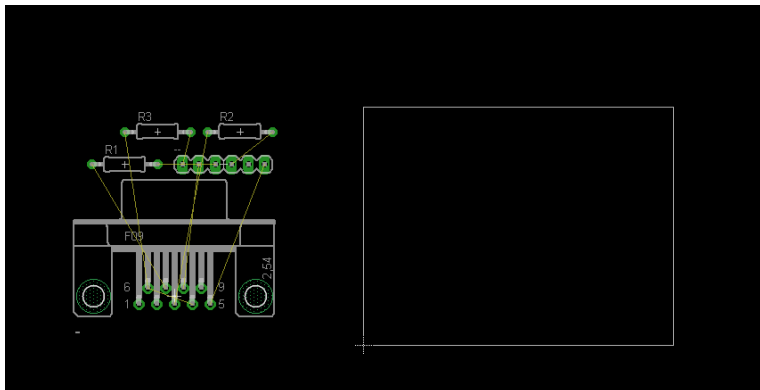
Figure 6.2   First screen when you open a board

The board outlines must be drawn with the WIRE command in layer 20, Dimension: click onWIRE and select layer 20 from the combo box in the parameter toolbar.
Position the cursor at the zero point of the coordinates, and left clickto define the starting point of the outline. Move the cursor slightly to the right, position the cursor near the coordinates say (4.00 3.00) and by double-clicking the left mouse button you will terminate the WIRE command. The board outlines are now defined. Using the MOVE command, the edges can be moved.

6.2   Component Placement

Now place the components as per your design considerations with following considerations the biggest IC somewhere in its centre. Move the components inside the board outlines. The component and the airwires remain attached to the cursor. → Press the right mouse button if you want to rotate the component. Left click to fix the position of the component. Place all of the components using the MOVE command.

Pleasenote : After generating a board file with the BOARD command EAGLE arranges all elements on the left side of the board outline in the negative coordinates area. In the freeware for example, you may drop elements within the limits of about 2.0 x 3.0 inch. To route the layout or to use the autorouter you have to move all components into this area first.

6.3   Auto-Router

If you would like to see a small demo of the Autorouter, click the icon for the AUTO command in the command toolbar. Choose a finer Routing Grid (default 50 mil) if necessary and click the OK button. It should be finished in no time at all, provided the placement is not too bad (watch the status bar).
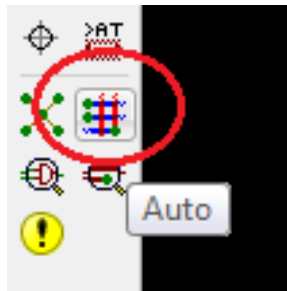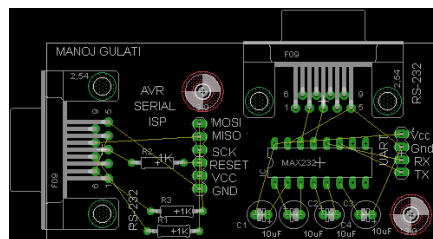
Figure 6.3   Auto Routing button

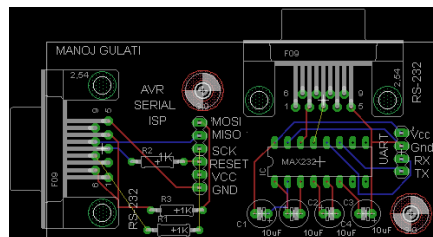**Figure 6.4**   Before Auto-Routing

Figure 6.5   After Auto-Routing

6.4   Routing Manually

The ROUTE command changes the airwires into routed tracks.
→ ROUTE in the command toolbar. Click starting point of an airwire. As for the WIRE command, further parameters, such as width or target layer, can be entered with in drop menus of the parameter toolbar. Move the cursor to route the signal again clickto fix the last segment and end the route operation for the whole signal.

Figure 6.6   Completed Board