

EAGLE

EASILY APPLICABLE GRAPHICAL LAYOUT-EDITOR

Tutorial

Version 4.1

Schematic - Layout - Autorouter
for Linux® and Windows®

CadSoft Computer Inc.
www.cadsoftusa.com

2nd Edition

Copyright © 2004 CadSoft

All Rights Reserved

If you have any questions please feel free to contact us:

USA and other countries:

Phone: +1 (561) 274 8355, USA also: 1-800-858-8355
Fax: +1 (561) 274 8218
Internet: www.cadsoftusa.com
Email: Info@cadsoftusa.com

Germany and other European countries:

Phone: +49 (0)8635 6989-10
Hotline: +49 (0)8635 6989-30
Fax: +49 (0)8635 6989-40
Internet: www.cadsoft.de
Email: Info@cadsoft.de

And remember that we offer a free hotline for our customers!

© Copyright 2004 CadSoft Computer, Inc. All rights reserved worldwide
No part of this publication may be reproduced, stored in a retrieval system, or transmitted, in any form or by any means, electronic, mechanical, photocopying, recording, scanning, digitizing, or otherwise, without the prior consent of CadSoft.

Windows is a registered trademark of Microsoft Corporation.
Linux is a registered trademark of Linus Torvalds.

Table of Contents

1 What to expect from this Manual	7
2 Features of EAGLE	8
System Requirements	8
Professional Version	8
General	8
Layout Editor	9
Schematic Module	9
Autorouter Module	9
Standard Edition	10
Light Edition (Freeware)	10
3 Installation and Program Start	11
Windows	11
Linux	11
4 Individual EAGLE Setup	12
The Script File eagle.scr	12
User Interface	12
Function Keys	12
Layer Colors	12
5 The Concept of the EAGLE User Interface	13
Selecting Menu Items	13
Mouse Click	13
Several Input Alternatives	14
Use of Key Combinations	14
Command and Parameter Input via the Command Line	14
6 Control Panel	17
EAGLE Files	18
Backup Files	18
Create EAGLE Projects	18
7 Load File and Select Monitor Zoom	20
8 Selecting Layers for Display	22
9 Setting up Grid and Unit	23
10 Wires, Circles, Arcs, Rectangles, and Text	24

The WIRE Command	24
Changing Line Width	25
Change Object to another Layer	26
Undo/Redo Function	26
The CIRCLE Command	26
The ARC Command	27
The RECT Command	27
The TEXT Command	28
Special Text Variables	29
11 Using Libraries	30
The ADD Command	30
The USE Command	32
The INVOKE Command	32
12 Drawing a Schematic	34
Grid	34
Adding a Frame to a Schematic	34
Adding and Changing Text	34
Entering a Schematic	35
The NET Command	37
The NAME Command	37
The LABEL Command	37
The DELETE Command	37
The JUNCTION Command	39
The SHOW Command	39
The MOVE Command	40
History Function	40
Completing the Schematic	40
The SMASH Command	41
The VALUE Command	41
The Electrical Rule Check (ERC)	42
Generating a Board from a Schematic	42
The BUS Command	42
13 Automatic Forward&Back Annotation	44

14 Designing a PC Board	45
Designing a Board without a Schematic	45
Defining Board Shape	45
Placement Grid	46
Placing Components	46
Placing SMD Packages	46
Providing Names	47
Providing Values	47
Defining Signals	47
Defining Signal Classes	48
Creating a Board from a Schematic	49
Generating a Board File	49
Component Placement	49
Autorouter: A Brief Example	50
Routing Manually	50
Board Changes	51
Further Usage of the Layout Editor	52
The DISPLAY Command	52
The MOVE Command	52
The GROUP Command	53
The SPLIT Command	53
The CHANGE Command	53
The ROUTE Command	54
The RIPUP Command	54
The SHOW Command	55
Refresh Screen	55
Undo/Redo Function	55
Inner Layers	55
Supply Layers	55
Copper Pouring with the POLYGON Command	56
15 Autorouter	58
16 Design Rule Check	60
17 Libraries	61

Resistor Package	61
Resistor Symbol	63
Resistor Device	63
18 Output of Drawings and Manufacturing Data	67
Output a Schematic with the PRINT Command	67
Generating Image Files for Documentation Purpose	68
Generating Gerber Data with the CAM Processor	68
Generating Drill Data	69
Further Manufacturing Data	69
19 Data Exchange	70
The EAGLE User Language	70
Script Files — A Flexible Input Interface	70

1 What to expect from this Manual

This tutorial provides a basic introduction to the EAGLE PCB-Design Package.

It covers the use of the EAGLE Schematic Editor, Layout Editor, and Autorouter. 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 autorouting. You will benefit most by going through the entire document.

You should be familiar with the use of the basic functions of your operating system. Expressions like *enlarge the editor window* will be used without further explanation.

Having completed this tutorial you should be able to start working on a serious project. While creating your initial designs, however, you should frequently use the help function and the EAGLE Reference Manual to learn more about specific details. Only then will you be able to take full advantage of EAGLE's capabilities.

You will learn how to use most of the program commands, although not all of the features which make EAGLE so powerful and flexible are discussed in this introduction — for example the possibilities of the SET, SCRIPT, and RUN commands (see help).

Before you begin you should consult the README file and the files with the extension **.txt* in *eagle/doc*.

Although this tutorial is based on the Windows version of EAGLE, the differences to Linux are minimal.

2 Features of EAGLE

System Requirements

EAGLE is a powerful graphics editor for designing PC-board layouts and schematics. In order to run EAGLE the following hardware is required:

- IBM-compatible computer (586 and above) with
- Windows 95/98/ME, Windows NT4/2000/XP or
- Linux based on kernel 2.x, libc6 and X11 with a minimum color depth of 8 bpp,
- a harddisk with a minimum of 50 Mbyte free memory,
- a minimum graphics resolution of 1024 x 768 pixels, and
- preferably a 3-button mouse.

Professional Version

General

- maximum drawing area 64 x 64 inches (about 1600 x 1600 mm)
- resolution 1/10.000 mm (0.1 microns)
- mm or inch grid
- up to 255 layers, user definable colors
- command files (Script files)
- C-like User Language for data import and export
- simple library editing
- composition of user-defined libraries with already existing elements by Drag&Drop
- easy generation of new package variants from any library by Drag&Drop
- free rotation of package variants (0.1 degree steps)
- library browser with powerful search function
- support of technology feature (e.g. 74L00, 74LS00..)
- generation of graphics output as well as manufacturing and testing output with the CAM processor or the help the User Language
- printouts via the OS's printer drivers
- user-definable, free programmable User Language to generate data for mounting machines, test equipments, milling machines or any other data format
- partlist generation with database support (*bom.ulp*)

- Drag&Drop in the Control Panel
- automatic backup function

Layout Editor

- full SMD support
- support of blind and buried vias
- rotation of elements in arbitrary angles (0.1-degree steps)
- texts can be placed in any orientation
- dynamic calculation of signal lines while routing the layout
- tracks can be layed out with rounded corners in any radius
- mitering to smooth wire joints
- Design Rule Check for board layouts (checks e.g. overlaps, measures of pads or tracks)
- copper pouring (ground plains)
- package variants support

Schematic Module

- up to 99 sheets per schematic
- simple copying of parts
- Online-Forward&Back Annotation between schematic and board
- automatic board generation
- automatic generation of supply signals
- Electrical Rule Check (error check in the schematic and consistency check between schematic and layout)

Autrouter Module

- fully integrated into basic program
- uses the layout's Design Rules
- change between manual and automatic routing at any time
- ripup&retry algorithm
- user-definable strategy by cost factors
- routing grid down to 0.02 mm (about 0.8 mil)
- no placement restrictions
- up to 16 signal layers (with user definable preferred directions)
- up to 14 supply layers
- full support of blind and buried vias
- takes into consideration various signal classes

Standard Edition

The following restrictions apply to the Standard Edition in the Layout Editor:

- 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.
- A maximum number of 4 signal layers are allowed (top, bottom, and 2 inner layers).

Light Edition (Freeware)

The following restrictions apply to the EAGLE Light Version, which is available as Freeware (for testing and evaluation):

- The board area is restricted to 100 x 80 mm (about 3.9 x 3.2 inches). Outside this area it is not possible to place packages and draw signals.
- Only two signal layers can be used (no inner layers).
- A schematic can consist of only one single sheet.

Larger layouts and schematics can be printed with the *smaller* editions. The CAM processor can generate manufacturing data as well.

3 Installation and Program Start

Windows

Insert the media into the CD-ROM drive. Select the desired menu item directly in the CD-ROM start window.

If the start window does not automatically appear, double-click on the CD-ROM symbol in *My Computer*.

Follow the instructions on the screen.

For the Freeware installation you do not need a *User License Certificate*. Answer the question for a valid license by clicking *Run as freeware*.

If you decide to uninstall EAGLE, use the *unInstallShield* program which will be installed along with the EAGLE program.

The EAGLE CD-ROM supplies a playable Freeware. You can start it without installing it on your harddisk. But there are some minor restrictions due to the fact, that EAGLE can't write files on the CD-ROM.

Linux

Insert the CD and mount the CD-ROM drive.

Choose the corresponding directory (*/english/linux/install*) and read the installation notes in the *README* file. While installing the program you will be asked if you want to run EAGLE as Freeware or as a licensed version. Choose *Run as freeware*, if you don't have a valid license.

The EAGLE CD-ROM supplies a playable Freeware. You can run it from CD-ROM directly. Therefor you have to mount the CD-ROM drive as *executable*. But there are some minor restrictions due to the fact, that EAGLE can't write files on the CD-ROM.

4 Individual EAGLE Setup

Apart from the basic installation, EAGLE allows the user to customize certain program features, such as the configuration of menus, function keys, or screen colors. A lot of these settings can be made in the *Options* menu in the Control Panel or in one of the editor windows.

The Script File *eagle.scr*

In the special command file (script file) *eagle.scr* preset values for the Schematic, Layout, and Library Editors can be entered in the form of EAGLE commands. Those who would like to use these possibilities should get acquainted with the EAGLE command language. The syntax of each EAGLE command is described in the EAGLE help.

User Interface

The user interface can be set individually. Click the *Options/User interface* menu in the Control Panel or in one of the Editor windows. The tutorial presupposes that you are using the default settings.

Function Keys

Several function keys are predefined with different commands. This layout can be changed by the user at any time. However, operating system specific keys (like F1 for the help function in Windows) must not be redefined. The current function keys layout can be found in the menu *Options/Assign*.

Layer Colors

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

Additional information concerning configuration can be found in the help function. See the items *SET*, *ASSIGN*, *User Interface*, *CHANGE*, and *Project*.

5 The Concept of the EAGLE User Interface

Internally, EAGLE has been set up in such a way, that any action is initiated by a command string. Normally the user activates these commands by clicking on menu items or toolbar icons. Values are normally entered into appropriate fields.

The knowledge of the internal command language is not necessary to successfully design schematics and boards with EAGLE. However, this concept offers further possibilities which make EAGLE a very flexible tool: Any command, for instance, can be entered in text format via the command line or can be read from a file. Furthermore, command strings can be assigned individually to function keys (ASSIGN command). This enables the user e.g. to execute command sequences with a key stroke or a few mouse clicks (see SCRIPT command).

This tutorial uses a simplified notation for various actions in EAGLE which is explained in the following examples.

Selecting Menu Items

The character \Rightarrow means, that a menu selection is to be made. For example
 \Rightarrow *File/Save*

means: click the *File* menu with the left mouse button and next click *Save*.

Mouse Click

Actions to be carried out with a click of the left mouse button are represented with a dot. For example:

- MOVE and F1

means: click the MOVE command with the left mouse button and then press the function key F1.

Actions to be carried out with a double click of the left mouse button are represented with two dots. For example

- linear.lbr


means: select *linear.lbr* with a double click of the left mouse button from the menu.

Some commands have special functions in combination with the *Shift*, *Ctrl*, and *Alt* key. Please see detailed information in the command reference of the help function.

Several Input Alternatives

EAGLE commands can be entered via keyboard, by clicking icons or by clicking menu items.

The following actions, for example, will execute the MOVE command:

- Clicking the icon 
- Typing MOVE in the command line, followed by the *Enter* key
- Pressing the function key F7 which is assigned to the MOVE command
- Selecting the menu item \Rightarrow *Edit/Move*

In this tutorial we will mainly work with the toolbars. For the sake of clarity the commands are shown as text:

- MOVE

means: click the MOVE icon 

Use of Key Combinations

A + character indicates that the first key is held down while pressing the second key. For example:

Alt+F2

The *Alt* key is held down while pressing F2, then release both keys.

Command and Parameter Input via the Command Line

Actions which need to be terminated with the *Enter* (i.e. return) key are symbolized with the character \leftarrow . For example

USE \leftarrow

means: type USE and next press the *Enter* key.

Anything that is to be typed exactly as it appears, will appear in the text as follows:

CHANGE WIDTH 0.024 \leftarrow

Normally EAGLE does not differentiate between upper and lower case characters. Therefore you can enter the above command as

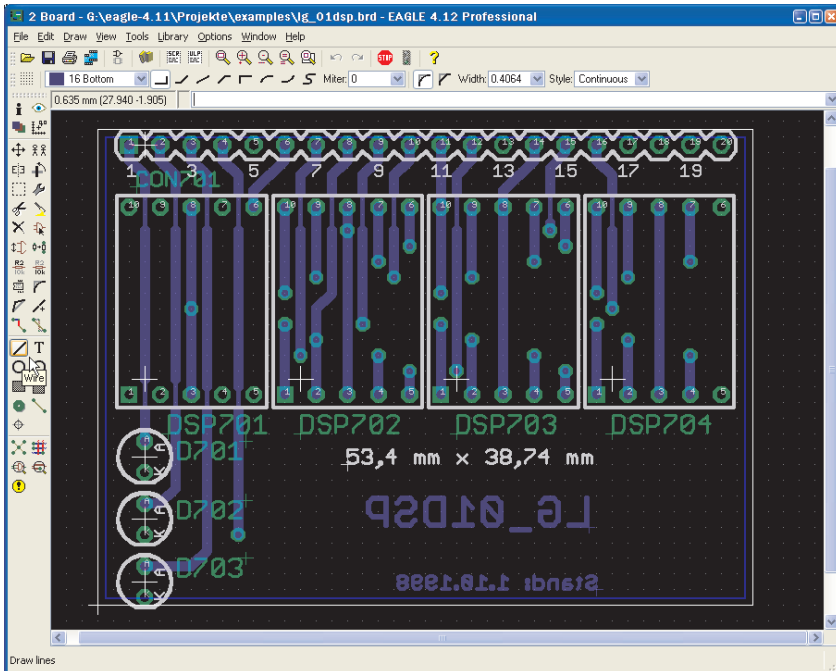
change width 0.024 \leftarrow

You may abbreviate the key words. The above input may therefore be simplified to

```
cha wid 0.024 ←
```

In this tutorial, however, the full commands are used.

The following figures show which commands are activated with the various toolbar icons. Additional help is offered by the Bubble Help text which appears as soon as the mouse cursor is positioned on an icon for a certain time. This text shows the command name.



The Layout Editor window

From top to bottom: title with information about EAGLE version and edition, menu bar, action toolbar, dynamic parameter toolbar and coordinates display with command line.

On the left the command toolbar. The Bubble Help text describes the WIRE icon. The status bar below shows a short description of the current command.

The toolbars can be displayed/hidden in the *Options/User interface* menu. Additionally one has the possibility to use a text menu instead of the shown command toolbar.

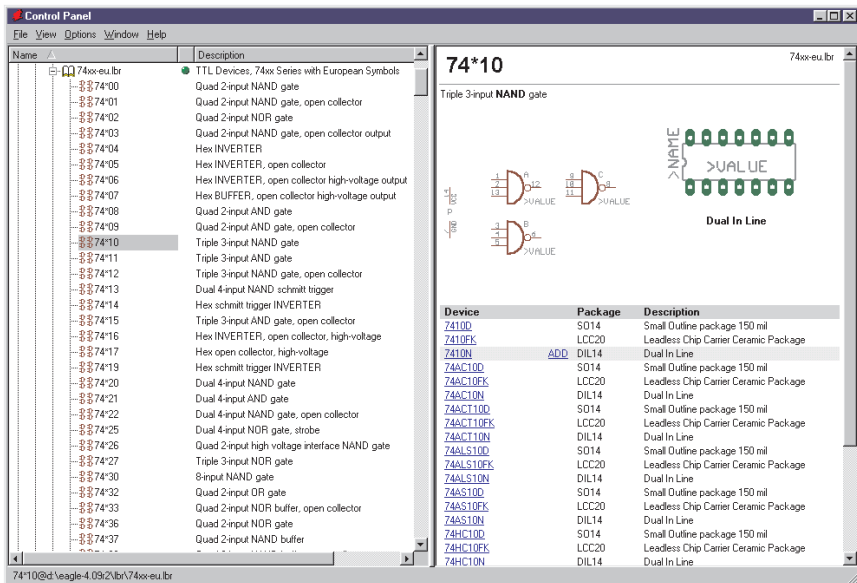
Info		Show	Info		Show
Display		Mark	Display		Mark
Move		Copy	Move		Mirror
Mirror		Rotate	Mirror		Rotate
Group		Change	Group		Change
Cut		Paste	Cut		Paste
Delete		Add	Delete		Add
Pinswap		Gateswap	Pinswap		Replace
Name		Value	Name		Value
Smash		Miter	Smash		Miter
Split		Invoke	Split		Optimize
Wire		Text	Wire		Text
Circle		Arc	Circle		Arc
Rectangle		Polygon	Rectangle		Polygon
Bus		Net	Via		Signal
Junction		Label	Hole		
ERC			Ratsnest		Auto
			ERC		DRC
			Errors		

Command toolbar of the Schematic Editor (left) and the Layout Editor (right)

6 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.

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



Control Panel: Preview of the library contents

The Control Panel offers also an overview of User Language programs, Script files, and CAM jobs. Try selecting various entries. On the right you will get the referring description.

The Control Panel supports Drag&Drop in usual manner. A right mouse click on any entry in the tree view opens a context menu that offers options like print, open, copy, etc.

The paths for each branch of the tree view are set in *Options/Directories*.

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	*.*

The Linux version only recognizes lower case letter file extensions!

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.

All settings concerning backups can be done in the *Options/Backup* menu of the Control Panel.

Create EAGLE Projects

Lets create a new project first. After starting the program, first • the + character of the *Projects* path, then • the + character of the entries *examples* and *tutorial* in the tree view. The contents of the *tutorial* directory appears. • *tutorial* with the right mouse button. Select the option *New Project* in the popup menu. Name the new project *MyProject*, for example and hit the *Enter* key. This way you are creating a subdirectory of *tutorial* that is named *MyProject*. This directory should contain all data files that belong to your project. Of course you may define additional subdirectories.

To define the path where your project directories will be stored, click ⇒ *Options/Directories* and enter it in the *Projects* field.

A right mouse click on the project entry and you can open new schematics, layouts and libraries. Each project directory contains a file named *eagle.epf* which stores project-specific settings, window positions etc.

The currently active project is checked (green) in the Control Panel. After starting the program again the previous situation will be restored. The last used project and other user-specific settings are saved in the file `~/.eaglerc` (Linux) or `eaglerc.usr` (Windows).

Before starting the following examples we want to copy the files *demo1.sch*, *demo2.sch*, and *demo2.brd* into the directory *MyProject*.

Press the *Ctrl* key, click the desired file and drag it to the *tutorial* entry. Release the mouse button now. Repeat this for the other files.

The *Ctrl* key effectuates that the files will be copied, otherwise they would be moved to the target directory.

Now open the schematic file *demo1.sch* with a double click.

If you end the program with *Alt+X* and start it again, you will get the previous settings and editor windows.


7 Load File and Select Monitor Zoom

Now let us start doing some exercises. Start EAGLE, and wait until the Control Panel appears.


Expand the entry *Projects/examples/tutorial/MyProject* of the tree view.


Now load the *demo2.brd* file. You can do this either by •• the entry *demo2.brd*, or by selecting the file from the menu \Rightarrow *File/Open/Board*. The schematic with the same name will be loaded along with the board.


Enlarge the board editor window.

First click the icon  in the action toolbar to zoom into the drawing. This WINDOW command can also be executed by pressing the function key F3. If you are working with a wheel mouse, you can zoom into or out of the drawing by turning the mouse wheel.

The setting for *Mouse wheel zoom = 0* in the *Options/User Interface* menu deactivates this feature. The zoom factor can be set with the option *Mouse wheel zoom*.

Click this icon  or press F4 to zoom out of the drawing.

By clicking this icon  the drawing will be shown in full size to fit your screen (you can also use Alt+F2 instead).

Click this icon  and then mark a rectangular area by dragging the mouse cursor while the left mouse button is pressed. Then release the mouse button. The marked area will now be displayed.

To move the chosen window simply click the middle mouse button and move the mouse. This also works while an editor command is active, for example, while drawing nets or wires.

To scroll beyond the drawing borders additionally press the Shift key.


If the cursor is over the vertical or horizontal scroll bar and you are working with a wheel mouse, you can move the screen up and down or to the right or left by turning the mouse wheel.

During certain actions it may happen that objects in the drawing disappear or get corrupted. In this case refresh the screen by clicking the Redraw icon



(also F2 possible).

The WINDOW command is more versatile than in other programs:

Click this icon  if you want to select a new center with the same zoom factor, mark the center with a click and finally click on the traffic light icon in the action toolbar.

If you want to select a new center and a new zoom factor simultaneously, click on the same icon. Three mouse clicks will give you the desired result: the first click will define the new center and both last clicks will define the zoom factor. If the third point is further away from the first, the program will zoom into the drawing and vice versa. Try it to find out how it works.

Further possibilities can be found on the help pages of the WINDOW command. These can be called up by simply typing in the command line:

HELP WINDOW ←

8 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 Via layers is used to generate a film for etching the component side of the printed-circuit board. Consequently the combination of Bottom, Pad, and Via 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 via 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* 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.

By selecting/deselecting layer 21 *tPlace* (silk screen upper side), the layers 23 *tOrigins*, 25 *tNames*, 27 *tValues*, and 51 *tDocu* are selected/deselected, too. The same applies to layer 22 *bPlace* (silk screen bottom side).

Very important: Components on layer 1 *Top* can only be moved or selected in the drawing if layer 23 *tOrigins* is on. The same applies to components on layer 16 *Bottom* and the layer 24 *bOrigins*.


To select a certain layer in the DISPLAY menu click on the layer name. Now you can use the *Change* button to modify the layer's properties like name, color, or fill style.

Please consult the help page of the LAYER command for the meaning of the different EAGLE layers.

9 Setting up Grid and Unit

Schematics should always be drawn on a grid of 0.1 inches (2,54 mm) since the libraries are defined 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. Please consult the help pages of the GRID command for detailed information.

For all settings in the Design Rules window (\Rightarrow *Edit/Design Rules...*) one can use values in mil or in millimeters (1 mil = 1/1000 inch). The default unit is mil.

If you prefer to work with millimeters simply add the unit to the value, for example:

0.2mm

Inch - Mil - Millimeter Table for the Most Usual Values:

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

The GRID dialog allows setting an alternative grid which can be activated by pressing the *Alt* key in the Editor window.

10 Wires, Circles, Arcs, Rectangles, and Text

Wires, circles, arcs, rectangles, and text are created with the WIRE, CIRCLE, ARC, RECTANGLE and TEXT commands. On one hand these objects serve as pure drawing elements for symbols, packages, frames etc., and on the other hand they can perform special functions, such as the definition of restricted areas.

First a new schematic file is to be created. Close all of the editor windows and select

⇒ *File/New/Schematic*

from the Control Panel.

A new file with the name *untitled.sch* is now created. Normally you should never save a file with the name *untitled*, but should use ⇒ *File/Save as* to choose a different name. However, in this tutorial no file is to be saved at all.

Now enlarge the editor window.

The WIRE Command

The WIRE command is used to draw lines.

Click the WIRE command in the command toolbar. All parameters for this command can be set up in the parameter toolbar. Next select layer 94, *Symbols*, from the layer-selection combo box. In this layer a rectangular line is to be drawn.



Parameters of the WIRE command

Define the starting point by a click on the left hand mouse button. Move the cursor slightly up to the right and press the right hand mouse button a few times. This way one changes the bend mode (wire bend). Among some diagonal and orthogonal modes you will find some to draw 90°-arcs and free definable arcs.

However, in the Schematic Editor the arc modes can be selected only in the parameter toolbar, not with the right mouse button.

Keeping the *Shift* key pressed while clicking the right mouse button will change the selection order. Pressing the *Ctrl* key toggles between corresponding wire bends.

When the connection is displayed in a square angle, press the left hand button to fix its position. Now move the cursor to the starting point and ●● to drop the line. Now you should be able to see a rectangular outline. As

observed before, an angle between wire segments can be created by clicking the right hand mouse button. This is more effective than using the symbols in the parameter toolbar.

Attention: Do not use the WIRE command to draw net or bus lines in schematics - use NET or BUS instead!

In the Layout Editor:

If the lines (WIRE command) are placed on the board layers Top, Bottom, or Route2..15 EAGLE treats them as electrically conducting tracks. Wires are also used to create board outlines. Let's start using this command.

Changing Line Width

As long as the WIRE command is active, you can select the line width from the combo box in the parameter toolbar or type in a specific value, separate for each segment.

To change the line width of an existing object,

- **CHANGE** icon in the command toolbar and a popup menu will open up.
- **WIDTH** entry and a further popup menu will show up where the present value is marked.

Select the desired value by a left mouse click, then click the object to be changed.

To change the line width to a value that is not shown in the menu of the **CHANGE** command, click the entry . . . and type in the value in the *Change Width* window.

Alternatively use the command line to type in the value, for example:

```
CHANGE WIDTH 0.017 ←
```

Then click on the wire segment you want to change or start drawing a new wire.

To change the wire style • **CHANGE** and • *Style*. Select the style and • the wire you want to change.

Change Object to another Layer

To move an object, for example a wire segment, to another layer

- CHANGE
- LAYER

Select the target layer, for example 94 *Symbols*, by •. Then • OK, and then • on the selected object(s). Note that some objects, such as bus or net lines, cannot be moved to another layer as they have a special meaning.

Undo/Redo Function

One of the most useful features of EAGLE is the unlimited Undo function. Click the left icon as many times as you want to undo previous actions. Use the right icon to redo the actions which have been cancelled by undo.



The CIRCLE Command

To activate CIRCLE, which is used to draw a circle,

- CIRCLE

EAGLE requires two mouse clicks to define a circle. The first click sets the center of the circle and the second click defines the radius.

Place the cursor at any grid point and •. Drag the cursor several grid points to the right. When the circle has the diameter you want, • to fix it and terminate the command. The line width of the circle can be changed as described before for wires. A circle with line width 0 will be filled.

Example for drawing a circle using coordinate values:

A circle with the origin at position $x = 10$ and $y = 25$ and a radius of 15mm should be drawn.

First set the grid to millimeter:

GRID MM ←

Draw the circle now:

CIRCLE (10 25) (10 40) ←

or

CIRCLE (10 25) (10 10) ←

The second pair of coordinates describes any location on the

circumference. So various values are possible to describe one certain circle. To find out more about the CIRCLE command press F1 as long as the command is activated or type

HELP CIRCLE ←.

To cancel a command, click the stop sign icon or activate another command. Pressing the *Esc* key generally unlocks an object from the cursor.

The ARC Command

To activate the ARC command, which is used for drawing arcs,

- ARC

An arc is defined with three mouse clicks: the first click defines the start point, the second the diameter and the third the end point.

Place the cursor at the desired starting point and •. Now move the cursor some grid units to the right but remain on the same Y-coordinate. A circle appears which shows the diameter of the arc. • and the circle will become an arc. Now you can change the direction of the arc with the right mouse button. Click several times with the right button and you will see what is meant. You can also enlarge or minimize the arc by moving the mouse. After reaching the desired form, • to fix the arc.

The parameters *flat* and *round* determine the shape of the arc's ends.

Practice by drawing some arcs. Use the help function to find out more about the ARC command.

All this can be done with the WIRE command as well!

The RECT Command

To activate the RECT command, used for creating filled rectangles,

- RECT

To define a rectangle two mouse clicks are required: The first one will determine one corner and the second determines the position of the opposite corner.

Move the cursor to the point where a corner of the rectangle should be and •. Move the cursor slightly to the right and up. When the rectangle has reached the desired size, • to fix it. The rectangle is filled with the color of the layer in use.

Use the help function to find out more about the RECT command.

The TEXT Command

To activate the TEXT command, used for placing text,

- TEXT

Now type the desired text and • OK. Then place the text with •. A copy of the same text is now attached to the cursor. To stop placing text simply click the next command icon. For placing a different text, type the text and terminate it with the *Enter* key. The text will show up in the command line.

Texts containing spaces or a semicolon have to be enclosed in single quotes, like this one:

'This is a text'

To change the text font:

- CHANGE
- FONT

EAGLE supports a vector, a proportional, and a fixed font.

To change the size of a text:

- CHANGE
- SIZE
- Value in the menu

or type in any desired value in the command line (confirm with the *Enter* key) and • lower left corner of the text. At a rotated text the point of origin can move to its upper right corner. A text in a schematic is always displayed in a way that it can be read from the front or from the right.

The Layout Editor allows to display texts in any orientation. Use the *Spin* flag which is located in the parameter toolbar while the TEXT, MOVE, or ROTATE command is active to get texts readable from all directions.

To change a text

- CHANGE
- TEXT

and • at the point of origin of the text, then edit the text and • on OK.

Using

- CHANGE
- RATIO

you can change the line width in a text in relation to the height of the vector font.

See help page for more information about TEXT and CHANGE.

Special Text Variables

If you place the text

`>SHEET`

this string will be substituted with the current sheet number, e.g. *1/1* (sheet one of totally 1).

EAGLE offers a number of similar text variables, e.g. for date/time which reflect the latest change in the file (`>LAST_DATE_TIME`) or the drawing output (`>PLOT_DATE_TIME`).

Library parts are defined with text variables for the name `>NAME` and the value `>VALUE` of a component. Furthermore one can use `>PART` and `>GATE` for symbols.

11 Using Libraries

EAGLE comes with a lot of library files that contain through-hole and surface mount devices. The tree view in EAGLE's Control Panel offers detailed information about the contents of the libraries.

In this section you will learn how to insert schematic symbols into a drawing and how to use them.

Open a new schematic to start with a blank drawing area

⇒ *File/New/Schematic*.

The ADD Command

To select symbols from a library, • ADD in the command toolbar, and a window pops up. On the left side a list of available libraries should appear. Each library entry can be opened by clicking the + character. Now its contents will be shown. Select an entry and you will see the corresponding preview on the right.

Now you can enter on or more search patterns in the *Search* field. You may use the name of a device or any word of the device description. Wild cards like * and ? are allowed.

If there are no entries visible after using ADD for the first time, no libraries have been loaded. In this case, please read the following chapter dealing with *the USE command*.

We want to place, for example, the device 74LS00. Enter in the *Search* field:

74*00* or 74LS00*

* is the wild card of the technology and/or the package variant. The search result shows the device in various technologies and package variants. Select the desired device and • OK. Now you can place it in the schematic.

Place the cursor slightly to the left of the display center and •. Move the cursor to the right, and place a second gate with the next mouse click. Place four gates around the center of the drawing area in this way.

Now place a fifth gate somewhere nearby. Please note that EAGLE has named the first four gates IC1A..IC1D, whereas the fifth gate has been named IC2A, since this gate requires a second IC.

If you now show the layer 93, *Pins*, either as described before or by typing

DISPLAY PINS ←

in the command line, further pin parameters are displayed in green. Zoom in on the drawing, so that a gate is shown on a large scale. You will see that the pins are marked as Input (In) or Output (Out), and that a number

shows the Swaplevel.

A Swaplevel greater than 0 indicates that this pin can be swapped with another pin of the same device which has the same Swaplevel assignment (see command PINSWAP). A pin with a swaplevel of 1, for example, can be swapped with any other pin that has a swaplevel of 1. Swaplevel 0 means that this pin cannot be swapped.

The layer 93, Pins, is not usually printed (PRINT command).

As long as the ADD command is still active, a gate symbol will be attached to the cursor.

Now use the Zoom-in icon or the *F4* key to view a larger portion of the schematic on the display. Then press the *Esc* key to the selection window of the ADD command.

Enter the following pattern in the *Search* field:

555N or *555*

Select, for example, the device LM555N from *linear.lbr* with ••, rotate it 180 degrees with two right mouse clicks, and place it somewhere on the drawing area with the left mouse button.

Repeat this with other symbols. You will find out that the libraries contain symbols drawn in the European and the American way. Choose whatever you prefer.

While the ADD command is active, you can return to the ADD menu by pressing the *Esc* key. Press the *Esc* key again and the command will be cancelled.

Another way to place devices in the schematic is to drag them from the tree view in the Control Panel into the Schematic Editor window.

Arrange the windows in a way that you can see both on the screen. Select, for example, the device LM555N from *linear.lbr* in the tree view (*Libraries* branch). Use Drag&Drop to move the device into the Schematic Editor. If you select a device that supports more than one package or technology variant, you will be asked to select the variant in a menu before dropping it.

EAGLE, by default, assumes that all active components will be attached to the same power source and ground. The power pins are therefore not shown, and are automatically connected to the Power Source and Ground when generating a board (unless the user connects them to other signals). Use the INVOKE command in case you want to place it in the schematic.

Most of the EAGLE library devices, which have only one VCC and one GND pin, are defined so that the power pins, by default, are not visible. In some cases it makes sense to make the power pins in an IC visible, as in the 555N stored in the linear library. In such a case connect the power pins with the

appropriate nets.

The help function in EAGLE offers information about further options of the commands ADD and UPDATE concerning the update of library objects in schematic and layout with their respective parts of the current libraries.

The USE Command

The default setting causes the ADD command to search in all libraries that are available in the given libraries directories (\Rightarrow *Options/Directories/Libraries* in the Control Panel). You can exclude libraries from the search function by clicking the green marker in the Control Panel's tree view, *Libraries* branch. Green means in use, gray not used. This is exactly the function of the USE command you can also type on the command line.

For example, the command

```
USE *
```

makes available all libraries that can be found in the given library paths. More about this can be found in the help function.

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

- INVOKE

and left click on the gate IC2A. A popup menu appears.

- • *PWRN* and the power pins for IC2 are attached to the cursor. You can now place them anywhere with a • and connect them to any net.

Another feature of the INVOKE command allows you to alter the sequence of the reference designators before EAGLE automatically makes an assignment. Assuming the INVOKE command is still active, • IC2A, and the popup menu appears. The asterisk assigned to gate A indicates that the gate has been used; those without an asterisk are available for use.

If you want IC2C to be placed before IC2B, • • C in the popup menu. The menu closes, and IC2C is attached to the cursor to be placed with a •. Once IC2C is placed, EAGLE will use up the remaining gates in that package before assigning an additional package.

If you want to place gates over more than one sheet, use the INVOKE command on the new sheet and type in the element's name in the command line. Now the invoke menu pops up.

Don't hesitate to experiment with different libraries and with placing and rotating schematic symbols.

You can place devices in a drawing from as many libraries as you want. Devices are saved in the schematic or board files in their entirety. When passing on a file, there is no need to supply the libraries with them.

12 Drawing a Schematic

In this section you will learn how nets and buses are used in a drawing. You will then be able to create a schematic.

To create an empty schematic, open a new drawing and enlarge the editor window.

Grid

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 adjust, for example, labels in a finer grid which will be activated by pressing the *Alt* key.

Adding a Frame to a Schematic

As a start, select a drawing frame from the library *frames.lbr*, which contains predefined frames in miscellaneous formats.

- ADD, and enter the word *letter* or *frame* in the *search* field. Select a suitable frame and •• for example `LETTER_P`. A frame which fits on a letter format page (portrait) is now attached to the cursor.

If you cannot see it completely, press function key F4 until it matches your screen, then place it with a click of the left hand mouse button so that its lower left corner is placed on the coordinates ($X=0$, $Y=0$).

Now a further frame is attached to the cursor. Click the icon with the stop sign to terminate the ADD command. Press *Alt+F2* to show the frame in full size or click the *Zoom-to-fit* icon in the action toolbar.

Adding and Changing Text

You can add lines, text and other objects to predefined frames and text fields in the library. Or you can design and save your own frames.

Variable texts, e.g. the project title or the revision number, can be inserted directly in the Schematic Editor where you are now.

Frames are saved as symbols in the library, therefore it makes sense to write the text in layer 94, *Symbols*.

Now bring the frame text field into the editor window so that it is completely visible. Next click the icon for the *TEXT* command and enter the following text

CadSoft

After clicking the OK button, the text is attached to the cursor and can be placed with the left mouse button. Move the text in the upper empty line of the text field and place it with a •. A further copy of the text, which will disappear as soon as another command is activated or the stop sign icon is clicked, is still attached to the cursor.

If you did not define the size of the text while the TEXT command was active, you can use the CHANGE command to set it to another value:

- CHANGE

From the menu select:

- SIZE

and a further window opens in which the presently selected text height is shown.

- 0.15

and move the cursor to the lower left corner of the text CadSoft. Click the left mouse button and the text height will be changed to 0.15 inches. Just in case you would like to set a size not present in the CHANGE SIZE menu, like 0.17, simply type:

CHANGE SIZE 0.17 ←

and then click the lower left corner of the text.

Attention: Use dots for decimals! The current grid setting determines the unit!

Practice manipulating texts by adding an address or a document number in the text field.

TITLE: contains the file name in use (text variable >DRAWING_NAME).

DATE: contains the date of the most recent save command (text variable >LAST_DATE_TIME).

Both fields are automatically filled with the actual data when the drawing is saved, since the frames stored in the *frames* library have been defined with the appropriate text variables.

Entering a Schematic

Now let's start drawing a schematic. We will be drawing the schematic shown in the following figure, which you can use for reference. If you don't want to enter the whole schematic you can use the file *demo1.sch* stored in the *eagle/examples/tutorial* directory.

Start by pressing ALT+F2 or clicking the Zoom-to-fit icon to fill the window with the drawing frame.

The schematic consists of the following elements:

Part	Value	Device	Package	Library	Sheet
C1	30p	C-EUC1206	C1206	rcl	1
C2	30p	C-EUC1206	C1206	rcl	1
C3	10n	C-EU025-025X050	C025-025X050	rcl	1
C4	47u/25V	CPOL-EUTAP5-45	TAP5-45	rcl	1
C5	47u	CPOL-EUTAP5-45	TAP5-45	rcl	1
D1	1N4148	1N4148	DO35-10	diode	1
IC1	PIC16F84AP	PIC16F84AP	DIL18	microchip	1
IC2	78L05Z	78L05Z	TO92	linear	1
JP1	PROG	PINHD-1X4	1X04	pinhead	1
JP2	APPL	PINHD-1X17	1X17	pinhead	1
Q1		XTAL/S	QS	special	1
R1	2,2k	R-EU_R1206	R1206	rcl	1
F1		DINA4_L		frames	1

Use the ADD command to place the listed devices.

Please keep in mind:

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

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.

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, then move the cursor to the part you want to move and **•**. EAGLE will highlight the part, to let you know that it is attached to the cursor and ready to be relocated.

Relocate the part, and **•** 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.

For duplicating parts you may use the COPY command (for example, C1 and C2). Thus you don't have to fetch each part with the ADD command.

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

Attention: Use the NET command, not WIRE!

The NET Command

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). In our example *demo1.sch* the net lines at C5 pin +, U1 pin 3(VI), and JP2 pin16 have the same name. The pins are connected to the same net, although the net lines are not draw continuously.

While the NET command is active, the status bar below shows properties of the selected net.

As mentioned before, nets with the same name define an electrical connection.

The NAME Command

EAGLE automatically allocates names such as B\$.. for buses, P\$.. for pins and N\$.. for nets.

- NAME and then • the net connected to IC1 pin OSC1 (16). A popup menu shows the predefined name of the net. Type in

OSC1

and • OK. The net immediately now has this name.

The names of components and busses can be changed in the same way.

The LABEL Command

The LABEL command allows you to place bus or net names on a schematic in any location.

- LABEL, locate the cursor on the net MCLR/PGM and • .

The name of the net is attached to the cursor and you can place it in any location. You can also rotate the label with the right mouse button. Locate the label approximately as shown in the figure and • to fix its position (near JP1 pin 3).

If net or bus names are changed, the relevant labels are also changed. Label text is not changed with the CHANGE TEXT command but with the NAME command and a click on the net line or the bus line.

CHANGE FONT or CHANGE SIZE changes the font or the text size.

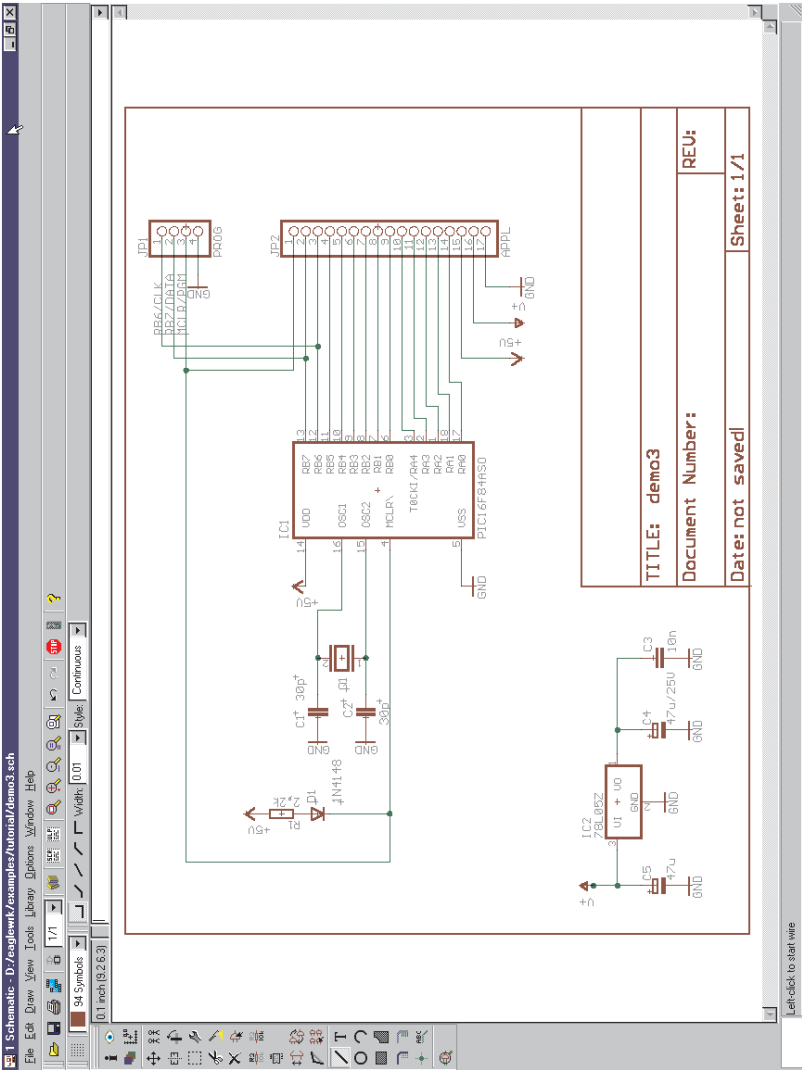
The DELETE Command

You can delete objects with this command. If it is applied to nets, wires or busses, a single segment is deleted at a time. To use this command, •

DELETE in the command toolbar, take the cursor to the object that is to be deleted, and ●.

Keep the *Shift* key pressed while deleting an object in order to delete the whole net or bus. More about this in the help function.

UNDO and REDO work here as well. GROUP, DELETE and a right mouse click delete whole groups.



Schematic demo1.sch

The JUNCTION Command

Dropping a net on another net line generates a connection between these two nets. The connection will be represented by a junction, that will be set automatically. Automatic setting of junctions can be switched off with the option *Auto set junctions* (\Rightarrow *Options/Set/Misc*).

In this case the JUNCTION command is used to draw a connecting node at the intersection of nets which are to be connected to each other.

- JUNCTION and a node is attached to the cursor. Locate the node at the junction of two net lines and • to fix it into place.

The SHOW Command

This is a good time to demonstrate the function of the SHOW command. This command is used to show names and other details of elements and objects. Complete signals and nets can be highlighted, as well as components.

To show for example the net V+, • SHOW in the command toolbar then move the cursor to the connection point of U1 pin VI (3) and •.

Please notice that EAGLE highlights the net wires and each pin connected by this net, as well as the pin name of each part to which it is connected. In addition, the signal is listed in the status bar as:

```
Net: V+, Class: 1 Power
```

While the SHOW command is active the net remains highlighted although you are panning the window by pressing the middle mouse button and moving the mouse or using the WINDOW command. Deactivate the SHOW command by clicking the stop sign icon and use WINDOW REFRESH (F2). Now the objects are no longer highlighted.

To show an object with a specific name, • SHOW and type the name (for example D0 \leftarrow) in the command line. You can subsequently type other names without the need to reactivate the SHOW command. This way you can mark one net after the other.

Do you wish to highlight several nets at the same time, enter in the command line:

```
SHOW RA4  $\leftarrow$   
SHOW RA3  $\leftarrow$   
SHOW RA2  $\leftarrow$ 
```

The MOVE Command

No electrical connection will be generated if you move a net line over a pin (using the MOVE command). On the other hand: if you move a pin over another pin or over a net line, an electrical connection will be generated, and a net line will be attached to the pin when the component is moved further. Remember the UNDO command if you want to detach the net line.

Check the connections with the SHOW command, as mentioned before. Additional one can export a net or pin list with the EXPORT command.

History Function

With the keys *up-arrow* and *down-arrow* you can recall the last keyboard instruction into the command line and execute it with the *Enter* key. The *Esc* key will delete the command line.

Use Alt+F2 to show the whole schematic on the screen, then type:

```
SHOW R1 ←
```

```
SHOW C1 ←
```

```
SHOW IC1 ←
```

Quit the SHOW command by clicking the stop sign icon. Redraw the screen, e.g. with F2 and press the *up-arrow* and *down-arrow* keys several times. As you can see, you can scroll through the list of the recently used commands. As soon as the desired command appears in the command line press the *Enter* key.

Completing the Schematic

Use the ADD command to add the remaining components and the symbols for +5V, V+, and GND from *supply1.lbr* (search pattern: *supply*). Supply symbols represent the power signals in your schematic and cause the ERC (Electrical Rule Check) to use special checks for them.

Remember that you can use the MOVE command to move objects around and that you can rotate elements attached to the mouse with a right mouse click.

Using the NET command, connect the pins of the components according to the schematic and connect the supply symbols to the related pins. Use the right mouse button to alternate between the orthogonal and diagonal modes while using the NET command. Use • to fix a segment.

If you place a net exactly on a connection point, the net is terminated at this location. Otherwise the net keeps following the mouse.

The SMASH Command

You will notice that when you rotate diodes and resistors from the horizontal to the vertical position, their reference designators and value texts rotate with the part. EAGLE provides a SMASH command that allows you to MOVE and ROTATE the name and value texts independently of the symbol. While moving a smashed text EAGLE draws a line from the text to the parts origin to show where the text belongs to.

To activate the command

- SMASH

Locate the cursor on the diode symbol and •. This separates the text from the symbol. Now click the MOVE icon, move the cursor to the name D1 for the diode, and •.

The text selection point is marked as a cross and resides, depending on the rotation, on the lower left or the upper right corner.

The name is now attached to the cursor. It can be moved to a better location and rotated with the right mouse button. When you have rotated and relocated D1, • to fix its location.

If you want to change the size of name and value texts which have been separated from the part with the SMASH command, use the CHANGE SIZE command (click the CHANGE icon and select *Size* from the menu).

SMASH may be used with groups.

Keep the *Shift* key pressed while clicking an object or inside the group in order to unsmash all texts. They will appear at their original positions.

The VALUE Command

EAGLE allows you to define or to change the value elements like resistors or capacitors. In the case of ICs the value informs you about the element type (e.g. 74LS00N).

- VALUE
- the resistor,

type the new value, 2.2k, • OK, and the new value is now displayed.

You can use the NAME command to change the names of resistors, capacitors, ICs, nets and buses accordingly. You can change the net names but you don't have to, unless you want to get a descriptive netlist.

The Electrical Rule Check (ERC)

If you haven't entered the complete schematic yourself you can now load the file *demo1.sch*.

The ERC command is used to test schematics for electrical errors.

The results are warnings and error messages that are generated and written into a file which has the same file name as the drawing but the extension **.erc*. This file is automatically displayed in a text editor window if messages were generated. To use the command click the ERC icon in the command toolbar.

The ERC quits our sample file with two messages:

```
WARNING: Sheet 1/1: POWER Pin IC1 VSS connected to GND
WARNING: Sheet 1/1: POWER Pin IC1 VDD connected to +5V
```

These messages inform you that the power pins are connected to other signals than expected. The power pins were named VSS or VDD in the library but are connected to GND and +5V. In our case this has been done on purpose, therefore the messages can be ignored.

Please note that the ERC can only discover possible error sources. It is up to you to properly interpret the ERC messages!

If you want to learn more about the ERC command, type

```
HELP ERC ←
```

in the command line.

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:



A board file will be generated in which the packages are positioned next to an empty board.

A further description follows in the chapter *Designing a PC Board*.

But now we want to introduce another important command that is necessary to design schematics first.

The BUS Command

Load the schematic *bus.sch* from the */eagle/examples/tutorial* directory.

A schematic with a bus structure appears. A bus has to be drawn with the

BUS command. It is named automatically (*B\$1..*).

A bus has no logical significance. It is a drawing element only. Logical connections (nets) are only defined with the NET command. Nets with the same name are identical even if they are on different pages of a schematic or optically not connected.

The bus name determines the signals contained in the bus. In our example the bus contains the signals VALVE0 to VALVE 11 and a signal named EN. Therefore the bus has been named EN, VALVE [0 . . 11] with the NAME command.

The bus in our example has not been finished, yet. There are still some connections to draw. Start to connect the following signals to IC7 by selecting the NET command and clicking on the bus line:

EN	IC7 Pin 14	EN
VALVE0	IC7 Pin 16	INA
VALVE1	IC7 Pin 15	INB
VALVE2	IC7 Pin 10	INC
VALVE3	IC7 Pin 9	IND

- NET in the commando toolbar and move the cursor over the bus, one grid line over the pin IC7-14. The net connection to the bus must originate from the bus and be drawn to the component pin, if you want to use this convenient way to name it.
- to set the starting point of the net, and a popup menu will appear with the net names for the bus.
- EN to select net EN, and move the cursor to IC7-14, using the right mouse button to change the line until it is drawn like the other net lines in this area.
- the pin's connection point to finish the net line.

Repeat this action for VALVE0 .. VALVE3.

Use the LABEL command to make the net names visible in the schematic.

If you want to cancel an action, click the UNDO icon, or use the *F9* key. Either by clicking on the REDO icon or by use of the *F10* key you can perform the cancelled action once again.

Use the MOVE command to move individual bus segments. Select a segment near to the end in order to move the end point. Select a segment somewhere in the middle, to move it to a parallel location. You can delete individual segments with DELETE.

The cursor takes on the form of four arrows when you want to select an object whose origin is very close to the origin of another object. In this sort of case, click the left mouse button to select the highlighted object. Click the right mouse button if you want to go on to the next possible object. Information about the selected object can be found in the status bar.

13 Automatic Forward&Back Annotation

You should always design your boards using Forward&Back Annotation controls; only then can you be sure that boards and schematics will be consistent with each other. This control mechanism is activated when you load a schematic and a board which have the same name and which are consistent with each other. EAGLE always loads both files if they exist in the same directory. *Consistent* in this context implies that the netlist, components, and values are identical.

If you load a schematic and a board which has the same name and which can be found in the same directory (or vice versa), EAGLE launches a consistency check. You have the chance to start an ERC if any differences are found. The results are displayed in a text editor window. They enable you to fix the inconsistencies manually. Using this method it is possible to draw a consistent schematic for an existing layout.

The Forward&Back Annotation will be cancelled if either only the schematic window or only the board window is activated. Any changes made can then lead to discrepancies in the files for the board and the schematic. Therefore always make follow this rule:

When working on a board, never close the schematic window (you can minimize it to an icon, however) — and vice versa.

EAGLE generates warnings before operations are carried out which would terminate the Forward&Back Annotation.

Under the control of the Forward&Back Annotation any change in the schematic results in an equivalent change of the board, and vice versa. Some changes can be made either in the board or in the schematic (e.g. naming components, nets, etc.). Others are possible only in the schematic (e.g. adding components). EAGLE prevents such operations in the board and prompts you to use the Schematic Editor.

To monitor the Forward&Back Annotation load the *demo2.sch* file. The board *demo2.brd* will be loaded automatically into the Layout Editor.

Now size both of the windows so that you can see them both on the screen. Change some names and values with the NAME and VALUE commands. You will notice that the names and values change in both windows. Experiment also with the DELETE command and remember the UNDO and REDO commands.

14 Designing a PC Board

In this section you will create a small PCB design and modify an existing design using the Layout Editor. First, you will create a board without a schematic.

This section is useful mainly for those users who have no Schematic Module. If you have the Schematic Module you would normally not have to deal with the steps described in the following section. You should, however, read through this section as it deals with some generally useful points.

Designing a Board without a Schematic

Open a new file (\Rightarrow *File/New/Board* in the Control Panel) and enlarge the editor window.

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 • the *Default* button and • *OK*.

The board outlines must be drawn with the WIRE command in layer 20, *Dimension*: • WIRE, and select layer 20 from the combo box in the parameter toolbar.

Position the cursor at the zero point of the coordinates, and • to determine the starting point of the outline. Move the cursor slightly to the right, click the right mouse button until both lines are orthogonal (90 degrees), and position the cursor near the coordinates (4.00 3.00).

Fix the outline at this point with • and move the cursor back to the coordinates' zero point.

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, or use UNDO and REDO to recall the previous actions and perhaps make changes.

Alt+F2, or clicking the Zoom-to-fit icon, will fit the board into the screen.

Placement Grid

Before placing components, it is important to set up the grid for component placement. The component placement grid may be different from the grid used for drawing the board shape, and is almost always different from the grid used for routing interconnect wires. For the following exercise we will use the default grid of 0.05, inches which is already set.

Placing Components

- ADD in the command toolbar and search for *DIL14*.

Double-click on a 14-pin DIL package entry. Now it is attached to the cursor. It can be rotated with the right mouse button and then placed with the left mouse button. Place two DIL14 packages.

Use the *F3* and *F4* key to zoom in and out.

If you like to place the component in any rotation in your layout it is possible to define any value in the *Angle* field of the parameter toolbar while the component is attached to the mouse. To do this click into the combo box, type in the value for rotation, and press the *Enter* key. Now the rotated component follows the mouse and can be placed.

Use the ROTATE command to change the orientation of components after they have been placed in the layout.

ROTATE works in 90-degree steps by default.

To rotate components in any angle enter the desired value in the *Angle* field of the parameter toolbar while the command is active. Now click the component to rotate it. Keep the mouse button pressed after selecting the component and you can rotate it while moving the mouse. The current angle will be shown in the parameter toolbar.

If you like to use another package than the predefined one (e.g. a smd instead of a through-hole package), you can use the REPLACE command. For detailed information please take a look into the help function.

Placing SMD Packages

Now use ADD to place two 1210 packages on the board (search pattern: *R1210*). If you know the package name, you can type

```
ADD R1210 ←
```

or

```
ADD R1210@smd-ipc ←
```

in the command line to fetch the package from a certain library.

If you intend to place the package in a certain angle, you can enter the value

directly:

```
ADD R1210@smd-ipc R22.5 ←
```

The SMD pads appear in red, which means, that they are on the layer 1, *Top*, of the board. To transfer them to the Bottom layer use the MIRROR command. Click on the MIRROR icon in the command toolbar and • on the package.

As long as the MIRROR command is active, you can move packages to the other side of the board. For the next exercise the packages should be placed on the Top layer (red).

Providing Names

To assign a name to the packages just placed:

- NAME in the command toolbar.

Move the cursor near the origin point (marked with a cross) of the first DIL14 and •. A popup window appears. Type

```
IC1 ←
```

and the new name is assigned to the package. Repeat this process to name the remaining packages IC2, R1, and R2.

Providing Values

To assign values to an element:

- VALUE in the command toolbar.

Move the cursor near the origin of IC1 and •.

A popup window appears. Type

```
CD4001 ←
```

and IC1 now has the value *CD4001*. Using the VALUE command assign *CD4002* to IC2, *100k* to R1, and *22k* to R2.

Defining Signals

The next step is to define signals and establish their connections using air-wires (rubberbands). First, connect the ground pads:

- SIGNAL and type

```
GND ←
```

• on pad 7 of IC1 (IC1-7) and move the cursor to IC2-7 and •• to terminate the GND airwire.

The two pads are now connected to the GND signal.

Next we will connect VCC. Type

VCC ←

- on IC1-14, move the cursor to IC2-14 and •• to terminate the VCC airwire.

Define further signals using the same procedure.

If you don't want to specify names for the signals at this time • a pad to start a signal and •• a pad to terminate it (or click the stop sign icon). EAGLE will then generate net names automatically which can be changed with the NAME command.

***EAGLE terminology:** Pads are through-holes for conventional components (used in packages). Pins are connection points for schematic symbols. Smd's are the pads of surface mounted devices (used in packages).*

Airwires can be deleted with the DELETE command if you don't work under Forward&Back Annotation control (in such a case you have to delete a signal by deleting the related nets in the schematic).

Please note that UNDO and REDO function under the control of the Forward&Back Annotation, too.

Defining Signal Classes

The CLASS command allows you to define signal classes and to assign certain values concerning wire width and minimum distance to other signals, and minimum drill diameter for vias to each class.

For example, power supply signals may be routed with a greater wire width (higher current) or a greater value for clearance (higher voltage). The autorouter uses these values for routing, too.

The default value is 0 for all attributes (no classes defined). This means the values set in the Design Rules are valid for all signals.

Various classes are used in the board file *hexapodu.brd*.

Creating a Board from a Schematic

If you have the Schematic Module and the schematic is already drawn, you only need a few steps to get the same result as that described in the previous section.

Generating a Board File

Load the file *demo1.sch* and click on the BOAR D icon :



With this command you create a board file with the same name as the loaded schematic (*demo1.brd*). Answer the *Create file?* question with • OK. Maximize the Layout Editor window.

The white frame on the right of the window symbolizes the board outlines. It is made up of wires in the layer 20, *Dimension*.

- MOVE, and • the right vertical edge of the board shape somewhere in the middle. Move the cursor a little to the left and • .

You have now reduced the size of the board. You can change the board size at any time.

Of course you could also DELETE the border and import, for example, predefined board outlines via a script file (SCRIPT command).

Component Placement

Click on the Zoom-to-Fit icon, to fit the drawing in the window. The components are located on the left side of the board.

- MOVE, • the biggest IC somewhere in its center and move the cursor 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. • to fix the position of the component. Place all of the components using the MOVE command.

An other way to move an element is to select it by its name. • MOVE and type in the command line.

JP1

This element is now attached to the mouse cursor.

Click the RATSNEST icon to calculate the airwires so that they show the shortest possible connections. Repeat this command whenever you want to check how good your current placement is (short airwires, no twisted buses etc.).

Please note:

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 3.9 x 3.2 inch. To route the layout or to use the autorouter you have to move all components into this area first.

Autorouter: A Brief Example

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). If it is taking too long, interrupt the Autorouter by clicking the stop sign icon. Confirm the question *Interrupt?* with • Yes.

If you don't like the result, reverse it with the command • RIPUP.

If you would like to change certain routed tracks into airwires, click these tracks and start the ripup process by a click on the traffic light icon in the action toolbar.

If you would like to change all routed tracks into airwires, • the RIPUP icon and then • the traffic light icon. Confirm the question *Ripup all signals?* with • OK.

You can start the Autorouter at any time, regardless of whether there are routed tracks or only airwires on the board. Typically, supply signals and other critical signal paths are routed manually, before the Autorouter is used.

Tracks which are layed out before starting the autorouter won't be changed anymore.

Routing Manually

The ROUTE command changes the airwires into routed tracks.

- ROUTE in the command toolbar.
- starting point of an airwire.

As for the WIRE command, further parameters, such as width or target layer, can be entered with help of the parameter toolbar.

All values relate to the current unit selected with the GRID command.

Move the cursor to route the signal, • to fix the current segment. • • to fix the last segment and end the route operation for the whole signal.

Since the ROUTE command is still active, you can immediately start routing a new signal.

While the ROUTE command is active, it is possible to change the bend mode between two wire segments with the right mouse button. Simply try the various options; it is also possible to lay out tracks as arcs (see also SET command, parameter *Wire_Bend*).

If you change the target layer during the routing process by selecting it from the combo box in the parameter toolbar, the following wire segments will be drawn on the new layer. The necessary via-hole will be generated automatically by EAGLE.

In case of ending a track at the same position as another track which belongs to the same signal but lies in another signal layer, EAGLE does not automatically set a via. To have one placed, keep the *Shift* key pressed while ending the wire.

While laying out tracks EAGLE calculates at any time the shortest connection to the next point which belongs to the same signal and shows this with an airwire.

Use the MITER command to bevel wire bends. You can do this with a straight line or with an arc, which will be determined by the miter radius. Some wire bend modes are influenced by the miter radius. More details can be found on the MITER, SET, WIRE commands' help pages.

The EAGLE Freeware respectively the Light edition does not support blind and buried vias due to their limitation of the numbers of signal layers. More information about this can be found in the help function and in the reference manual.

Board Changes

Once you have completed the routing of the board you can make changes, e.g. you can:

- move and arrange wire segments and components with MOVE and SPLIT,
- use the RIPUP command to change routed tracks to airwires,
- use DELETE to erase signals (only without Forward&Back annotation),
- replace package variants with CHANGE PACKAGE or REPLACE (without schematic). In *demo3.brd* the package of IC1 has been replaced by a SMD package,
- modify the Design Rules (for example, Restricting settings),
- use SMASH to detach texts from components in order to MOVE,

- ROTATE, or CHANGE them,
 - miter wire joints.

Further Usage of the Layout Editor

In this section you will modify a routed demo board. Load the file *demo2.brd*, and enlarge the editor window.

Next a few important commands will be repeated.

The DISPLAY Command

It is often easier to maintain a good overview if some information is not shown.

- DISPLAY

and select with your mouse the layer 21, *tPlace*. This layer contains information for the top side silkscreen of the board. By selecting or deselecting it, the layers 23 *tOrigins*, 25 *tNames*, 27 *tValues*, and 51 *tDocu* will be shown or hidden.

To activate this change, • OK.

The MOVE Command

With the MOVE command you can move for instance wires (lines or signal tracks). Selecting a wire segment near an end point will move the end point of the wire. Keep the *Ctrl* key pressed while selecting it and the end point will jump on the current grid.

Selecting the wire in the middle will move it in parallel. Keeping the *Ctrl* key pressed changes the wire into an arc.

You can also move vias (through-holes which connect Top and Bottom layer tracks). When moving vias, the attached wires are also moved.

To move components placed on the Top layer, layer 23 *tOrigins* has to be displayed. The same applies to components placed on the Bottom layer and layer 24 *bOrigins*.

As long as the MOVE command is active, you can rotate the object with the right mouse button by 90 degrees or by any angle given in the *Angle* combo box in the parameter toolbar.

The *Spin* flag (only available in the Layout Editor) determines if texts with a rotation of 180 degrees will be shown upside down or stay readable from the bottom of the drawing (default). This also works with the ROTATE command.

The status bar shows information about the selected object.

The GROUP Command

One of the most useful commands of EAGLE is the GROUP command. It allows you to select several objects, change their attributes and move, rotate or mirror them all at once. To use the GROUP command

- GROUP

Then, by clicking and releasing the left mouse button, draw a polygon around a group of objects and close the polygon by pressing the right mouse button (don't use the POLYGON command). The selected objects are now highlighted.

Please make sure that you only select objects which are in a visible layer. Packages on the Top layer can only be selected if layer 23 tOrigins is visible, and packages at the Bottom layer can only be selected if layer 24 bOrigins is visible. Use the DISPLAY command to show or hide layers.

Next select the MOVE command and use the right mouse button to attach the group to the cursor. This way you can move all the objects simultaneously, rotate them with the right mouse button and fix them with the left mouse button.

After a group has been defined with the GROUP command, the attributes of the contained objects can be changed with the CHANGE command. Select a group that contains some wires, • CHANGE in the command toolbar, • Width, and • 0.032. Then click the right mouse button somewhere in the editor window. You can reverse the action with the UNDO command.

A rectangular group can be defined by selecting the GROUP icon, clicking one corner of the area of interest, keeping the mouse button pressed and dragging the mouse button until the rectangle contains the desired objects.

The SPLIT Command

With the SPLIT command you add a bend in a wire.

- SPLIT in the command toolbar
- on a wire segment near its target point

Drag the wire on the screen a bit. You will see that the longer segment remains as a direct line to the selected point, while the shorter segment splits into two. The angle of the two new segments is controlled with the right mouse button. • fixes the wire segments.

The CHANGE Command

Use the CHANGE command, to change the width of wires or to move a wire to another layer. To change the width of the wire:

- CHANGE
- WIDTH in the popup menu
- the value for the new width

Then move the cursor to the wire segment to be changed and •.

To change the width to a value not present in the menu, e.g. to 0.23 inches,

- the entry A window pops up where you can type in the value.

You could also use the command line:

```
CHANGE WIDTH .23 ←
```

and click the wire segment.

To move a wire segment to another layer :

- CHANGE
- LAYER
- desired layer
- wire segment

In case a via is needed to complete the signal path, EAGLE will automatically insert it. If a via becomes redundant, EAGLE will automatically remove it.

The ROUTE Command

Use the ROUTE command to change an airwire into a wire. You can fix the position of a wire segment with •, then change direction and fix the next segment until the airwire is completed.

See also the section *Routing Manually* some pages before.

The RIPUP Command

If you would like to turn for example the routed signal tracks GND and VCC into airwires (this process is called *ripup*), click the RIPUP icon in the command toolbar and type:

```
GND VCC ←
```

Pressing F9 twice reverses the action.

If you would like to ripup all signals except GND and VCC, type:

```
! GND VCC ←
```

after the RIPUP command has been activated.

The SHOW Command

Use the SHOW command to highlight airwires, wires or components. Click on the Zoom-to-fit icon and then the SHOW icon. Then type:

```
IC1 ←
```

to locate IC1. The status bar shows information about the object.

Refresh Screen

Use the *F2* key to refresh the screen contents or click the Redraw icon.

Undo/Redo Function

All actions mentioned can be reversed with the UNDO icon in the action toolbar (F9) and repeated with the REDO icon (F10).

Inner Layers

Inner layers (Route2...15) can be used for routing in the same way as the Top and Bottom layers.

This is not possible in the freeware (light edition).

Supply Layers

Only possible in standard and professional edition!

Inner layers (Route 2..15) can be used as a signal or ground plane simply by renaming the layer *\$signalname*, when all signals with that name will then be connected to this layer. To demonstrate this feature load the board file *demo2.brd* and type:

```
SHOW GND ←
```

The GND signal will be highlighted.

Then type:

```
RIPUP GND ←
```

The signal GND will now be displayed as airwires only.

Now we will define layer 2 as the ground layer by naming it *\$GND* (the name of the signal must be preceded with the \$):

```
LAYER 2 $GND ←
```

You can also select Layer 2 *Route2* in the DISPLAY menu, click the *Change* button, activate the option *Supply Layer* and type in the name GND.

Click the RATSNEST icon to make the airwires of the signal GND

disappear.

To have a look at the result switch of all layers except for the \$GND layer. This can be achieved by clicking on the DISPLAY icon, or simply by typing the following command on the command line:

```
DISPLAY NONE $GND ←
```

Remember you can use lower case characters and abbreviate key words. So you can type

```
dis none $gnd ←
```

Now type

```
SHOW GND ←
```

and the thermal symbols which connect the GND signal to the supply layer are highlighted. The annulus (round) symbols isolate the through-holes not belonging to the GND signal.

In our example the elements C1 and C2 don't have a connection to the inner layer yet. You have to ROUTE a short track out of the smd and place a VIA on the tracks end. Now the connection to the inner layer is made.

Supply layers defined with \$name are plotted inversely, i.e., objects with the color of the supply layer define copper-free areas. The thermal symbols connect the ground plane with the through-hole using four conducting paths.

Copper Pouring with the POLYGON Command

The POLYGON command enables you to define areas which belong to a signal, connecting all of the related pads to this signal with thermal symbols. Such a signal retains a user-defined distance to any other signal path. You can design layers that contain multiple polygons such as different ground areas, and you can design polygons on multiple layers.

To demonstrate this feature of EAGLE, let's fill the Top layer of a board with the GND signal. Reload the board *demo2.brd* once again, enlarge the window, and ripup the GND signal:

```
RIPUP GND ←
```

Use the DISPLAY command to switch on the layer 1 *Top*, 17 *Pads*, 18 *Vias*, and 20 *Dimension*. Use the button *None*, in the menu, to switch off the display of all other layers first.

Click the POLYGON icon in the command toolbar and type:

```
GND ←
```

to provide the name *GND* for the polygon to be defined. Only then will it belong to the GND signal.

Select the Layer *Top* from the combo box in the parameter toolbar. Then:

- left hand upper corner of the board outlines,
- right hand upper corner,
- right hand lower corner,
- left hand lower corner.

The double click closes the polygon.

To start the calculation of the filled area, click the RATSNEST icon.

Since this is a very complex operation it can take some time.

As before, the pads belonging to the GND signal are connected with thermal symbols. Check this with:

SHOW GND ←

In this case everything shown in the layer color is copper, since this layer is not plotted inversely (only supply layers defined with *\$name*).

After a board has been loaded, polygons are displayed with their outlines. The filled areas are displayed only after the RATSNEST command has been executed. Inversely, the RIPUP command and a single click on the edge of a polygon results in the outline display of this particular polygon.

Call up the help function to find out more about the POLYGON command.

15 Autorouter

No Autorouter on earth will lay your board out exactly as you would like. But it can free you of a lot of boring work. In this section we want to demonstrate that you can easily combine manual and automated routing.

Load the board *hexapodu.brd*.

Switch off layer 21 *tPlace* using the DISPLAY command, so that the components are not shown anymore.

This board contains manually routed signals named AC1 and AC2. Rectangles in the layers 41, *tRestrict*, and 42, *bRestrict*, have been used to create restricted areas for the Autorouter. Within these areas the Autorouter is not allowed to route tracks on the Top or Bottom layers. Component B1 is covered by a restricted area drawn in layer 43, *vRestrict*. This means the Autorouter must not set vias there.

Start the Autorouter by clicking the AUTO icon in the command toolbar.

A popup menu appears where you can enter individual settings (see help).

You should choose a routing grid of 10 mil (0.254 mm) for *hexapodu.brd*.

You can also load the autorouter parameters for this file from the control file *hexapodu.ctl* by clicking the *Load..* button.

As we want to route all of the unrouted signals • OK.

In case you don't want to change the settings of the Autorouter you can start it by typing:

```
AUTO; ←
```

in the command line. The menu is skipped in this case.

Watch the status messages appearing in the status bar. They inform you, for instance, of how many signals have been routed, or of how many vias have been placed at the moment. You will notice that the number of vias goes down during the Optimize passes.

If you want to interrupt the Autorouter click on the stop icon.

A protocol of the routing run is stored in the file *hexapodu.pro*. Load it into a text editor window to have a look at it.

The board routed by the Autorouter can be edited like any other board. Should the routing result be less than 100%, you can ripup a few critical signals and route some of the remaining airwires manually. If you want to restore the original status, turn routed tracks into airwires (with the exception of AC1 and AC2) with the command

RIPUP ! AC1 AC2 ←

The autorouter uses the width given in the Design Rules (\Rightarrow *Edit/Design Rules, Sizes* tab, *Minimum width*) for his tracks. If there are values given in the CLASS command to define various net classes (as in the example file *hexapod.brd*) the autorouter will also take care of them. In this case the greater value will be taken.

To define restricted areas for the autorouter use layer 41, *tRestrict*, for the Top layer, respectively layer 42, *bRestrict*, for the Bottom layer.

Restricted areas in layer 43, *vRestrict*, forbids setting vias.

The autorouter is not able to route rounded tracks.

16 Design Rule Check

When starting the layout development you should already think about the Design Rules. Rules that have to be confirmed with the board manufacturer as well. You can define them with the help of the DRC dialog. Click the DRC icon and verify or change the default values. A click into one of the parameter fields shows a describing picture (for example in the *Restricting* tab). • *Apply* to save the Design Rules in the board file. *Check* starts the Design Rule Check. The *Select* button allows you to select a certain area of the layout to be checked. Simply drag a rectangle with the mouse around the area.

The DRC command checks if the board corresponds with the user-defined Design Rules.

Load the file *demo3.brd*. To run the test, click on the DRC icon in the command toolbar. A menu appears which allows you to set up your Design Rules. • *Check* to start the DRC.

After finishing the checks the status bar should show the message *No errors*. Now you can be sure the board matches your Design Rules.

Move a red wire across a few other red wires. Then start the DRC once again by typing

```
DRC; ←
```

The semicolon prevents the DRC menu from appearing.

The status bar shows the number of errors. An errors window will open automatically showing an error list. As soon as you have corrected the errors in the layout, you can delete them by clicking the button *Del all*.

Press F1, to find out more about the ERRORS command.

If you find objects in your board which cannot be deleted with the DELETE command it might be the result of a DRC. To delete them type in the command line:

```
ERRORS CLEAR
```

17 Libraries

The components you are adding to schematics and boards are stored in libraries. The Library Editor has the same user interface as the Schematic and the Layout Editor. Therefore, you only need to know a few additional commands for defining your own components.

A library normally consists of three basic elements:

- **Package:** The footprint in the layout
- **Symbol:** The drawing for the schematic
- **Device:** The real component, consisting of symbols and packages

Here a short example for library creation:

Open a new library file via the menu *File/New/Library* in the Control Panel. The Library Editor windows opens.

Resistor Package



Select the package editing mode via the icon in the action toolbar, and enter the package name *R-10* in the *New* field. Answer the question *Create new package 'R-10'?* with *Yes*. Later when creating a new symbol and a new device you will again have to answer the corresponding questions with *Yes*.



Use the GRID command to set an appropriate grid size for the pad placement. 0.05 inch (i.e. 50 mil) is usual for standard components with lead wires.



For a resistor with lead-wires, select PAD, and set the pad shape and the drill diameter in the parameter toolbar. Default value for pad diameter is 0. It should not be changed. The final diameter in the layout results from the values given in the Design Rules. Then place two pads at the desired distance. The origin of the drawing will later be the identifying point with which a component is selected. For this reason it should be somewhere near the center of the device.



For a SMD resistor, select SMD, and set the pad dimensions in the parameter toolbar. You can either select one of the offered values, or directly type the length and breadth into the entry field.

Select *Top* as the layer, even if the component will later be placed on the underside of the board. SMD components are located on the other side of a board using the MIRROR command. This moves the elements in all the *t..*-layers into the corresponding *b..*-layers.

Place the two SMD pads (which in EAGLE are just called SMDs) at the

desired distance.

To use round SMDs (BGAs) define a square one first, then CHANGE the value for *Roundness* = 100%.



You can now enter the names, such as *1* and *2*, for the pads or SMDs using the NAME command.

A different procedure is however recommended for components with many sequentially numbered pads:

Select the PAD command, type in the name of the first pad, e.g. '*1*' (the inverted commas must also be entered), then place the pads in sequence.



Now use the WIRE command to draw the silkscreen symbol in layer 21 *tPlace*. This layer contains what will be printed on the board. It is up to you how much detail you give to the symbol. Set a finer grid size if it helps. Take the information provided in *library.txt* (in *eagle/doc*) as a guideline for the design of components. You may also use the ARC, CIRCLE, RECT and POLYGON command to draw silkscreen symbols.

Please take care in layer 21 *tPlace* not to cover any areas that have to be soldered. In layer 51 *tDocu* a more realistic appearance can be given which is not subject to this limitation. Layer 51 *tDocu* is not used to print onto the board itself, but is a supplement to the graphical presentation which might be used for print documentation. In the example of the resistor, the symbol can be drawn in layer 21 *tPlace*, but the wires, which go over the pads, are drawn in layer 51 *tDocu*.



With the TEXT command you place the texts *>NAME* in layer 25 *tNames* and *>VALUE* in layer 27 *tValues* in those places where in the board the actual name and the actual value are to appear.

SMASH and MOVE can be used later to change the position of this text relative to the package symbol on the board.



The CHANGE command can be used at a later stage to alter object properties such as the stroke thickness of texts (*ratio*), text height, or the layer in which the object is located.

If you want to change the properties of several objects at one go, define a group with the GROUP command, click the CHANGE command, select the parameter and the value, and click into the group with the right mouse button.

Example: Use GROUP to define a group that contains both pads, then select CHANGE and SHAPE/SQUARE. Click on the drawing surface with the right mouse button. The shape of both pads changes.

The DESCRIPTION command allows an info text about the package. This text and the package's name will be taken in consideration by the search function of the ADD command.

Resistor Symbol



Select the symbol editing mode, and enter the symbol name *R* in the *New* field. This name only has a meaning internal to the program, and does not appear in the schematic.

Now check that 0.1 inch is set as the grid size. The pins in the symbol **must** be placed on this grid, since this is what EAGLE expects.



Select the PIN command. You can now set the properties of these pins in the parameter toolbar, before placing them with the left mouse button. All these properties can be changed at a later stage with the CHANGE command. Groups can again be defined (GROUP) whose properties can then be altered with CHANGE and the right mouse button. See help function for further details.



The NAME command allows you to name pins after they have been placed.



The schematic symbol is drawn in layer 94 *Symbols* using WIRE and the other drawing commands. Place the texts *>NAME* and *>VALUE* in the layer 95 *Names* and 96 *Values* (TEXT command). Place them where the name and value of the component are to appear in the schematic.

For fine adjustment choose a finer grid. This can be done while the TEXT command is active. Afterwards change the grid to default value 0.1 inch again.

Resistor Device



Create the new device *R-10* with this icon. When you later use the ADD command to fetch the component into the schematic, you will select it by using this name. It is only a coincidence that in this case the name of the package and the name of the device are the same.

To define devices that are available in several technologies and package variants you have to use wild cards in the device name to determine the position of these names.

* represents the position of the technology name, ? the package name. Defining for example a device like a 7400 in two technologies (**L**, **LS**) the correct device name is *74*00*. The name of the package variant will be

added at the end of the name automatically. If you wish to see the package variant's name, for example, at the beginning of the device name you have to use the ? like this: ?74*00.

Click the *New* button on the right lower area of the Device Editor window to assign a package. For our example, please choose the package *R-10*. To allow further package variants click *New* again.

The PREFIX command is used to specify a prefix for a name. The name itself will initially be automatically allocated in the schematic. For a resistor this would, naturally enough, be *R*. The resistors will then be identified as *R1*, *R2*, *R3* etc.. The names can be altered at any time with the NAME command.

You can specify with the VALUE command whether the device's value can be altered in the schematic or in the board. Value must be *On* for resistors. For other devices it may be wise to set value *Off*.



The previously defined resistor symbol is fetched into the device with the ADD command.

If a device consists of several schematic symbols which can be placed independently of one another in the circuit (in EAGLE these are known as *gates*) then each gate is to be individually brought into the schematic with the ADD command.

Set an addlevel of *Next* and a swaplevel of *0* in the parameter toolbar, and then place the gate near the origin.

The swaplevel of a gate behaves very much like the swaplevel of a pin. The value of *0* means that the gate cannot be exchanged for another gate in the device. A value greater than *0* means that the gate can be swapped within the schematic for another gate in the same device and having the same swaplevel. The command required for this is GATESWAP.



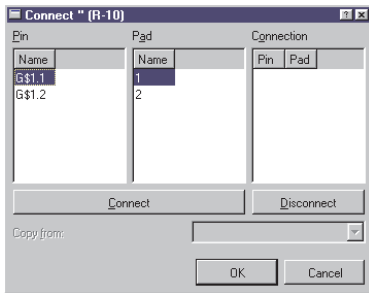
You can change the name of the gate or gates with the NAME command. The name is unimportant for a device with only one gate, since it does not appear in the schematic. If a device consists of several gates the element's name in the schematic will be expanded by the gate name.

Example:

The gates are called *A*, *B*, *C* and *D*, and the name of the component in the schematic is *IC1*, so the names which appear are *IC1A*, *IC1B*, *IC1C* and *IC1D*.

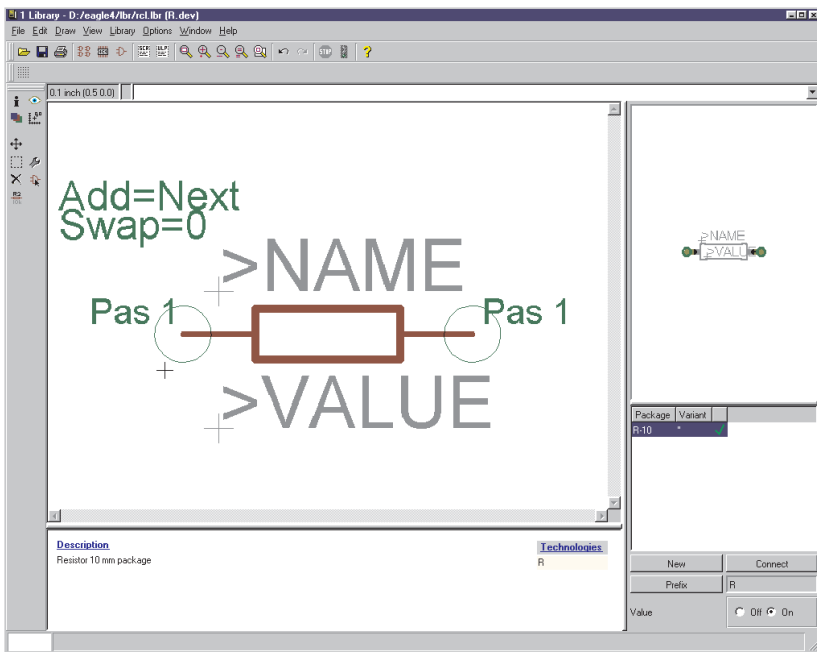
With the CONNECT command you specify which pins are taken to which package pads.

Click the *Connect* button now.



The Connect window

In this example the resistor gate has been named G\$1 automatically. This is the reason why you see the pin name G\$1.1 and G\$1.2 in the column *Pins*. The column *Pad* shows the pads placed in the package. Click on a pin and a pad entry and click the *Connect* button. If you want to disconnect a pin from a pad, select the pair in the *Connection* column and click *Disconnect*. *OK* ends the CONNECT command and closes the window.



The Device Editor

You can enter information concerning the device after clicking the *DESCRIPTION* command. The entered text will be displayed in the

Control Panel, when you select the device in the tree view. It will also be checked by the search function of the ADD command.

Now the definition of the resistor is complete. You can use it in a schematic now.

Keep in mind to load the new library with the USE command. Otherwise it is not available for ADD.

18 Output of Drawings and Manufacturing Data

EAGLE can output drawings, for example for documentation purposes, using the **PRINT** command. This command can be found in the *File* menus of the Schematic or Layout Editor.

You can use the printers defined under Windows with this command. The Linux version generates Postscript files that can be sent to *lpr* or to a file.

A printout of the drawing in the active editor window with the actual layer settings (**DISPLAY** command) will be generated.

Film and manufacturing data are generated with the CAM Processor. Start it with the icon in the action toolbar of the Layout Editor window.

The CAM Processor uses its own drivers, which can be defined or modified by the user (see file *eagle.def* in directory *eagle/bin*).

Data for a bill of material, for mounting, milling or testing machines etc. can be generated with the help of EAGLE User Language Programs. Information about ULP's can be found in the file headers or with the help of the descriptions in the Control Panel.

Output a Schematic with the **PRINT** Command

The schematic *demo1.sch* is to be printed in black/white and full format on one page.

Load the file *demo1.sch* and choose the layers which should appear on the print-out with the **DISPLAY** command. All layers visible on the screen will be printed.

Click the **PRINT** icon in the action toolbar. Check the boxes *Black*, *Solid* and *Rotate* (as the drawing is in landscape format). The boxes *Mirror* and *Upside down* are not marked.

For both *Scale factor* and *Page limit* type in *1*. This specifies that the drawing is to be output at a scale factor of 1, provided it fits onto one page. If not, EAGLE changes the scale factor, so that the drawing does fit onto one page. With *Page limit* 0 the drawing will always be printed with the set scale factor.

The printer can be selected by using the button *Printer*.

The button *Page* leads you to the page setup parameters. If the box *Caption* is checked, the drawing will be printed with a footer, containing the file name, date, time, and scale factor.

Generating Image Files for Documentation Purpose

To generate image files use the EXPORT command with its IMAGE option. You can generate different pixel graphic file formats (like bmp, tif, png and others) or use the clipboard.

Generating Gerber Data with the CAM Processor

The same steps are usually required for each board whenever films and manufacturing data are generated. This process can be defined as a CAM Processor job.

The file *gerber.cam*, which can be found in the default subdirectory for CAM jobs, automates the output of Gerber data for double sided boards.

Attention: *The process described can only be used for Gerber photoplotters with flexible aperture wheels, not for photoplotters with fixed aperture wheels.*

Please contact your board house to confirm which data are needed.

Load the job into the CAM Processor, either by double-clicking the entry with the name *gerber.cam* in the Control Panel's tree view (*CAM Jobs*), or by clicking the CAM Processor icon in the Layout Editor window and selecting *gerber.cam* in the file dialog (\Rightarrow *File/Open/Job*).

In case you have started the CAM processor from the Control Panel, load the board file *demo3.brd*:

\Rightarrow *File/Open/Board* and •• *demo3.brd*

Click the button *Process Job* and confirm both prompts *Delete name.\$\$\$...* and *More than ...* with • OK.

Now all of the necessary files will be written into the *Project* directory (where the drawing and board files for this project are located).

The files have the following meanings:

<i>demo3.cmp</i>	Component side
<i>demo3.sol</i>	Solder side
<i>demo3.plc</i>	Silkscreen for component side
<i>demo3.stc</i>	Soldering mask for the component side
<i>demo3.sts</i>	Soldering mask for the solder side
<i>demo3.whl</i>	Aperture wheel file
<i>demo3.gpi</i>	Information file, not relevant
<i>demo3.\$\$\$</i>	Temporary file, to be deleted

The first six files need to be sent to your board manufacturer.

Generating Drill Data

Drilling data can be generated accordingly by using the job *excellon.cam*. This job consists of one single step. The EXCELLON device generates a file that contains both drill data and drill table. The output file has the file extension *.drd*.

This file has also to be sent to your board manufacturer.

Further information can be found on the CAM Processor help pages.

Further Manufacturing Data

EAGLE provides a number of so called User Language Programs, which are able to generate a variety of data. For instance, data for mounting machines, test equipment, statistic data for the boardhouse (number of drills, parts, layer and so on), bills of material, or even data for milling a prototype board.

A list of all User Language Programs can be found in the tree view of the Control Panel, *User Language Programs* branch. Select an entry to get the description on the right.

More information can be found in the next chapter of this tutorial.

19 Data Exchange

The EAGLE User Language

EAGLE comes with an integrated C-like User Language interpreter which has access to virtually all data (in EAGLE or external files). User Language programs can create any file type and therefore generate any data format used by other software or hardware.

A good example is the file *bom.ulp* which generates a bill of material.

With the EAGLE User Language one has a powerful tool to define user-specific commands which even could be assigned, for example, to a certain function key.

Please have a look at the files with the extension **.ulp* and the User Language section of the help function to get a better idea of the possibilities inherent in the program.

Additional User Language programs can be found on our web pages:
<http://www.cadsoftusa.com/download.htm>.

Script Files — A Flexible Input Interface

Script files are text files which can contain any EAGLE command (see SCRIPT command). Script files enable the user to implement his own functions. Thus you have a flexible input interface defined by the EAGLE command syntax (see EAGLE help).

Start the Library Editor and use EXPORT SCRIPT after loading a library to generate a script file. This is a good example to learn about the EAGLE command syntax.