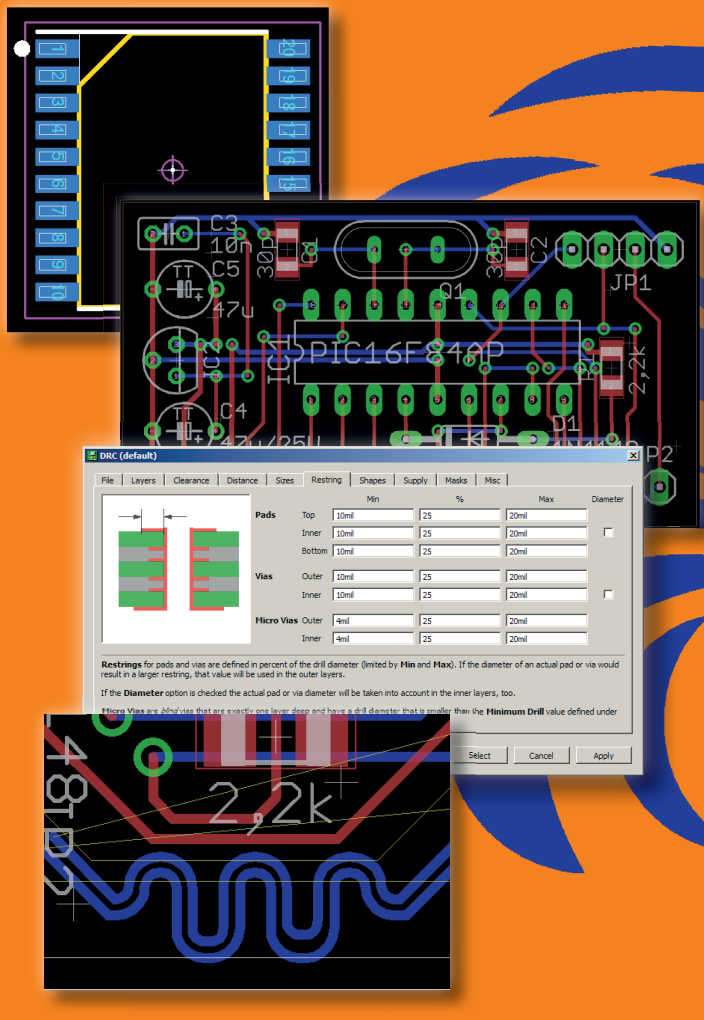
AN ADVANCED USER'S GUIDE

THE EAGLE COMPANION



Mitchell Duncan

elektor

LEARN • DESIGN • SHARE

Table of Contents

Chapter 1 ● An overview of EAGLE Software

Introduction

This book; the 'EAGLE Advanced User Guide' provides the experienced EAGLE user with insight into using some of the more advanced features of EAGLE software. It is not a guide to teach the reader the basic concepts of EAGLE, nor does it discuss the 'how to' of the EAGLE interface and the simpler operations and commands of the software. That is the purpose of the author's previous title 'EAGLE V6.0 Getting Started Guide'.

This book is intended as an enduring document covering the more advanced modules, commands, and functions which make up EAGLE. It is hoped that this book will sit on the desk or the bookshelf of the EAGLE user, and provide a quick, succinct reference to assist with more complex applications and uses of EAGLE – an 'EAGLE User's Companion', if you like.

Any book of this type can not, by its nature, be exhaustive. This is principally due to the fact that EAGLE software is continually evolving as CadSoftincorporates, as part of their normal software development cycle, increased functionality within the application.

At the time of writing, the material in this book covers version 7 of the EAGLE software suite.

What is EAGLE?

EAGLE stands for "Easily Applicable Graphical Layout Editor "; a professional-grade Computer Aided Design (CAD) software package for the design and drafting of electronic circuit schematics; and the design and fabrication of Printed Circuit Boards (PCBs).

EAGLE is a cross-platform application that can be used on Windows, Linux or Mac operating systems, with minimal differences. It uses the default language of the operating system, but can be configured to use other languages including; German, English, Hungarian, and simplified Chinese.

EAGLE allows the user to create schematic and PCB drawings and then output all the required files required to enable a fabricator to manufacture PCBs to the required specifications and tolerances.

The reader should refer to the company's websites for more information at:
www.CadSoftusa.com (English version) or www.CadSoft.de (German version).

Chapter 2 ● The EAGLE User Interface

The EAGLE Control Panel

When EAGLE starts and opens the Control Panel (see Figure 2-1), which is the main focus for the high level operations of EAGLE. It is EAGLE's control center, and brings together the management functionality of EAGLE into a single, unified interface.

The Control Panel's presentation is similar to a file manager utility, in that it presents 'panes' to allow the manipulation of files via a familiar 'tree –like' display structure. In general, the right hand panel displays a description, or more details, about the item selected in the left hand pane.

Once the user has selected an item in the left hand pane, that item can be manipulated through a set of menus. For example, a context menu is opened by Right-Clicking, with the mouse, on an entry in the tree view. This allows the user to, depending on the object, carry out a variety of actions (rename, copy, print, open, create new etc.).

The left pane tree structure provides a quick overview of the libraries, Design Rules, User Language programs, Script files, CAM jobs and Projects.

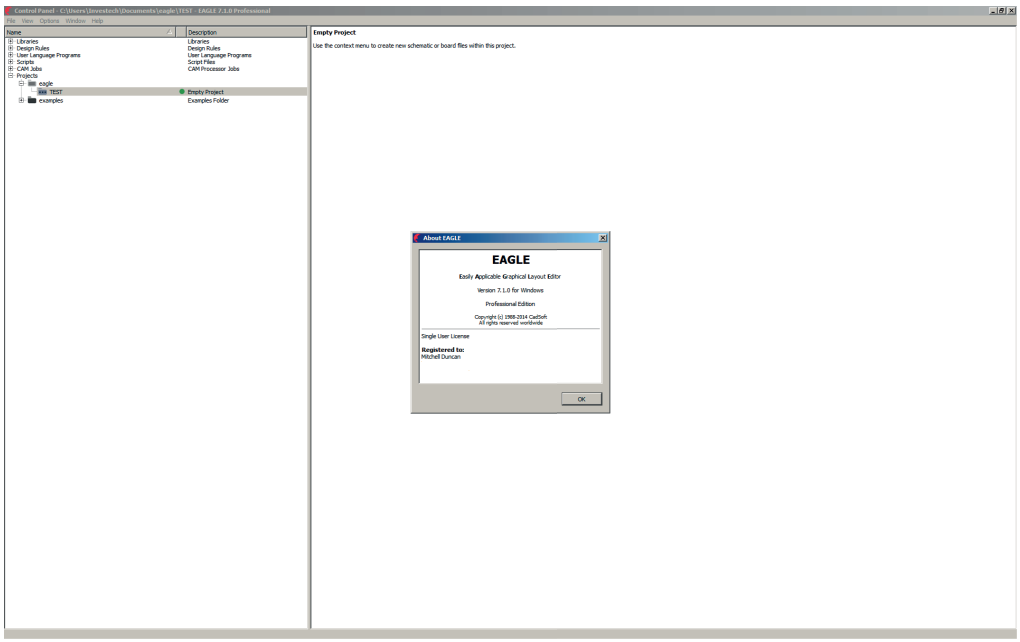


Figure 2-1: EAGLE Control Panel

Library Summary

A Library is a file which contains detailed information and specifications about devices which are available for use in both the Schematic and the PCB Layout Editor modules of EAGLE.

The user can quickly display the contents of a library by expanding any branch within the 'Libraries' tree. By doing this, the user can obtain a quick overview of the available devices within a particular library. Libraries can also be manipulated and edited using commands from within EAGLE.

If the user expands the Libraries entry in the left hand pane of the Control Panel, the user can view the available libraries. In the Description field the user will see a brief description of the contents (as in Figure 2-2). If a library is selected, the user will see more extensive information about the library in the right hand pane of the Control Panel.

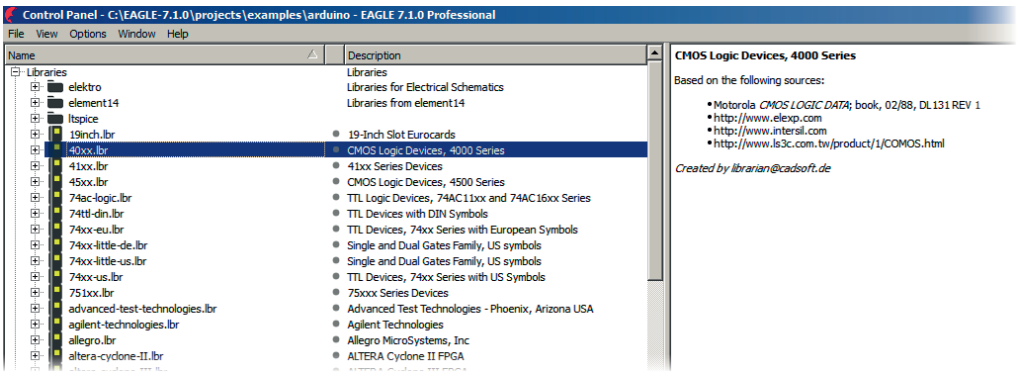


Figure 2-2: Libraries with description

Further, if the user then expands a library entry, the contents will be displayed together with a short description of each element. Devices and Packages are marked with a small icon; as in the left hand pane of Figure 2-3.

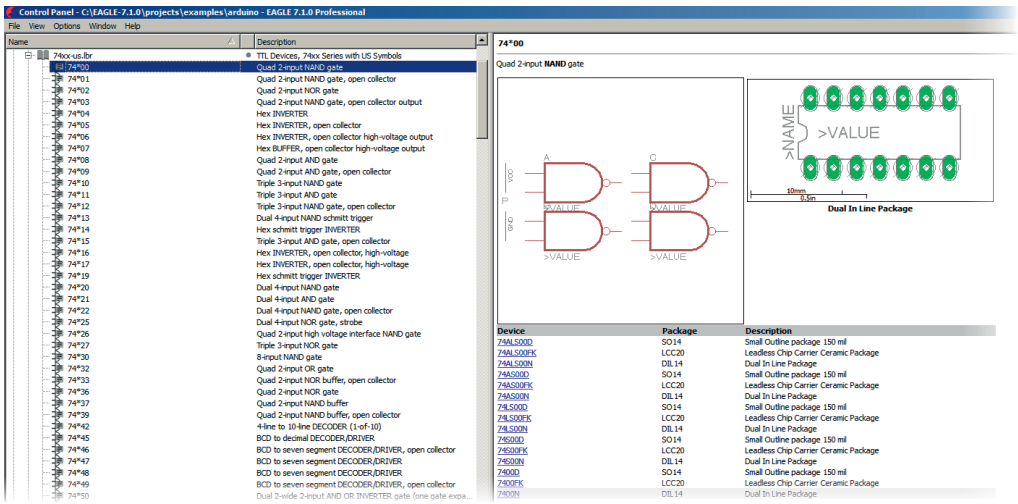


Figure 2-3: Libraries' Devices and packages

If one selects, for example, a Device; the description of the Device and a graphical representation of it appears in the right hand pane. The available Package and technology variants are also listed.

If one clicks onto one of the Package versions, the Package preview shown in the right hand pane will also change accordingly (see Figure 2-4).

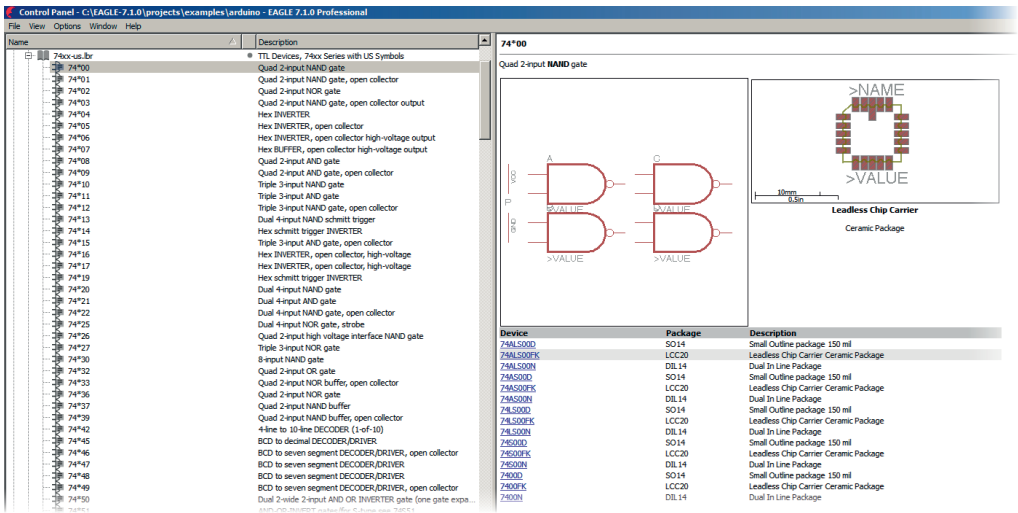


Figure 2-4: Libraries' alternative Device packages

The INFO Command displays the properties of the selected object. The name of the object may also be used as a parameter in the command line. Some of the properties of the selected object can be altered in this dialog.

The SHOW Command highlights the object selected with the mouse Left-Click. In case of objects that consist of more than one part, like elements with several gates or nets that spread over several sheets, the window will highlight all elements.

The DISPLAY command allows the user to select and deselect the layers to be displayed in the current drawing.

MARK Command allows the user to define the new origin for the coordinate display, relative to the drawing Datum point. Relative coordinates (R x-value y-value) and polar values (P radius angle) are shown in addition to absolute coordinates in the coordinate display box.

The MOVE Command allows the user to move and rotate any visible object or Group.

The COPY Command permits the copying of devices, components, Nets, Busses and Groups.

MIRROR Command transforms an object into its horizontal mirror image equivalent.

The ROTATE Command allows the use to rotate the orientation of an element or group, by 90 degrees.

Figure 2-35 — Schematic Editor command toolbar

The GROUP Command allows for a selection of devices to be associated and managed together, such that they may be manipulated using subsequent commands.

CHANGE Command enables the user to alter the Properties of the current drawing; or of an object. (See the INFO Command).

PASTE Command inserts objects from the clipboard buffer into the drawing

DELETE Command allows deletion of a visible device or group.

ADD Command permits adding/inserting device elements from a pre-selected Library.

PINSWAP Command swaps two nets connected to equivalent pins of a Device, that have the same defined Swap-level.

REPLACE Command replaces a device or component with another from any Library.

GATESWAP Command Swaps two equivalent Gates of a Device that have the same defined Swaplevel.

NAME Command allows the user to create or change the name of Devices, Nets or Buses.

VALUE Command allows the user to create or change the value of devices.

SMASH Command separates the name, value and any attribute texts from a Device, allowing them to be manipulated individually.

MITER Command allows the user to round off or bevel the joints of Wires, Nets, Buses and Polygon contours.

SPLIT Command inserts an angle joint into a Wire or Net.

INVOKE Command allows the user to fetch & add individual Symbols (Gates) from Devices that consist of more than one Symbol (Gate). The command also allows the fetching and adding of a Gate from a Device which is located on another sheet.

The WIRE Command is used to draw a line (WIRE is used to define electrical connections in the Layout Editor).

The TEXT Command is used to place text within the drawing.

CIRCLE Command is used to draw a circular line within a drawing.

ARC Command is used to draw an arc, or partial circle, within a drawing

Chapter 3 ● Working with EAGLE

Command Input Alternatives and Navigating Around EAGLE

Icons for Commands

Within each Window of EAGLE there are many graphical icons. Icons appear in menus and toolbars; and typically represent, and may substitute, an EAGLE Command (which could be entered via the Command Line). Many of EAGLE's Command icons are shown in the 'Toolbar' sections outlined earlier in this book.

The first action in using an icon is to activate the corresponding command. These may be:

Left-Click on an icon and the user will typically invoke the corresponding Command.

The next action for a Command is usually displayed in the Status Bar at the bottom, left hand corner of the Window display.

Right-Click on an icon, or a component, and EAGLE typically activates the associated Context Menu.

Use of Mouse Buttons & Wheel

Left Mouse Button

The left mouse button is generally used to select, draw or place objects.

Mouse Wheel

The user may zoom in and out on a drawing, whilst inside an editor window, by scrolling the mouse wheel. The mouse wheel usually doubles as a Center Mouse Button. The option setting 'Mouse wheel zoom' determines the zoom factor for each 'step' of the mouse rotation. (refer to the Options User Interface)

Center Mouse Button

Using the center mouse button changes the current layer of, or mirrors, the object currently attached to the mouse cursor during an active command.

Table 3-1 lists the commands' functionality and response when used with the center mouse button:

Mirror Part	Change Active Layer
ADD	ARC
COPY	CIRCLE
INVOKE (mirror gate)	LABEL
MOVE (mirror object or group)	POLYGON
PASTE (mirror group)	RECT
	ROUTE
	SMD
	TEXT
	WIRE

Table 3-1: Command actions for center mouse button

The center mouse button may also be used to Click&Drag a drawing i.e. it will pan the drawing within the editor window.

Right Mouse Button

The right mouse button is mostly used to select a group, rotate objects attached to the mouse cursor, change wire bend styles and several other command specific functions.

If an object is selected with the right mouse button, then a context-specific menu is displayed. The menu contains commands which may be used, and that apply to this object.

If a group has been selected and there is currently a command active that can be applied to a group, then using the right mouse button will invoke a context-specific menu that will contain an entry for this group.

Table 3-2 lists the command with which the user may use the Right Mouse Button, together with the command action response to this.

	Command	Right Mouse Action Response
'Change' Commands		
	ARC	change direction of arc
	BUS	change wire bend
	CHANGE	apply change to group
	NET	change wire bend
	POLYGON	change wire bend
	ROUTE	change wire bend
	SPLIT	change wire bend
	WIRE	change wire bend
'Rotate' Commands		
	ADD	rotate part
	INVOKE	rotate gate
	LABEL	rotate label
	PAD	rotate pad
	PASTE	rotate group
	PIN	rotate pin
	ROTATE	rotate group
	SMD	rotate SMD
	TEXT	rotate text
	MOVE	rotate object
'Group' Commands		
	DELETE	delete group
	GROUP	close polygon
	MIRROR	mirror group
	MOVE	select group
	RIPUP	ripup group

Table 3-2: Command actions for Right Mouse Button

Selecting objects in densely populated areas

When the user tries to select an object at a position where several objects are placed close together, a four way arrow and the question;

Select highlighted object? (left=yes, right=next, ESC=cancel)

will be displayed, indicating that the user may now choose one of the following:

Right-Click will switch to the next object;
Left-Click will select the highlighted object;


will cancel the selection procedure.

If the original selection was done with the right mouse button, a context specific popup menu will be displayed. This menu applies to the first selected object, and contains Next as the first entry. Repeatedly Left-Clicking on this entry will cycle through the objects within the selection radius.

EAGLE Project File

When a new project is created (for example, from the Control Panel Menu File ▷ New ▷ Project) a directory is created in the Projects branch of the tree view. Further, the *eagle.epf* configuration file (Project File) is automatically created in that project directory.

The Project File stores the following information about a project:

1. changes to object properties that are made with the CHANGE command during editing;

2. the contents of the *Width*, *Diameter*, and *Size* menus;

3. the libraries in use for this project;

4. the position and contents of the active windows at the time the program is closed.

   (This assumes that the *Automatically save project file* option under *Options* ▷ *Backup* in the Control Panel is active. This state will be recreated the next time the program starts.)

Configuration Menus & Commands

Most of the options within EAGLE are set by means of the Options menus within the individual EAGLE editor window.
The Control Panel allows settings to be configured for *Directories*, file *Backup* and the visual appearance of the editor window (*User interface and Windows Positions*). These options are described in the earlier section of the book titled; 'Options Menu'.

The Options Menu

Using the *Options* ▷ *User interface* settings it is possible to select the icon-based menu or a configurable text menu, the latter being one of the most powerful methods of configuring EAGLE. Specifically, it is possible to construct custom menus, comprising a hierarchical structure of text items and associated icons, by using the MENU command within a script file.

The User interface dialog is shown in Figure 3-7

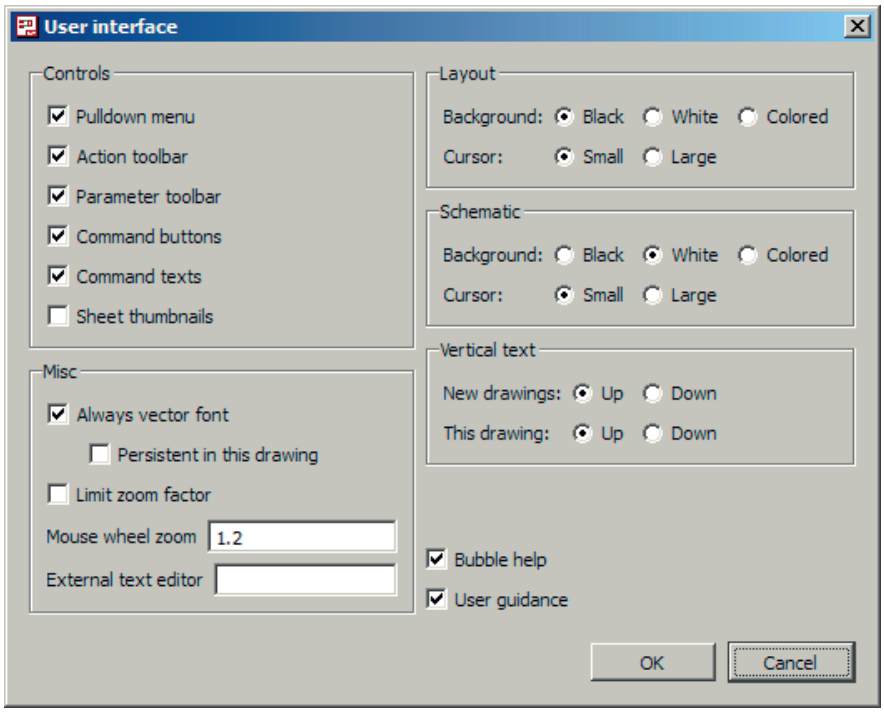


Figure 3-7: User interface dialog

The *Options* menu in the editor windows of schematic diagrams, layouts and libraries, contains two other entries: the *Assign* and *Set* items.

The ASSIGN command alters and displays the assignment of the function keys. This was discussed previously in the section titled:

Selecting objects in densely populated areas

When the user tries to select an object at a position where several objects are placed close together, a four way arrow and the question;

Select highlighted object? (left=yes, right=next, ESC=cancel)

will be displayed, indicating that the user may now choose one of the following:

Right-Click will switch to the next object;

Left-Click will select the highlighted object;

will cancel the selection procedure.

If the original selection was done with the right mouse button, a context specific popup menu will be displayed. This menu applies to the first selected object, and contains Next as the first entry. Repeatedly Left-Clicking on this entry will cycle through the objects within the selection radius.

The selection radius may be defined using the command:

```
SET Select_Factor select_radius command.
```

Overall EAGLE parameters are altered with the SET command. These can be invoked using SET from the command line or from the Options ▷ Set… menu option. In either case, these invoke the dialog shown in Figure 3-8.

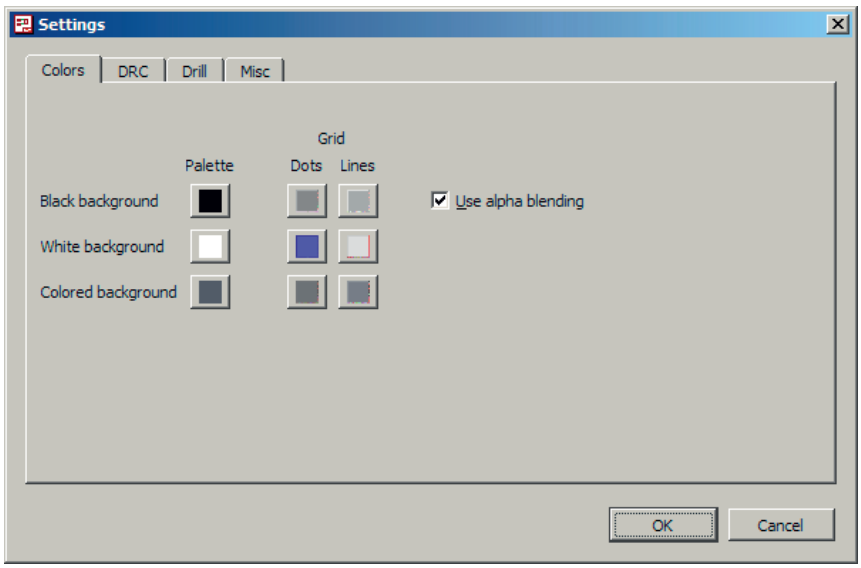


Figure 3-8: Options Settings dialog

Setting Colors

The first tab in the Options Settings Dialog is the *Colors* tab. This allows the user to set colors for the layers and background of a drawing as well as for its grid lines or dots.
EAGLE offers three color palettes. These are for black, white and colored backgrounds. **Left-Clicking** on any of these 3 palette icons activates the Color Palette Configuration Dialog of Figure 3-9.

It can be seen that each palette allows a maximum of 64 color entries, which can be configured to any RGB or Alpha channel value. This can be seen in the 'grid' of color boxes in the Color dialog.

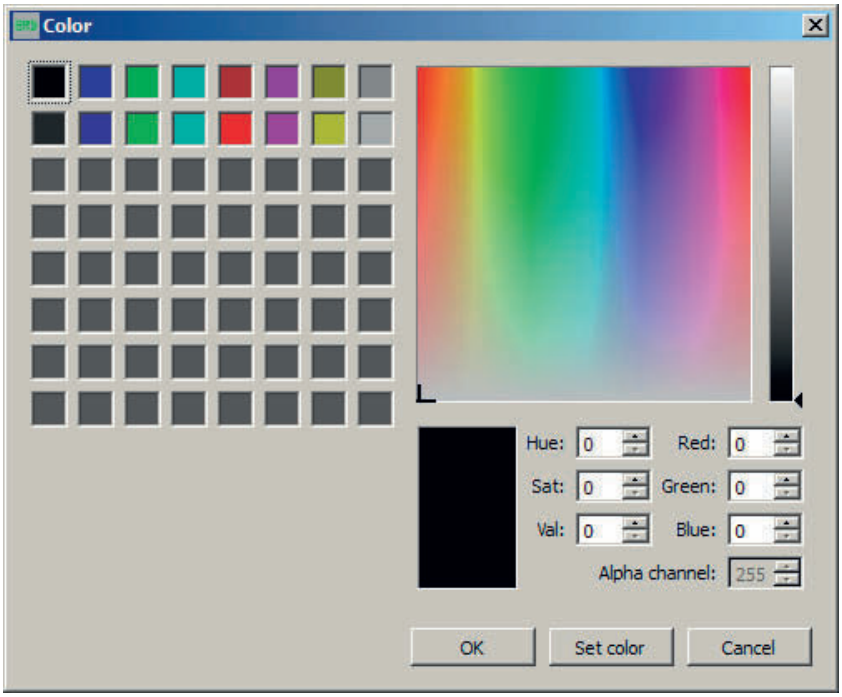


Figure 3-9: Color palette configuration dialog

By default EAGLE uses the first 16 values (entries 0..15) in the palette; eight colors followed by a further eight 'so-called' highlight colors. The first entry of the palette determines the background color. However, in the white palette, the background color is locked, since printed output is normally made on white paper. A color of the palette at position *x* can be given the corresponding highlight color at position *x*+8.

The user may define a new color by selecting an undesignated box in the matrix and then adjust the color in two ways:

i. using the color selection area and the saturation bar to the right of the matrix;

ii. entering values for *Red, Green, Blue , Hue, Sat, Val* and Alpha channel directly into the numbered data windows.

The *Alpha channel* determines the transparency of the color. (The value 0 means the color is totally transparent (invisible), the maximum value 255 means the color is opaque). For printouts the value of the alpha channel should be set to 255 for each color. The alpha value is ignored when using a black background.
Older versions of EAGLE did not use the current technique of mixing colors using an OR function. To turn off the newer functionality, uncheck the *Use alpha blending* check box.

Chapter 4 ● EAGLE Window Commands

This section of the book describes the most commonly used commands of EAGLE; and is grouped in the context of how the command is used within the application. These commands will normally be entered on the command line of the user interface; however some commands may be activated via a menu item or icon.

Author's Note: Some material in this chapter is taken directly from a number of EAGLE documentation sources. This has been done in order present this information in a more convenient, accessible and integrated form; and to relate it contextually to illustrated user actions.
The author acknowledges that full ownership and rights of this material reside with CadSoftComputer GmbH.

Commands for Changing Modes or Files

This section discusses the most frequently used file manipulation commands. They apply to all EAGLE modules, except where indicated.

OPEN

Function
Opens a library for editing.

Syntax
OPEN library_name

See also CLOSE, USE, EDIT, SCRIPT

The OPEN command is used to open an existing library or create a new library. Once the library has been opened or created, an existing or new symbol, device, or package may be edited.

This command is mainly used in script files.

Examples
The command:

```
OPEN linear
```

would open the library 'linear.lbr' in a new window.

Commands to Change the Screen Display and User Interface

ASSIGN

Function
Modify key assignments.

Syntax

```
ASSIGN
ASSIGN function_key command..;
ASSIGN function_key;
function_key = modifier+key
```

The options are as follows:

`modifier` any combination of S (Shift), C (Control), A (Alt) and ⌘ (Cmd, Mac OS X only)
`key` representations of keys in the ranges; F1..F12, A-Z, 0-9, BS (Backspace)

If `key` is one of A-Z or 0-9, the modifier must contain at least A, C or M.

The ⌘ modifier is only available on **Mac OS X**.

See also SCRIPT, KeyboardandMouse

The ASSIGN command can be used to define the meaning of the function keys F1 through F12, the letter keys A thru Z, the (upper) digit keys 0 thru 9 and the backspace key (each also in combination with modifier keys).

Keys can be assigned a single command, or significantly, multiple commands. The command sequence to be assigned should be enclosed in apostrophes.

If assignmed to a letter key together with the modifier A, (e.g. Alt+F), a corresponding hotkey from the pulldown menu will no longer be available.

To remove an assignment from a key, the user may enter the ASSIGN command with only the function_key code, omitting the command.

Note: any special operating system function assigned to a function key, will be overwritten by the ASSIGN command (depending on the operating system, ASSIGN may not be able to overwrite certain function keys).

The ASSIGN command without parameters displays the present key assignments in a dialog, as shown in Figure 41, which allows the user to modify these settings.

The values assigned to function keys may also be viewed from within a Schematic or PCB Layout Window by using the sub-menu: Options ▷ Assign, again, activating the dialog show in Figure 4-1.

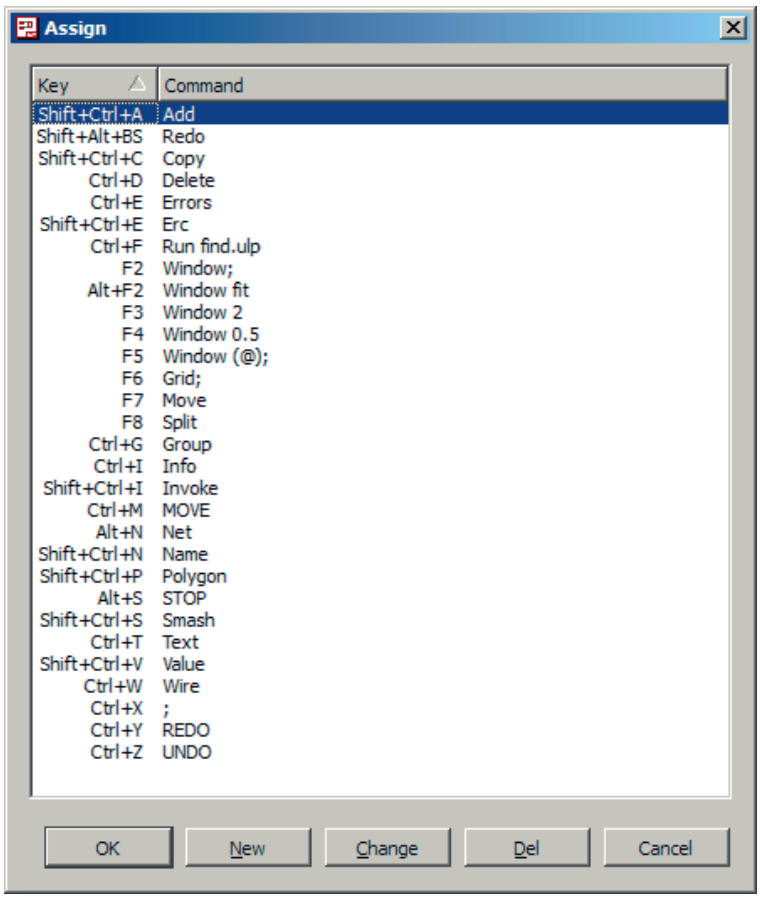


Figure 4-1: function key assignment dialog

Chapter 5 ● Using EAGLE's Library Manager & Editor

The EAGLE application is supplied with a large number of library files containing components ranging from small surface mount devices through to large board mounted hardware (like connectors and sockets). This can be seen simply by expanding the tree view of the 'Libraries' entry of the EAGLE Control Panel.

As the user develops a circuit and realizes this design in the schematic and PCB layout respectively, components are accessed from the appropriate libraries and inserted into the (schematic or layout) drawing. The target drawing file will then contain all necessary information about those included components. This precludes the need to distribute libraries when sending the drawings to reviewers or to a PCB fabricator.

EAGLE provides a powerful editor for creating and manipulating components and their constructs (i.e. Package, Symbol and Device).

Library and Component Management

EAGLE's Library Editor enables the user to manage libraries, parts and components in order to tailor the application to most needs, or conversely, to create a specific library for a particular project environment.

A library contains the basic elements of an electronic/electrical component, which comprise the following:

- **Package:** the footprint of the component when used in the layout drawing
- **Symbol**: the representation of the functional description of the component which is used in the schematic drawing
- **Device** the real component, consisting of a combination of symbols and packages

The Library Editor allows the user to create, copy, edit and delete any library elements within any library.
For example, elements may be created from nothing, or they may be copied from an existing combination of elements to create a wholly new component. Therefore all elements may be freely manipulated with the Library Editor.

This can be done from the Library Editor window by **Left-Clicking**  (on the left-hand side of the window), which will open the component Edit window (as shown in Figure 5-1). It is from here that Packages, Symbols and Devices may be edited and created.

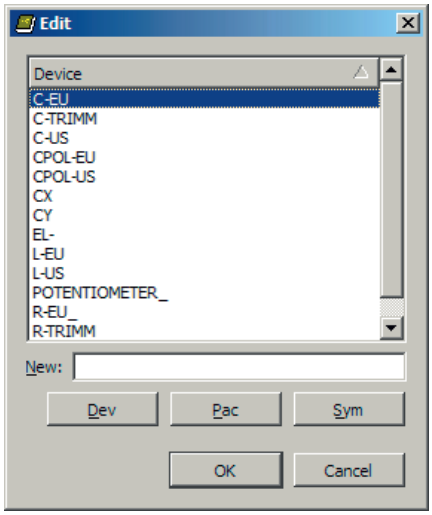


Figure 5-1: the component Edit window

Overview of Defining a Complex Device or Component

Using a combination of EAGLE's functionality; the Library Manager and the Package and Symbol Editors, it is possible to construct complex components for use in PCB design.

Initially we will outline here, a summary of the steps required to create a new component for inclusion into a library:

Creating a new or use an existing library

The user may choose to

i. create a new library using File ▷ New ▷ Library from the menu toolbar in the EAGLE Control Panel.

or

ii. expand an existing library using File ▷ Open ▷ Library from the menu toolbar in the EAGLE Control Panel (or alternatively use the context menu within the Control Panel's tree view).

Creating the housing or packaging

The next step is to create the package or housing for the new component. This may be copied into the open library, from another library, using EAGLE's COPY command or by Drag&Drop from the Control Panel.

However, if the package does not exist, the user might need to create their own package library element. In general there exists two major types of packages: pinleaded and SMD (surface mount devices). Of course as advances are made in the component packaging industry, other types of packaging technologies will emerge.

The overall process of creating a package is similar to both types of connection. In general, the basic stages for package construction are:

Edit package
Open the package Editor and create a new package within the dialog, naming it accordingly.

Set the grid parameters
Set the appropriate grid parameters for the package. These are determined by the type of package (e.g. 0.1 inch for DIL packages). It is advised to set an alternative grid in another linear measure (e.g. inches with alternative millimeters).

Place the Pins/SMD Pads
Using the Pad or Pin command place the pins/pads according to the specifications in the package datasheet. Select the appropriate parameters/options for each contact type (i.e. diameter, shape, orientation etc. for Pads; x_width, y_width, roundness etc. for SMD pads).

Name the Pins/Pads
EAGLE normally automatically names Pins/Pads as they are placed. This can be turned on or off and can be set by Options ▷ Set ▷ Misc menu and choosing the *Display pad names* option.

Draw the Silk Screen

The silk screen is important insofar as it is the visible indicator to the PCB fabricator and user of the board, as to package placements on the board. Care should be taken to avoid screening over potential solder areas.
The silk screen is normally created in the print layer21, *tPlace* ; although other silk screening may be placed in layer 51, *tdcou* for additional documentation purposes.

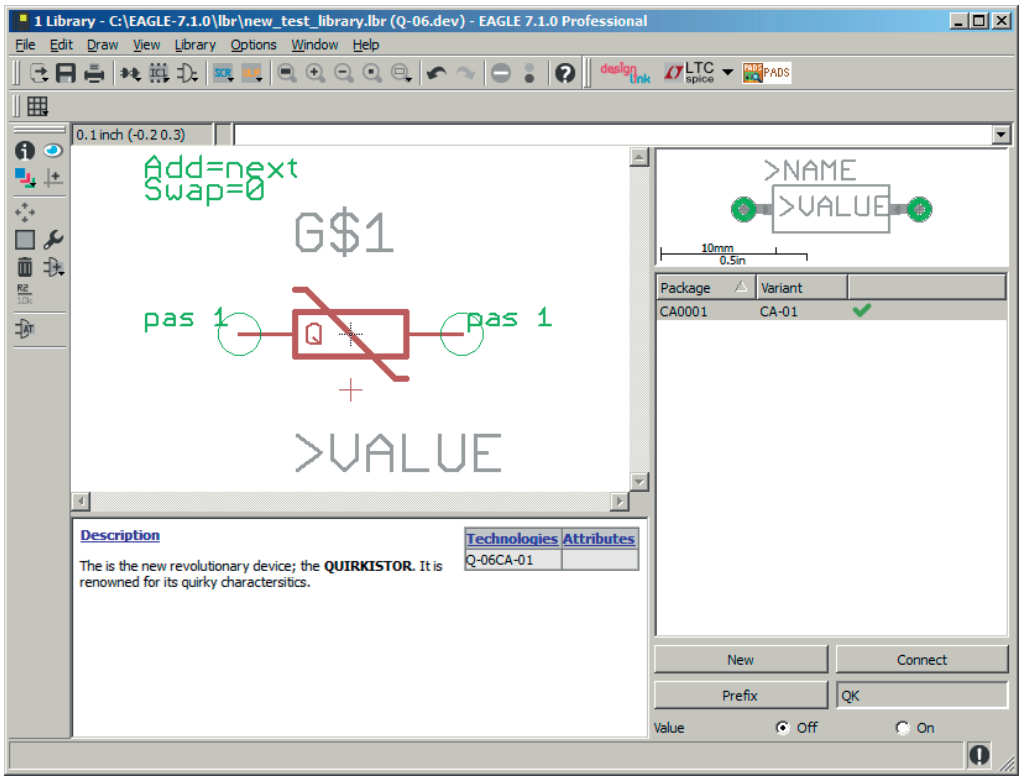


Figure 5-25: the completed QUIRKISTOR Device

Creating the 'Kunda Buffer' Component

The 'Kunda Buffer' is an analogue component with buffering (time delay) characteristics. It has a single input and 4 in-line outputs and comes in 2 standard packages: SOIC8 and DIL8.
The first step is to define the packages for this device. It is a small-signal, linear, analogue, low-power device and can be housed in a range of packages. This example will use the pre-existing specifications, '`SOIC8`' and '`DIL8`', from the library '`ref-packages.lbr`'.

Step 1 - Copy the packages to the new library
As described previously, the user may copy the package to the new library, or use the following method:

i. Open the library '`new_test_library.lbr`' (our nominated target library)

ii. From the command line in the new library, copy the first Package using the command:

```
COPY SOIC8@ref-packages.lbr
```

This will copy/insert the Package specifications into the new library, the new window appearing as in Figure 5-26.

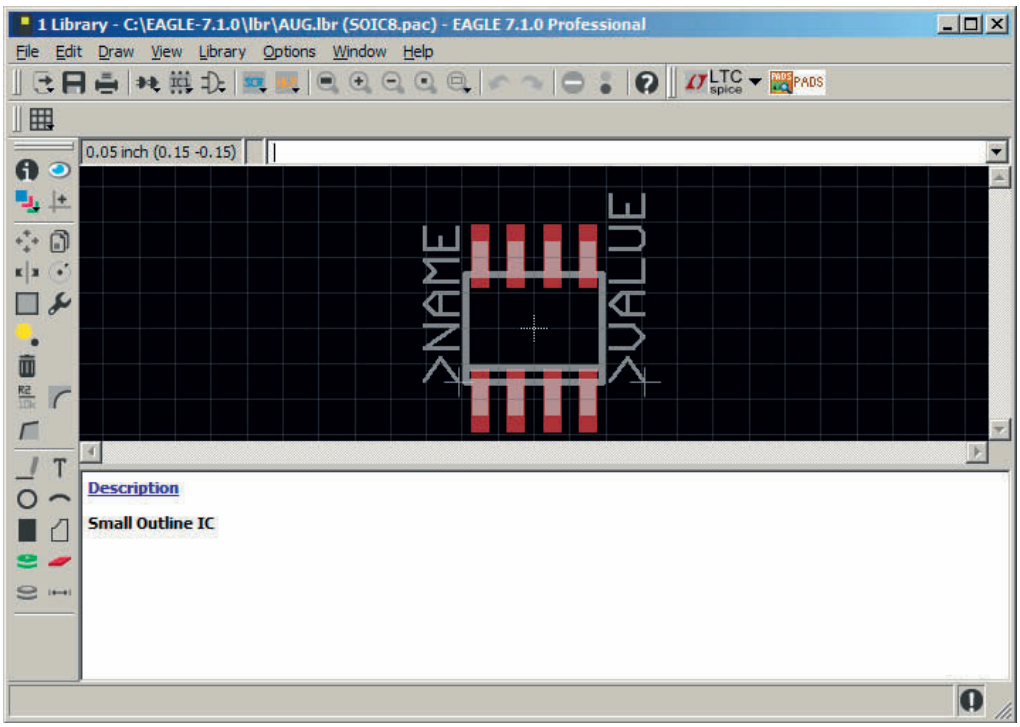


Figure 5-26: copied SOIC8 package

iii. Save the library by using File ▷ Save, `Ctrl-S` or `Left-Click`  .

iv. Again, from the command line in the new library, copy the second Package using the command:

```
COPY DIL08@ref-packages.lbr
```

These Package specifications will also be copied into the new library, the new window appearing as in Figure 5-27.

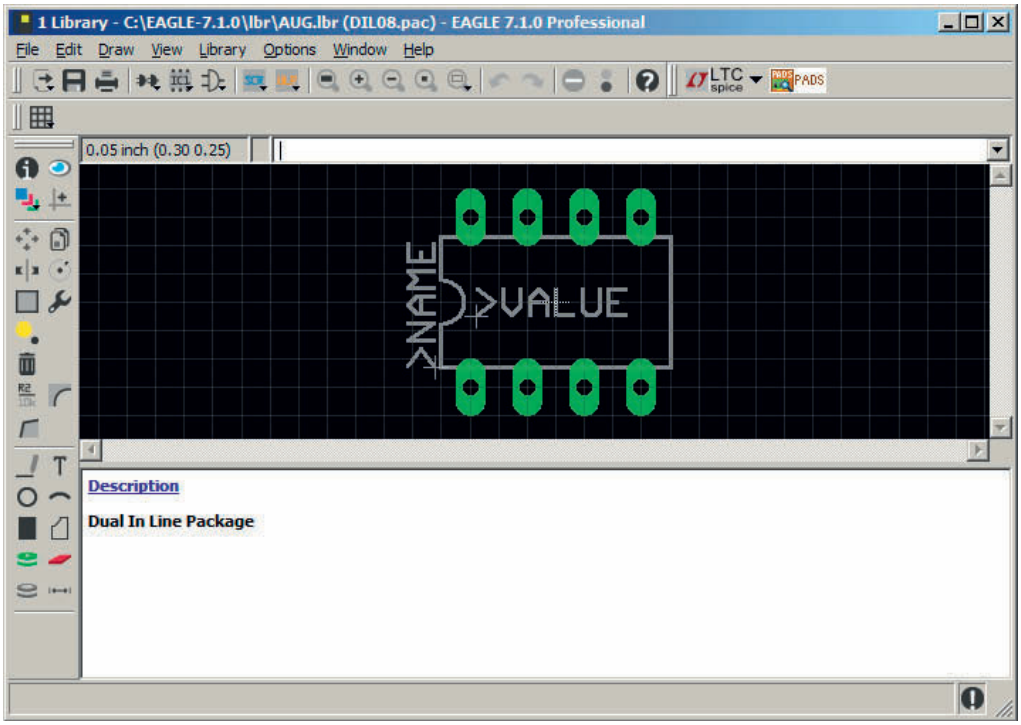


Figure 5-27: copied DIL08 package

v. Again, save the library by using File ▷ Save, `Ctrl-S` or **Left-Click**  .

The user can check that the packages have been added to the library by using the Edit Package icon (**Left-Click** ).

Step 2 – Create a new Symbol
In this step the user will create a new Symbol (the 'Kunda Buffer') and add it to the library. The Symbol is illustrated in Figure 5-28.

Chapter 6 • Using EAGLE's ROUTE command and Follow-Me router

The EAGLE application provides the user with the ability to perform the routing of tracks automatically (using the Autorouter); manually by using the ROUTE command alone, or with the additional functionality of the 'follow-me' router.

The ROUTE command

This section discusses the application of the ROUTE command, which is activated by **Left-Clicking** on  .

Upon activation, the command's parameter toolbar is displayed (as shown in Figure 6-1). These parameters can be set prior to the commencement of routing a track (or trace); or dynamically, while the routing is in progress.

ROUTE Parameters

The ROUTE parameter toolbar is shown in Figure 6-1. The toolbar encompasses a number of parameters which determine how the command behaves. They are *Grid*, *Layer*, *Wire Bend Style*, *Miter Radius*, *Miter Style*, *Wire Width*, *Via Shape*, *Via Diameter* and *Drill Diameter*.

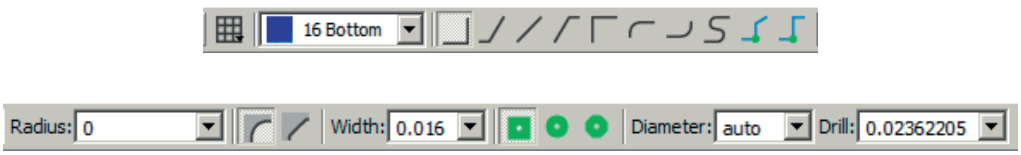


Figure 6-1: ROUTE parameter toolbar (split)

The parameters : *Grid*, *Layer*, *Wire Bend Style*, *Miter Radius*, *Miter Style* & *Wire Width* have the same functionality as described for the WIRE command.

The parameters : *Via Shape*, *Diameter* and *Drill* have the same functionality as described for the VIA command.

The route layer

The initial layer of a track is generally determined by the object from which the selected airwire starts i.e. the track adopts the currently selected layer if the object is in it, otherwise another layer of the commencement object will be adopted.

The **Center-click** mouse action may be used to change the track layer.

The wire width and via parameters

When selecting an airwire for routing , the wire width is determined by the setting in the Design Rules and the net class of the selected signal and the status of the flag Options ▷ Set ▷ Misc ▷ Auto set route width and drill. This is shown in Figure 6-2 . If this is set off (i.e. unchecked) the user is able to manually set the wire width from the drop-down menu in the parameter toolbar. Otherwise, if set on (i.e. checked), the width is automatically calculated and set. This option also controls the via Diameter and Drill parameters.

The width of an airwire that starts at an already existing track, may be automatically set to that track's width by using the key/mouse combination **Shift+Left-Click** before selecting the airwire.

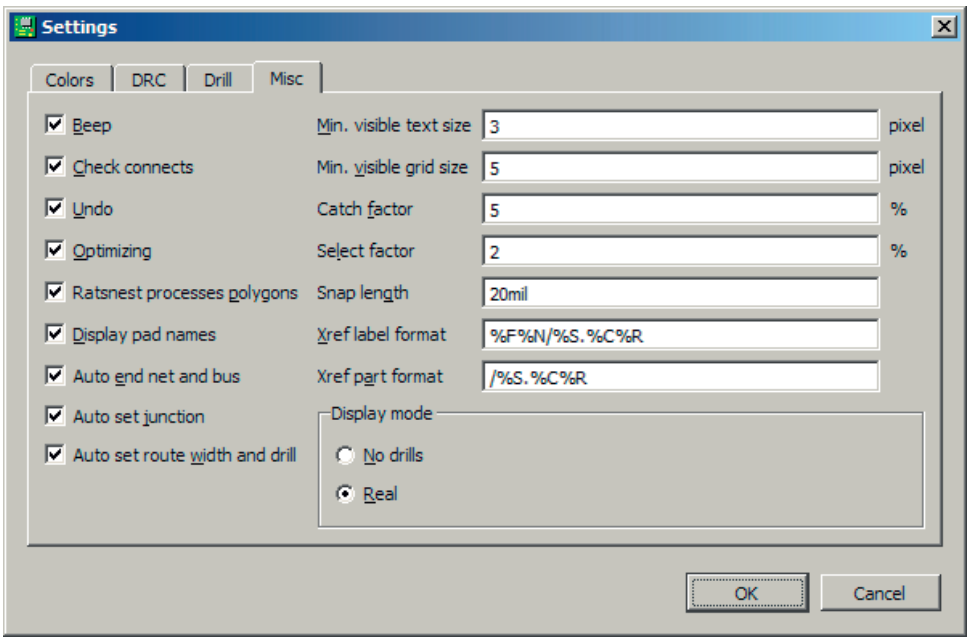


Figure 6-2: Options settings

The user can select a different wire trace width, while the airwire is attached to the cursor, by using the relevant SET command from the keyboard. The most recent track segment will be rerouted with the new width. e.g. `SET width 25`

The same comments apply to the via parameters (i.e. those displayed in the toolbar).

Chapter 7 • Using New EAGLE V7 Features

Autorouter Features

In V7 of EAGLE, enhancements to the Autorouter module allow it to support the calculation of multiple autorouter jobs at a time, and for it to use multicore processors.

The enhanced features incorporate some new setup options including; the automatic choice of the routing grid, or automatically determining the preferred directions in the signal layers. The setting for *effort* determines the number of calculated routing jobs. Each of the proposed jobs can be customized.

Additionally, a new capability allows the user to select the socalled “TopRouter” as a kind of prerouter. It functions with a new gridless algorithm which first tries to establish the trace connections, and then uses the classic EAGLE Autorouter for optimization. By way of example, Figure 7-1 shows a sample PCB which has been routed without selecting the TopRouter capability, while Figure 7-2 shows exactly the same layout using the AutoRouter with the TopRouter capability selected.

This section does not discuss the Autorouter functionality, since it is already comprehensively explained in the EAGLE V7 Manual (see Chapter 7 – The Autorouter).

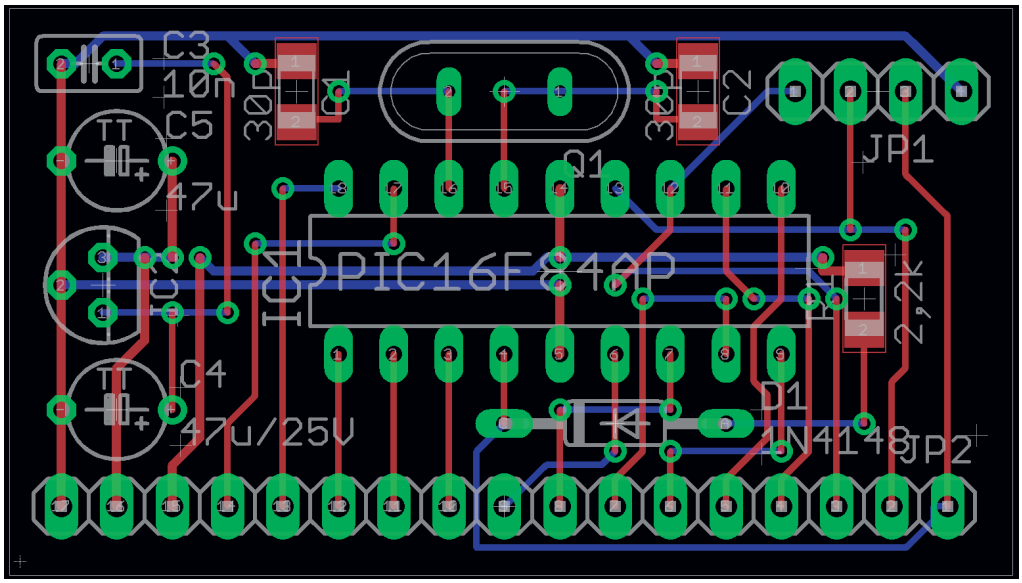


Figure 7-1: Sample PCB Layout executing the Autorouter without using the TopRouter

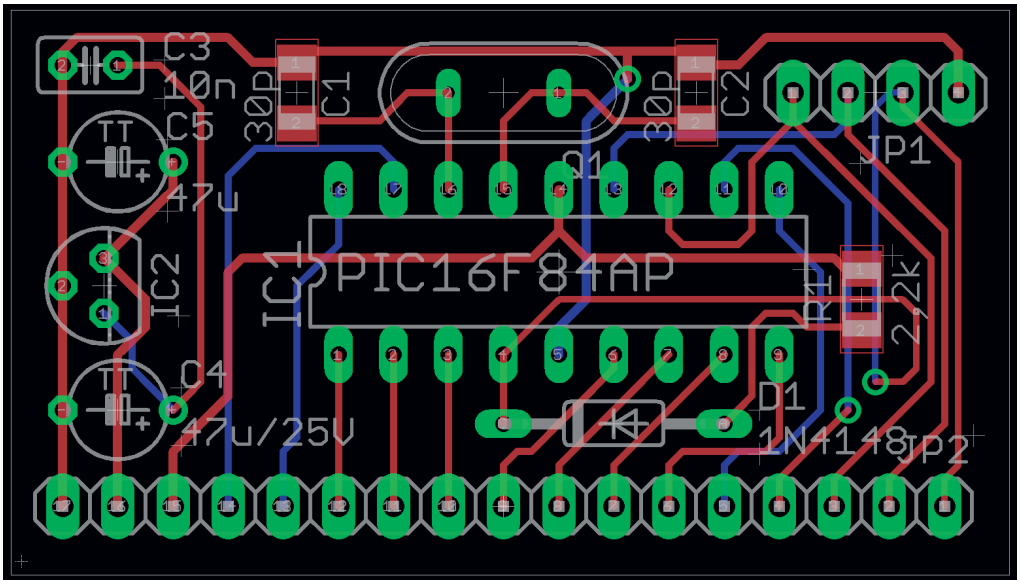


Figure 7-2: Sample PCB Layout executing the Autorouter and using the TopRoute

Hierarchical Schematics in EAGLE

One of the new features of EAGLE V7 is the ability to create a structured schematic, organized in a hierarchical form. It achieves this by using sub-units called 'Modules' and their functional interfaces called 'Ports'. The hierarchy can have any depth. The resultant PCB layout is comparable to the layout generated from a non- hierarchical schematic of the same design.

Figure 7-3 illustrates a project schematic which has been constructed using this hierarchical method. A brief examination of this project shows that it differs from a normal schematic diagram. It consists mainly of 'blocks', which are interconnected via nets at the interface points. In fact, the 'blocks' are examples (instances) of EAGLE *Modules* which have *Ports* for the interconnection of *Nets* (their behavior is somewhat similar to that of Pins).

Modules are schematics, represented by a 'block' in the higher level schematic diagram of the circuit. These blocks represent instances of the constituent circuit within the complete design and may be used repeatedly within that design. These can be seen in the Sheet toolbar of the main Schematic Editor window (see the left hand side of Figure 7-3).

Ports allow for the interconnection of Modules at the main circuit diagram, or at some other level i.e. schematics which make up a Module may themselves contain Modules. The depth of the diagram hierarchy may be arbitrarily deep.

So, in the top level circuit diagram of the Project, the Modules (instances) use a network (of Nets or simple Buses) to interconnect via Ports. A Port therefore serves as an interface for the nets within a module to the outside world (or the next level 'up' in the hierarchy)

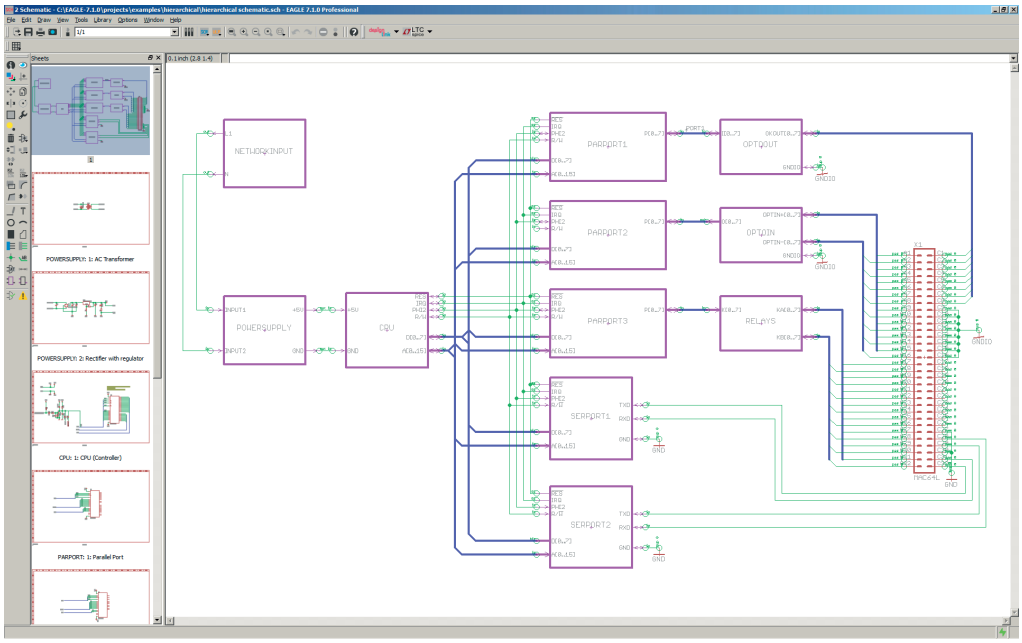


Figure 7-3: Editor window displaying a hierarchical schematic project

Working with the Hierarchical Schematics Elements

This section will illustrate the features and methods of working with the basic elements of the new hierarchical structure of EAGLE V7. The schematic in Figure 7-3 will serve as a reference for some following examples.

It is assumed that the reader has a working knowledge of EAGLE, in particular; how to create schematic diagrams with the Schematic Editor module of the EAGLE software.
For the purposes of this example, we assume that a new, blank schematic has been created. This will be used to illustrate how to work with the various elements of the hierarchical structure functionality.

Working with Modules

As stated previously, Modules are a building block of the hierarchical schematic drawing. In this section the process of creating, editing and deleting a Module will be illustrated.

Appendix 1 ● Useful Industry References

This Appendix brings together a collection of data from diverse sources to serve as a convenient reference point for the reader.

Some of this data is extracted from industry standards and presented in a form which is more useable and/or does not breach any intellectual property rights.

Standards in PCB design manufacture

The Association Connecting Electronics Industries (known as IPC – see www.ipc.org) is the pre-eminent industry body for reference in design, production and manufacturing of printed circuit boards in all their forms. This body, among other things, produces standards to which manufacturers refer, when consistency and quality assurance are essential.

The following page presents a 'road map' summary of the standards documents produced by the IPC. Some of these standards are referenced in sections of this book.

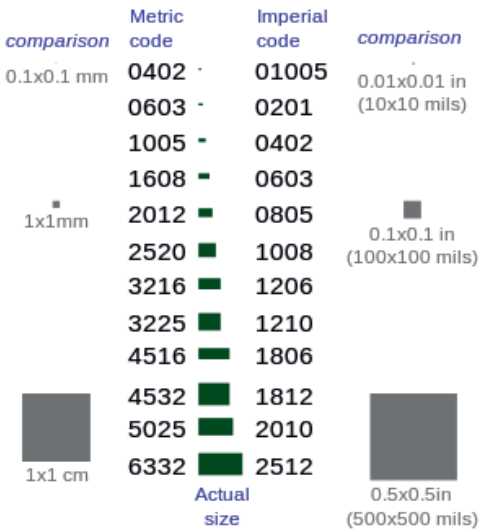
Appendix 2 ● Common Component Packaging Specifications

This appendix is intended to provide guidance to parameter packaging and dimensions of some of the more commonly available components. The contents is by no means exhaustive; rather it is simply illustrative of the extent and variety of global component stock, which exceeds many tens of thousands, and grows daily.

The specifications cited in this appendix give an indication of the dimensions of common components. Whenever the exact size needs to be known, the user should always check the manufacturer's datasheet. All measurements are in millimeters, except where otherwise stated.

SMD Resistor and Capacitor Packages

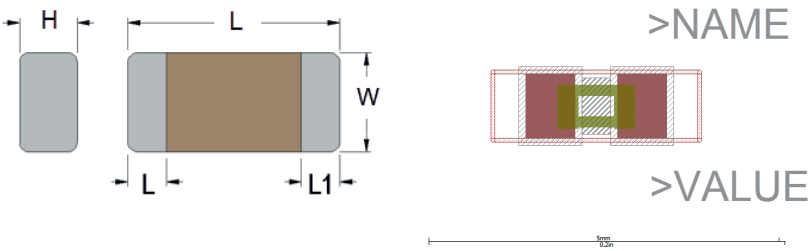
Small surface mounted resistors and capacitors share, at some level, a common set of packaging specifications. These have been defined by the Electronic Industries Alliance (EIA) and have been adopted widely in industry. Figure A2-1 provides a comparison of the sizes of these respective packages.



Source: Wikipedia

Figure A2-1 Comparison of Case Dimensions of common EIA SMD packages

The nominal packaging dimensions and the Land Pattern for EAGLE for these devices, are given in Table A3-1.



Source PCB Libraries, CadSoftComputer GmbH

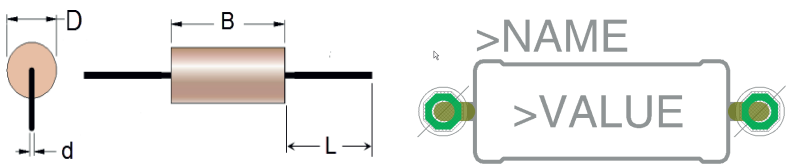
Figure A2-2 Dimensions & EAGLE Land Pattern for EIA SMD packages

Case Designator		Length (L)		Width (W)		Height (H)	
Imperial	Metric	inch	mm	inch	mm	inch	mm
0201	0603	0.024	0.6	0.012	0.3	0.01	0.25
0402	1005	0.04	1.0	0.02	0.5	0.014	0.35
0603	1608	0.06	1.55	0.03	0.85	0.018	0.45
0805	2012	0.08	2.0	0.05	1.2	0.018	0.45
1206	3216	0.12	3.2	0.06	1.6	0.022	0.55
1210	3225	0.12	3.2	0.10	2.5	0.022	0.55
1218	3246	0.12	3.2	0.18	4.6	0.022	0.55
2010	5025	0.20	5.0	0.10	2.5	0.024	0.6
2512	6332	0.25	6.3	0.12	3.2	0.024	0.6

Table A2-1 – Case dimensions for some common EIA SMD packages

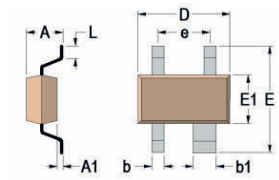
Through–hole axial resistor packages

The dimension of through-hole axial resistors is standardized as are SMD resistors packages. As a result their specifications differ with each manufacturer, power rating and type of resistor (i.e. carbon composition, wirewound, carbon or metal film).



Source: PCB Libraries Inc, CADSoft Computer GmbH

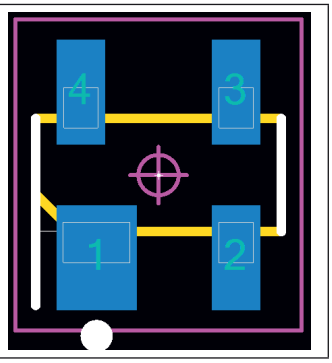
SOT143 4 PIN Transistor Outline



Pitch (e) 1.92

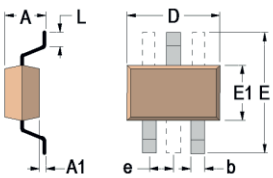
Set tab stops for: ○ Nom/Tol ◉ Min/Max

Ref.	Nom	- Tol	+ Tol	Min	Max
A					* 1.22
A1				* 0.01	
b	0.425	-0.075	0.075	* 0.35	* 0.50
b1	0.825	-0.065	0.065	* 0.76	* 0.89
D	2.92	-0.12	0.12	* 2.80	* 3.04
E	2.37	-0.27	0.27	* 2.10	* 2.64
E1	1.30	-0.10	0.10	* 1.20	* 1.40
L	0.50	-0.10	0.10	* 0.40	* 0.60



Source: PCB Libraries Inc.

SOT353 5 PIN Transistor Outline

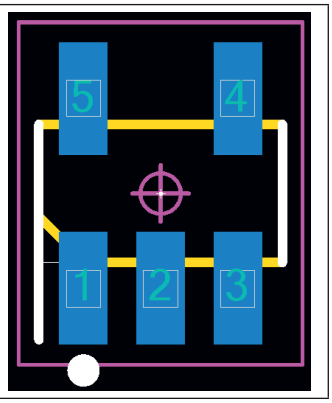


Pitch (e) 0.65

Pins 5

Set tab stops for: ○ Nom/Tol ◉ Min/Max

Ref.	Nom	- Tol	+ Tol	Min	Max
A					* 1.10
A1				* 0.00	
b	0.225	-0.075	0.075	* 0.15	* 0.30
D	2.05	-0.20	0.20	* 1.85	* 2.25
E	2.125	-0.125	0.125	* 2.00	* 2.25
E1	1.25	-0.10	0.10	* 1.15	* 1.35
L	0.335	-0.125	0.125	* 0.21	* 0.46



Source: PCB Libraries Inc.

Appendix 3 ● EAGLE User Language Reference Manual

Author's Note: Material in this appendix is taken directly from EAGLE documentation sources. This has been done in order present this information in a more convenient, accessible and integrated form.

The author acknowledges that full ownership and rights of this material reside with CadSoftComputer GmbH.

Index of Figures and Illustrations

Alphabetical Index

AN ADVANCED USER'S GUIDE

THE EAGLE COMPANION

Mitchell Duncan

Mitchell Duncan is an electrical engineer with experience in designing modem equipment, writing telecoms software and planning & management of Telco Operational Support Systems. He consults to the global investment community about technology and its impact, and writes technical documentation to help others. Mitchell lives with his wife in the Blue Mountains of NSW, Australia.

EAGLE , the "Easily Applicable Graphical Layout Editor" is a professional-grade CAD (computer aided design) software package for the design and drafting of electronic schematics as well as the design and fabrication of printed circuit boards (PCBs).

This publication, the 'EAGLE Advanced User Guide' provides the experienced EAGLE user with insight into using some of the more advanced features of EAGLE software. It is not a guide to teach the reader the basic concepts of EAGLE, nor does it discuss the 'how to' of the EAGLE interface and the simpler operations and commands of the software. That is the purpose of the author's previous title 'EAGLE V6.0 Getting Started Guide' also published by Elektor.

This book is intended as an enduring document covering the more advanced modules, commands, and functions which make up EAGLE. It is hoped that this book will sit on the desk or the bookshelf of the EAGLE user, and provide a quick, succinct reference to assist with more complex applications and uses of EAGLE — an 'EAGLE User's Companion', if you like.

Complementing the EAGLE Advanced User's Guide, the EAGLE User Language manual is included in this book in unabridged form, reproduced with permission of CadSoft GmbH.

At the time of writing, the material in this book covers version 7 of the EAGLE software suite.

ISBN 978-1-907920-42-4

9 781907 920424

Elektor International Media BV
www.elektor.com