EAGLE

***E**ASILY **A**PPLICABLE **G**RAPHICAL **L**AYOUT **E**DITOR*

Tutorial

Version 6

Schematic – Layout – Autorouter
for Linux® Mac® Windows®

CadSoft Computer
www.cadsoftusa.com

1th Edition

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

USA and other countries:

Phone:	+1 (954) 237 0932
Fax:	+1 (954) 237 0968
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!

Table Of Contents

1 WHAT TO EXPECT FROM THIS TUTORIAL

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 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 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*.

The differences between EAGLE for Linux, Mac, and Windows are minimal.

> ***General comment about EAGLE component libraries:***
> *The component libraries supplied with EAGLE have been compiled with great care as an additional service to you, our customer. However, the large number of available components and suppliers of these components means that the occasional discrepancy is unavoidable. Please note, therefore, that CadSoft takes no responsibility for the complete accuracy of information included in library files.*

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 is required:

- Windows XP, Windows Vista, or Windows 7,
- Linux based on kernel 2.6 for Intel computers, X11
  with a minimum color depth of 8 bpp, 32-bit runtime environment with the following libraries: libpng14.so.14, libssl.so.1.0.0, libcrypto.so.1.0.0, libjpeg.so.8
- Mac OS X version 10.6 or above for Intel computer,
- a minimum graphics resolution of 1024 x 768 pixels,
- preferably a 3-button wheel mouse.

Professional Edition

General

- maximum drawing area 150 x 150 inches (about 4000 x 4000 mm)
- resolution 0,00325 μm (micron)
- grid in mm or inch
- up to 255 drawing layers
- command (Script) files
- C-like User Language for data import and export
- readable, fully documented XML data structure
- easy library editing
- composition of self-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)
- arbitrary pad shapes in the Package Editor
- library browser with powerful search function
- support of technology feature (e.g. 74*L*00, 74*LS*00..)
- easy definition of labelled drawing frames
- free definable attributes, applicable for Devices in the Libraries and in Schematic or Layout
- support of assembly variants
- easy-to-use dimensioning tool
- merging of different projects with maintaining consistency

(Design Reuse)
- integrated PDF data export function
- export function for graphic files (BMP, TIF, PNG...)
- printouts via the OS's printer drivers with print preview
- partlist generation with database support (*bom.ulp*)
- Drag&Drop in the Control Panel
- user-definable context menu with object-specific commands for all objects, available through a right mouse click
- properties of objects can be accessed and edited via context menu
- automatic backup function

Layout Editor
- full SMD support
- support of Blind and Buried vias
- rotation of objects in arbitrary angles (0.1-degree steps)
- components can be locked against moving
- texts can be placed in any orientation
- dynamic calculation of signal lines while routing the layout
- magnetic-pads function
- tracks can be drawn 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
- Differential Pair routing
- automatic creation of meanders for length compensation of signals
- user-definable, free programmable User Language to generate data for mounting machines, test equipments, milling machines or any other data format
- output of manufacturing data for pen plotters, photo plotters and drilling machines with the CAM Processor

Schematic Editor
- up to 999 sheets per schematic
- icon preview for sheets
- sorting sheets with Drag&Drop
- cross references for nets
- automatic generation of contact cross references

- simple copying of parts
- replace function for parts without loss of consistency between schematic and layout
- 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)

Autorouter Module

- fully integrated into basic program
- uses the layout's Design Rules
- change between manual and automatic routing at any time
- basic engine for the Follow-me router, a tool that supports you in manual routing; the trace of a selected signal will be calculated automatically
- 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)
- full support of Blind and Buried vias
- takes into consideration various net classes

Standard Edition

The following restrictions apply to the Standard Edition:

- 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, Route2, Route3, Route14, Route15, Bottom).
- The Schematic can have a maximum of 99 sheets.

Freemium Edition

The Freemium Edition is a *Free Premium*, which is available only after registration on *http://www.element-14.com/eagle-freemium* and has the following limitations:

- The board area is restricted to 100 x 80 mm (about 3.9 x 3.2 inches), which corresponds to half of a Eurocard.
- Only 4 signal layers can be used (Top, Route2, Route15, Bottom).

◆ A schematic can consist of a maximum number of 4 sheets.

◆ The Freemium license is limited to one single user and computer, and requires an active connection to the Internet in order to work.

◆ The license expires 30 days after installation.

Light Edition

The following restrictions apply to the EAGLE Light Edition:

◆ 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.

◆ Merging of different projects is not supported.

Larger Layout and Schematic files can be printed with the *smaller* editions. The CAM processor can generate manufacturing data as well.

> *It is not possible to combine modules of different editions!*
> *The Light Edition is available as Freeware for testing, evaluation, and non-commercial use.*

3 INSTALLATION

Windows

EAGLE for Windows is available as a self-extracting archive, which is named, for example, *eagle-win-6.0.0.exe*. Double-click this file and the *WinZip Self-Extractor window* will appear. The installation routine starts with extracting the files from the archive. Click the *Setup* button to begin.

The version number in the file name may, depending on the current version, differ.

Now the actual installation starts. Follow the instructions step by step.

At the end of this process you will be asked how you want to license EAGLE:

Use License file

This implies that you have already bought an EAGLE license. In this case the following dialog asks you for the path to your license file "*serialnumber*".*key* and for your personal installation code which you have got from CadSoft.

Use Freemium Code

If you registered at *http://www.element-14.com/eagle-freemium* and got a Freemium code, use this option.

Run as Freeware

EAGLE will be licensed with the limits of the Light Edition. It's possible to upgrade to a purchased license without reinstallation at any time.

Don't license now

postpones the licensing on the first EAGLE start.

The Windows Start menu now contains in the *Programs* section the new entry *EAGLE Layout Editor 6.0.0*. Tutorial and manual in pdf file format are also part of the installation.

Linux

EAGLE for Linux is available as self-extracting shell script with set-up dialog. You may install the program as *Root/Administrator* or as a normal *User*.

Initiate the installation process in a file manager with a double-click on the shell script file *eagle-lin-6.0.0.run*.

Please check, whether the file is marked as *executable* before. If this is not the case, you could, for example, use the *chmod* command in the console to change its attributes.

You can start the shell script form the console, as well, by typing:

```
/path_to_file/eagle-lin-6.0.0.run
```

The version number in the file name may differ, depending on the currently available EAGLE version.

The installation routine starts with extracting the files from the archive. Then the actual installation process starts. Follow the instructions step by step.

At the end you will be asked how you want to license EAGLE:

Use License file

This implies that you have already bought an EAGLE license. In this case the following dialog asks you for the path to your license file *"serialnumber".key* and for your personal installation code which you have got from CadSoft.

Use Freemium Code

If you registered at *http://www.element-14.com/eagle-freemium* and got a Freemium code, use this option.

Run as Freeware

EAGLE will be licensed with the limits of the Light Edition. It's possible to upgrade to a purchased license without reinstallation at any time.

Don't license now

postpones the licensing on the first EAGLE start.

To start EAGLE, execute *bin/eagle* in the installation directory.

Tutorial and manual in pdf file format are also part of the installation. They can be found in the *doc* directory of the EAGLE installation.

Mac OS X

In order to start the installation process double-click onto the icon that represents the EAGLE archive which is named, for example, *eagle-mac-6.0.0.zip*. The archive will be extracted automatically. A new folder icon named *eagle-mac-6.0.0.pkg* will appear on the Desktop now. Click it with the mouse and the installation process starts.

The version number in the file name may differ, depending on the current EAGLE version.

Follow the instructions step by step.

Confirm the EAGLE Software License Agreement by clicking *Continue*, if you agree with it. If not, you can abort the installation by clicking the *Go back* button.

The next step sets up the EAGLE destination folder and possibly a destination volume. By clicking *Continue* you accept the suggested directory. Click the *Choose* button and you may select any other location for the EAGLE folder. After confirming the installation path the files are copied into the EAGLE folder.

After finishing the installation you are able to start EAGLE by clicking the *EAGLE* icon in the *Finder's Applications/EAGLE* entry.

While starting EAGLE for the first time, you will be asked how to license EAGLE:

Use License file
This implies that you have already bought an EAGLE license. In this case the following dialog asks for the path to your personal license file *"serialnumber".key* and for your personal installation code you have got from CadSoft.

Use Freemium Code
If you registered at *http://www.element-14.com/eagle-freemium* and got a Freemium code, use this option.

Run as Freeware
EAGLE will be licensed with the limits of the Light Edition. It's possible to upgrade to a purchased license without reinstallation at any time.

Tutorial and manual in pdf file format are part of the installation and are located in the *doc* folder of the EAGLE installation.

In case you want to uninstall EAGLE from your computer, simply delete the *EAGLE* entry in the the *Finder's Applications* folder.

4 LANGUAGE SETTINGS

EAGLE decides due to the operating systems' language which language to use. If the systems' language, for example, is set to German, EAGLE will use German language. In case you don't like the automatically selected language, you have the following possibilities to change it.

Windows

EAGLE takes care on a variable named *LANG*. For changing it go to the Windows Control Panel where you can define environment variables. Set up a variable named LANG. For English language the value is typically set to *en_US* or *en_GB. F*or German language the value should be set to *de_DE, de_CH*, or *de_AT*.

In case you would like to use a batch file to start EAGLE, it could look like this:

```
SET LANG=en_GB
cd C:\Program files\eagle-6.x.x
start bin\eagle.exe
```

This is of use, if there are other applications that react on the *LANG* variable. The batch affects EAGLE, only.

Linux and Mac OS X

The same as described for the Windows EAGLE can be done for Linux and Mac OS X. There you have to define the variable with the systems' EXPORT command.

You could also use a script file in order to start EAGLE:

```
LANG=en_US
/home/user/eagle-6.x.x/bin/eagle
```

EAGLE is planned to support more and more languages. Please refer to the file *UPDATE.txt* for information about currently supported languages.

5 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*.

The following keys are assigned by default:

Alt+BS	UNDO	undo action
Shift+Alt+BS	REDO	redo action
F2	WINDOW;	redraw window content
Alt+ F2	WINDOW FIT;	zoom to fit
F3	WINDOW 2	zoom in with factor 2
F4	WINDOW 0.5	zoom out with factor 0.5
F5	WINDOW (@);	center cursor position
F6	GRID;	display/hide grid lines
F7	MOVE	activate MOVE
F8	SPLIT	activate SPLIT
F9	UNDO	undo action
F10	REDO	redo action

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*.

6   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.

As an alternative you can click on an object with the right mouse button and select one of the object-specific commands from the context menu.

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 ⇒ means, that a menu selection is to be made. For example

> ⇒ *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.

In this context Mac OS-X users have to use the Cmd key instead of Ctrl!

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 $\oplus$, then • on the object

◆ Typing MOVE in the command line, followed by the *Enter* key, then clicking the object

◆ Typing MOVE and the object name in the command line, followed by the Enter key, for example: MOVE IC1

◆ Opening the context menu with a right mouse click on the object, then selecting *Move*

◆ Pressing the function key *F7* which is assigned to the MOVE command

◆ Selecting the menu item ⇒ *Edit/Move*

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

• MOVE

means: click the MOVE icon $\oplus$

Use of Key Combinations

The + 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.

The Context menu

You may also click with the right mouse button on an object to open the context menu that offers object-specific commands.

Furthermore the context menu contains the entry *Properties*, that allows you to change the object's properties, if applicable. This window can be opened with the INFO command, as well.

Command and Parameter Input via the Command Line

Actions which need to be terminated with the *Enter* key are symbolized with the character ←. For example

USE ←

means: type USE into the command line 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 ←

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

change width 0.024 ←

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 images 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.
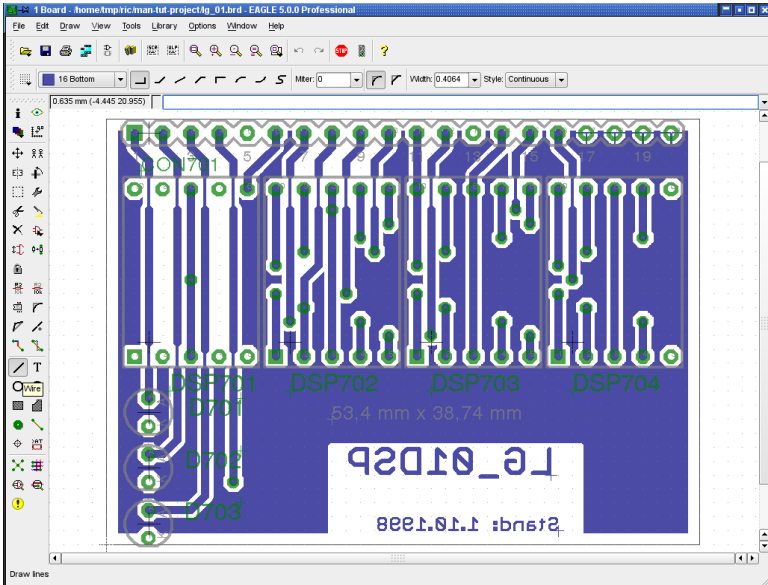
➢ *The Layout Editor window*

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.

The Layout Editor displays the board the same way as if holding the manufactured board in your hands. You are looking through it from above.

Info			**Show**	**Info**			**Show**
Display			**Mark**	**Display**			**Mark**
Move			**Copy**	**Move**			**Copy**
Mirror			**Rotate**	**Mirror**			**Rotate**
Group			**Change**	**Group**			**Change**
Paste				**Paste**			
Delete			**Add**	**Delete**			**Add**
Pinswap			**Replace**	**Pinswap**			**Replace**
Gateswap				**Lock**			
Name			**Value**	**Name**			**Value**
Smash			**Miter**	**Smash**			**Miter**
Split			**Invoke**	**Split**			**Optimize**
Wire			**Text**	**Meander**			
Circle			**Arc**	**Route**			**Ripup**
Rect			**Polygon**	**Wire**			**Text**
Bus			**Net**	**Circle**			**Arc**
Junction			**Label**	**Rect**			**Polygon**
Attribute			**Dimension**	**Via**			**Signal**
Erc			**Errors**	**Hole**			**Attribute**
				Dimension			
				Ratsnest			**Auto**
				Erc			**Drc**
				Errors			

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

7 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. A 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 ($\Rightarrow$ New/Project).

The tree view allows a quick survey of EAGLE's libraries. If you expand one of the library entries in this branch, for example *40xx.lbr*, the content of the library will be shown.

Select a Device or Package entry to display the preview of this object on the right.
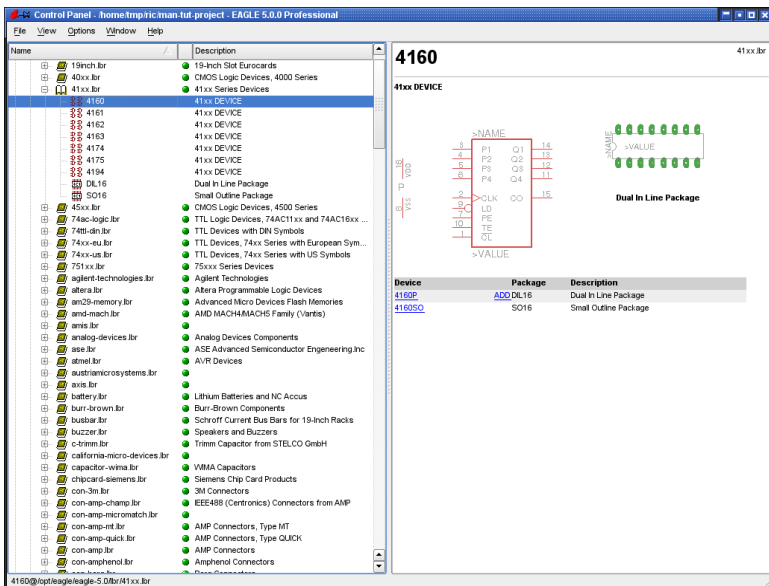
> *Control Panel: Preview of the library contents*

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, Rename etc.

The cursor keys allow you to navigate efficiently within the tree view. The *cursor-right* key expands a branch. *Cursor-left* jumps back to the superior node. Hit *Cursor-left* again to close the branch. *Cursor-up/-down* leads you to the previous/next entry.

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	*.*

> *EAGLE uses only lower case letters for 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, Mac) or *eaglerc.usr* (Windows).

Before starting the following examples, we want to copy the files *demo1.sch*, *demo2.sch*, and *demo2.brd* from the tutorial directory into the directory *MyProject*. Press the *Ctrl* key, click the desired file and drag it to the *MyProject* entry. Release the mouse button now. The *Ctrl* key effectuates that the files will be copied, otherwise they would be moved to the target directory.

EAGLE for Mac behaves different here:

Clicking onto a file entry with pressed *Ctrl* key opens a context menu. Select the *Copy* option and choose the target folder in the file dialog then.

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.

8 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. If *MyProject* does not yet exist, please create it as explained in the previous chapter.

Now load the *demo2.brd* file. You can do this either by • • the entry *demo2.brd*, or by selecting the file from the menu ⇒ *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.

If you want to deactivate this feature, set *Mouse wheel zoom = 0* in the *Options/User Interface* menu.

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.

It's possible to define a certain area of the drawing as a so-called alias that can be used as a parameter with the WINDOW command. Open the context menu of the *Window select* icon with a right-click on the icon. Select *New...* and name the alias.

> *To pan the current drawing area 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.*

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).

WINDOW LAST restores the previously displayed window area.

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 ←

9 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* (⇒ *File/Open/Board*) in the Control Panel or in the Layout Editor window and click in the command toolbar on the icon for the

DISPLAY command 🔳. 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. A click onto *Apply* immediately shows the changes in layer selection in the editor window. With a click onto *New* you can create a new layer (above layer number 100). Click onto *Delete* in order to remove layers, provided the layer does no longer contain objects.

> *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.

To restore the recent layer selection, use DISPLAY LAST ←.

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

10 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 by clicking on the GRID icon ▦ in the parameter

toolbar.

Clicking with the right mouse button on the GRID icon opens a popup menu. It contains the entry *Last* which switches to the grid used before. With *New..* you are allowed to define so-called aliases representing certain grid settings. The alias name can be used as a parameter with the GRID command. Quick switches from one grid setting to another are possible now.

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 (⇒ *Edit/Design Rules...*) one can use values in mil or in Millimetres (1 mil = 1/1000 inch). The default unit is

mil.

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

```
0.2mm
```

Inch - Mil – Millimetre 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.

11 WIRES, CIRCLES, ARCS, RECTANGLES, TEXT

Wires, circles, arcs, rectangles, and text are created with the WIRE ✏️, CIRCLE ⭕, ARC ↷, RECTANGLE ▦ and TEXT T 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.

➤ *Parameters of the WIRE command*

Click the WIRE icon in the command toolbar. All parameters for this

command can be set up in the parameter toolbar. Select layer 94, *Symbols*, from the layer selection box. Draw a rectangular line in this layer now:

Define the starting point by a click on the left hand mouse button. Move the cursor slightly up to the right and select the various bend modes in the parameter toolbar. Among some diagonal and orthogonal modes you will find some to draw 90° arcs and free definable arcs.

If the connection is displayed in a square angle, press the left mouse button to fix its position. Move the cursor to the starting point and • • to drop the line. Now you should see a rectangular outline.

As observed before, various wire bend styles can be used by clicking the wire bend icons in the parameter toolbar. More efficient is using the right mouse button while drawing a wire. Activate the WIRE command, start drawing the line, and click the right mouse button a few times to go through the different wire bend modes.

However, in the Schematic Editor, the three arc modes can be selected only in the parameter toolbar, not with the right mouse, as it is in the Layout Editor.

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

> *Do not use the WIRE command to draw net or bus lines in the schematic. Use NET or BUS instead!*
>
> *Use ROUTE and not WIRE to dissolve airwires to tracks in the Layout editor. Otherwise EAGLE would not set vias automatically when changing signal layers.*

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 in layer 20, *Dimension*. 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 ⚡ 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.

Wire properties like Width, Style or Layer can be altered through the Properties entry of the context menu. Select the wire in question by a right mouse click to open the context menu.

As an alternative use the INFO command to open die properties dialog.

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.

This can be done with groups, too:

> • GROUP and draw a frame around all objects, then
> • CHANGE and in the popup menu • Layer, then select the layer in the Layer list, now hold the *Ctrl* key pressed and click with the right mouse button into the group

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.

UNDO LIST

in the command line displays a list of all the commands and actions you executed before. It is possible to go navigate through the list and specifically undo/redo actions.

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 Millimetre:

    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

T To activate the TEXT command, used for placing text,

- TEXT

Now a text input window opens where you can type the text. Then • *OK*. 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 or the Stop icon. For placing a different text, type the text in the command line and terminate it with the *Enter* key. The text shows up at the mouse cursor and you can place it.

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

    'This is a text'

> *Selecting the Properties entry of the context menu allows you to alter properties like Size, Ratio, Font, Alignment, Layer, and the* text *itself. To open the context menu click with the right mouse button on the text's origin.*

Alternative methods for changing text properties:

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
- at the point of origin of the text

Then edit the text and • *OK*.

Using

- CHANGE
- RATIO

you are able to change the line width in a text in relation to the height of the Vector font (not Proportional or Fixed font).

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.

Apart from the predefined placeholder texts, EAGLE interprets text that begins with the '>' character as a user-definable, component-specific or global attribute. For more, see the help pages of the ATTRIBUTE command.

12 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 components 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 it's contents will be shown. Select an entry and you will see the corresponding preview on the right.

The cursor keys allow you to navigate efficiently within the Library list. The *cursor-right* key expands the Library or Device entry. *Cursor-left* jumps back to the superior entry. Hit *Cursor-left* a second time to close it. *Cursor-up/down* leads you to the previous/next entry.

If you type in any letter, the first entry beginning with this letter will be selected. Typing more letters swiftly one after the other, for example *con,* the first entry beginning with *con* will be selected.

You can enter one 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.*

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.

The quickest way to add a component you already placed in the drawing is to right-click the ADD icon. A popup menu opens that contains a list of the previously placed components. Select one of them and place it.

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* 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 grey font assigned to gate A indicates that the gate has been used; those printed in black font 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 com-

mand on the new sheet and type in the part'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 it.*

13 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 0.025 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 with document filed 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 symbol 🛑 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.

> *You could also use the FRAME command (⇒ Draw/Frame..) to draw a frame by yourself. See the help function about FRAME for details.*

Adding and Changing Text

You can add lines, text and other objects to predefined frames and text fields in the library.

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.

It is also possible to type in the value in combination with its unit. If you prefer, for example, a value in Millimetres (without changing the grid), type in:

```
CHANGE SIZE 3.5mm
```

Depending on the situation it can be useful to change the alignment of text. There are nine different possibilities consisting of combinations of left, right, center, top and bottom.

In case the text is already placed in the drawing you can use the CHANGE command (`CHANGE ALIGN`) or the *Properties* entry of the context menu. If

you are just placing the text you can choose the alignment in the *Align* box of the TEXT command's parameter toolbar.

> *Use dots for decimals! The current grid setting determines the unit, unless you specify it in the command line!*
>
> *Property changes can be made in the context menu's Properties entry, as well.*

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

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

DATE: shows 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 lets 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 Devices' 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.

Use the NET command, not WIRE to draw net connections!

Keep in mind:
A right mouse click on an object opens the context menu. It offers all commands available for the object.

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 drawn continuously. Activate the SHOW command and click on a net segment: the whole net is displayed highlighted now.

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

Nets with the same name are affiliated with each other!

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 in any location. • `LABEL`, locate the cursor on the net MCLR/PGM and •.

The name of the net is attached to the cursor and a line points to the nearest part of the related net. Place the text at 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.

If a schematic consists of more than one sheet (not possible in the Light Edition), it is possible to set the *XREF* option for labels. This can be used for generating a cross reference to the referring net on the next sheet. This option works only in combination with drawing frames that are drawn with the FRAME command. More details about this can be found in the help function of FRAME and LABEL.

The DELETE Command

You can delete objects with this command. If it is applied to nets, wires or buses, 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.

To delete whole groups, use *Ctrl* + DELETE and a right mouse click into the group.

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* (⇒ *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 objects 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 IC2 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 or pressing the *Esc* key. Now the objects are no longer highlighted.

To show a certain object, • SHOW and type the name (for example RA4 ←) in the command line. Hit the *Enter* key then.

You can subsequently type other names without the need to reactivate the SHOW command.

      SHOW RA4 ←
      RA3 ←
      RA2 ←

This way you can mark one net after the other.

The SHOW command works also with placeholders. Type

      SHOW RA* ←

to get all nets beginning with RA highlighted at the same time.

In case you wish to highlight several nets at the same time, click with the left mouse button, for example on net RA2. Now press the *Ctrl* key and click on nets RA3 and RA4. All three nets are highlighted now.

Click again with pressed *Ctrl* key on net RA3 and the net won't be highlighted anymore. *Ctrl* inverts the highlight.

If the SHOW command doesn't find the object, the *SHOW* window pops up and marks its *Sheet* column with a ' - ' character. This tells us that the object doesn't exist on any of the sheets.

The MOVE Command

⊕ No electrical connection will be generated if you MOVE a net line over a pin.

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 (net and connected pins are highlighted), as mentioned before. Additionally you can export a net list 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. 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 objects 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.
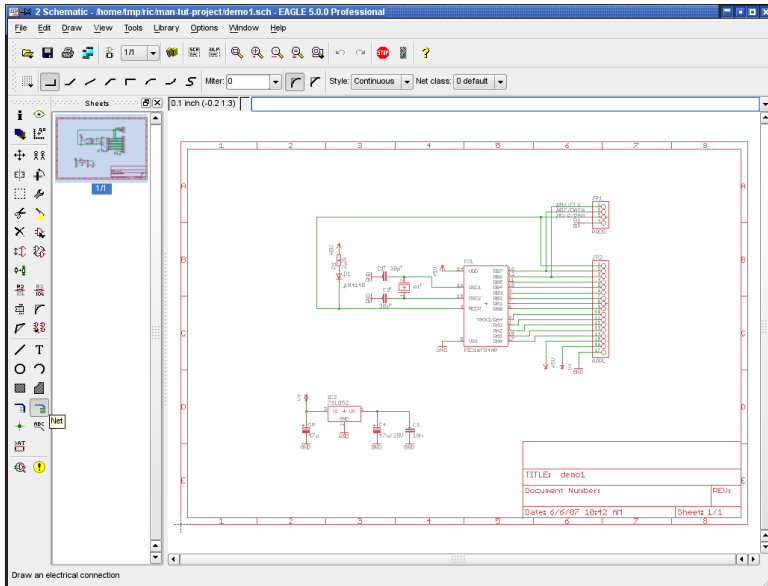
> ➤ *The Schematic demo1.sch*

The SMASH Command

🔲 You will notice that when you rotate an element from the horizontal to the vertical position, the reference designator and value text 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 such a detached (smashed) text, EAGLE draws a line from the text to the part's origin to show where the text belongs to.

To activate the command

• SMASH

Locate the cursor on the diode symbol and •.

You could also type in the part's name in the command line and hit the enter key.

There is still another method to execute the SMASH command.:

Click onto the part with the right mouse button to activate the context menu. Select the entry *Properties* and activate or deactivate the *Smashed* option.

After separating the text from the symbol, • MOVE, move the cursor to the designator D1 of the diode, and •. The text selection point is marked as a cross and resides, depending on text's 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, too. Therefore activate the SMASH command, press the *Ctrl* key, and click with the right mouse button into the drawing.

If you keep the *Shift* key pressed while you are clicking with the left mouse button onto a part, all texts will return to their original locations and will be fixed to the part as it was before SMASHing them (unsmash).

For groups you have to keep the *Shift* and the *Ctrl* key pressed. The command will be executed immediately. If you press the *Shift* key only, the context menu opens and you have to select *Smash:Group* there.

The VALUE Command

EAGLE allows you to define or to change the value for parts 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.

If you want to define the same value for several components, click onto the icon with the right mouse button. A list of recently used values shows up. Select one of the entries and then click successively onto the components to apply the value.

You can use the NAME command now 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 listed in the ERC window. To start the Electrical Rule Check click the ERC icon in the command toolbar.

The ERC finds two warnings in our sample:
```
POWER Pin IC1 VSS connected to GND
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. Click on one of the message entries and EAGLE will show where the reason for the problem is located in the schematic.

Both warnings don't cause any problems and could be avoided by changing

the names of the power pins in the library definition. But you could also *Approve* these warnings. The messages are now shown in the *Approved* branch, no longer in the *Warnings* branch of the menu.

> *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 object 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 command toolbar and move the cursor over the blue bus line, one grid line over the pin IC7-14. The net connection must begin on the bus line, if you want to name it in a convenient way. • onto the bus to set the

starting point of the net. Now a popup menu will appear which contains the net names deviated from the bus name. • the entry EN to select the net name EN. Move the cursor to IC7-14 and use the right mouse button to change the bend style until it is drawn like the other net lines in this area. • the pin's connection point to fix 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.

If you want to move the bus the already drawn nets won't follow. To move them together you have to use GROUP and MOVE.

> *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 is shown in the status bar.*

14 AUTOMATIC 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, for example naming components, nets, and so on. Other actions are possible only in the schematic, for example adding components. In such a case EAGLE prevents this operation in the board and prompts you to do this in the Schematic Editor instead.

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 ought to start the ERC if any differences are found. The results are shown in the ERC window. The *Consistency errors* branch lists the differences between schematic and layout. Using this information you are able to make them step by step consistent again.

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.

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 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.
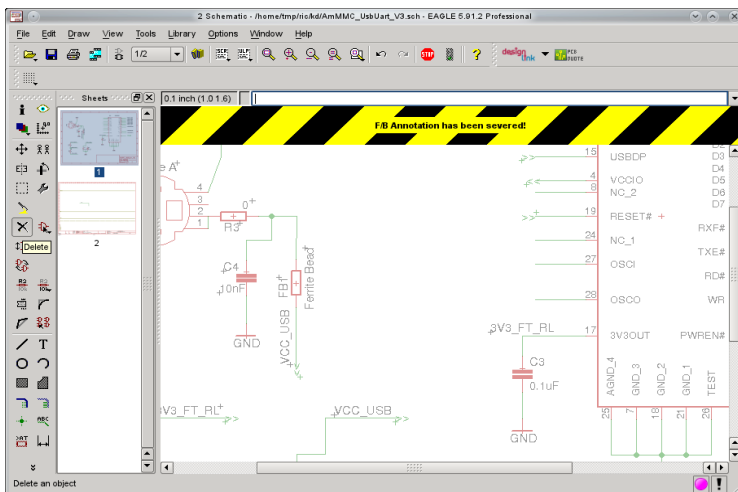
➢ *Warning in yellow and black: Forward&Back Annotation severed!*

As soon as you close one of the Editor windows, for example by clicking the X button of the window, EAGLE will prompt a warning that Forward&Back Annotation has been severed.

If you did this by intention, click into the warning in order to hide it.

15 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 Editor. If you have the Schematic Editor 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.

> *The Layout Editor displays the board in top view. Imagine you are holding your printed circuit board in your hands and are looking through it from above. This is the reason why, for example, the texts on the bottom side are displayed mirrored.*

Designing a Board without a Schematic

Open a new file (⇒ *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.

• • will terminate the WIRE command. The board outlines are now defined.

Using the MOVE command ⊕, the edges can be moved, or use UNDO (F9) or UNDO LIST and REDO (F10) 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*.

- ● 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 the MIRROR icon ⟦▣⟧ in the command toolbar and ● on the

package or type the name of the part in the command line.

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).

As soon as a component is positioned and you don't want to have it moved again, you can fix it with the LOCK command. For clarification the origin cross of a locked component is displayed like an 'x'.

Shift + LOCK and a mouse click on the component releases it again.

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.

It's also possible to change the object's name in the command line, for example:

        NAME R22 R2 ←

In this case R22 is renamed R2.

Providing Values

To assign values to a component:

- 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 airwires (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 (*F9*) and REDO (*F10*) function under the control of the Forward&Back Annotation, too.

Defining Net Classes

The CLASS command (⇒ *Edit/Net classes..*) allows you to define different properties for nets:

Width defines the minimum wire width,

Clearance defines the minimum distance to signals of other net classes,

Drill defines the minimum drill diameter of vias.

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/follow-me router uses these values for routing, too.

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

If there are values defined in the Design Rules and for Net Classes, the higher one of both will be valid.

In the case you have to use different values for the clearances between each single net class, you can set them in the *Clearance Matrix* which can be accessed with the >> button in the Net Classes window.

As an example various classes are used in the board file *hexapodu.brd*.

Creating a Board from a Schematic

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

A board containing parts that have already names and values and whose pads or SMDs are connected through airwires.

Generating a Board File

Load the file *demo1.sch* and click the BOARD icon [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.

As a first step we recommend to draw the board contour. It is made up of wires in the layer 20, *Dimension*.

• WIRE and choose a fine wire width in the parameter toolbar. Now begin to draw the board contour. Start at the coordinates origin (0 0) with a left mouse click. Move the mouse cursor and draw the outline. Each mouse click fixes the line segment that follows the mouse cursor. The board contour should end at the starting point.

Please keep in mind, that the Freeware and the Freemium license is limitted to a maximum board size of 100mm x 80mm. You should not draw the board outlines beyond this limit.

Component Placement

Click on the Window-Fit icon [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 a component is to select it by its name. • MOVE [icon]

and type in the command line.

    JP1 ←

This part is now attached to the mouse cursor.

Do you want to have a component fixed at its location? Then use the LOCK command.

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.).

> *After generating a board file with the BOARD command EAGLE arranges all packages on the left side of the board outline in the negative coordinates area. In the Light Edition for example, you may drop elements within the limits of about 3.9 x 3.2 inch. To route the layout manually or to use the Autorouter or the Follow-me router 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 [icon] in the command toolbar. Choose a finer Routing Grid (default 50 mil), if necessary and • *OK*.

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.

> *If you want to have a board routed single sided, draw a restricted area for the top side over the whole board in layer 41, tRestrict, with the RECT command.*

Routing Manually

The ROUTE command [icon] transforms the airwires into routed tracks.

- ROUTE in the command toolbar.
- starting point of an airwire or type in the signal's name in the command line. The starting point of the track is next to the current mouse cursor position.

You can set further parameters, like width or layer, in the parameter toolbar. The selected width is valid until you decide to choose a new value.

If an airwire begins at an already routed track which has another width than the currently chosen, and you would like to lay out the new track with this width, keep the *Shift* key pressed while clicking onto the airwire. The new wire adopts the width of the already present track.

> *All values are related to the currently selected GRID unit.*

Activate ROUTE, • an airwire, move the cursor to route the signal, • to fix the current segment, and move the mouse again to change the routing direction

for the following segment. • • to fix the last segment and end the routing operation for the signal at the current mouse position. 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*). Two *follow-me* modes (wire_bend 8 and 9) are available in the parameter toolbar only. They are explained in the next section.

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. EAGLE sets the necessary via automatically.

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.

In case you are approaching a pad or SMD which belongs to the current signal, and the length of the calculated airwire is getting shorter than the so-called *snap length*, the wire will be snapped into the center of the pad or SMD (magnetic-pads function).

The value for *snap length* is defined in the *Options/Set/Misc* menu.

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.

Follow-me Router

The ROUTE command knows two operating modes, wire bend 8 and 9, that help you to route signals. Simply select an airwire with the mouse and let the Follow-me router calculate the trace of the connection. The position of the mouse cursor influences the process of routing. The so-called Follow-me router works either in *partial* mode ⌐ or in *full* mode ⌐ , depending on the icon you click on in the parameter toolbar.

> *The Follow-me router is supported only, if the Autorouer module is available.*

After selecting an airwire in *partial* mode, the track will be calculated from the current mouse position to the nearer end of the airwire and displayed on

the screen. Change the mouse cursor and the Follow-me router beginns to calculate and display the new trace. The connection follows the mouse. If you like the result, fix it with a mouse click. Go on with the next section now. Move the mouse cursor to a new positon and the Follow-me router calculates the further trace of the connection.

In *full* mode the Follow-me router calculates the whole connection. Click onto an airwire and the trace will be calculated beginning with the current mouse position into both directions of the airwire.

The Follow-me router respects the standards given in the Design Rules and the net classes. Settings that concern the routing strategy are defined in the *General* and *Follow-me* tab of the *Autorouter Setup* window. To access this window click onto the AUTO icon ![icon] in the parameter toolbar of the ROUTE command. This icon is not displayed until clicking on one of the follow-me icons.

The current grid setting of the Layout Editor serves as routing grid. Wire width, drill diameter of vias, and the layer the wire starts in at the mouse cursor position can be set in the parameter toolbar.

Additional information concerning the Follow-me router is found in the help function of the ROUTE command and in the EAGLE manual.

> *The Light Edition does not support Blind and Buried vias due to their limitation of the number of signal layers.*

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 🔧 (also PACKAGE) or REPLACE ⬍ . In *demo3.brd* the package of IC1 has been replaced by a SMD package,

◆ modify the Design Rules (for example, Restring settings),

◆ use SMASH 🖳 to detach texts from components in order to

   MOVE ✛, ROTATE 🔁, or CHANGE 🔧 them,

◆ MITER 𝐫 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 deselect with your mouse the layer 21, *tPlace,* which shows the top side silkscreen of the board, and 23, *tOrigins*, 25, *tNames*, 27, *tValues*, and 51, *tDocu*.

To execute this change • *Apply*, or • *OK* in order to close the display menu.

It is possible to define so-called aliases. This allows you to switch quick and comfortably from one layer combination to another. An alias can be defined with the popup menu of the DISPLAY icon. See also the help pages of the DISPLAY command.

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 properties 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.

Alternatively keep the left mouse button clicked and drag a rectangle to mark the group's area.

The command `GROUP ALL` selects all objects, if the respective layers are displayed and if components are not locked by the LOCK command.

Holding down the *Shift* key while clicking the mouse the first time for the group selection, an already existing group, if any, will be extended by the selected objects.

Ctrl + GROUP allows you to toggle the group membership for single objects.

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

Next select the MOVE command, press the *Ctrl* key, and use the <u>right</u> mouse button to attach the group to the cursor. Now you can move all the objects simultaneously. Click the right mouse button to rotate the group by a given angle counter-clockwise. If you additionally press the *Shift* key, the group will be rotated clockwise. The angle can be specified directly in the parameter toolbar's *Angle* combo box. The left mouse button finally fixes the group again.

After a group has been defined with the GROUP command, the properties 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 <u>right</u> mouse button somewhere in the editor window while the *Ctrl* key is pressed.

You can reverse the action with the UNDO command.

The SPLIT Command

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

- SPLIT in the command toolbar
- on a wire segment

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. Click the right mouse button several times to explore the various bend styles.

A further click with the left mouse button fixes the wire segment. The following wire segment is already attached to the mouse cursor now. It's very easy to re-route a track with this method. The hitherto existing track can be erased, for example, with MOVE and RIPUP.

The Esc key or clicking the Stop icon ends the command.

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. Now both signals are displayed highlighted. Click the *Go* icon to execute the command now.

You could also add a semicolon in the command line to have the command executed immediately

        ! GNC VCC ; ←

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.

If you are looking for a small object, it might happen that you can't recognize the object although it is displayed with highlighted colors. In this case use the SHOW comand with the option @:

        SHOW @ IC1 ←

will draw a frame around the object and show you a pointer towards it. In case there is a consistent schematic drawing loaded, the same will happen to the corresponding object there.

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*).

        UNDO LIST

in the command line opens a list of the recently executed actions. You can go back and forth a certain number of steps there.

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 Light Edition!

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.

The contour of a polygon in the outline mode is displayed as a dotted line.

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 ←

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.

16 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 layers 21, *tPlace*, 23, *tOrigins*, 25 *tNames*, 27, *tValues*, and 51, *tDocu*, using the DISPLAY command, so that the components are not shown any more.

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 function and EAGLE manual).

Depending on the way EAGLE has been installed, it might happen that you don't have enough rights to start this job. In this case please contact your system administrator or copy the contents of this folder into a private folder where you have full access rights.

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.*

17 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 *Restring* 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 and starts the check at once.

Now the status bar shows the number of errors. An *DRC Errors* window will

open automatically showing the error list.

Selecting one of the entries, EAGLE shows where the error is located in the board. Errors are marked with so-called error polygons.

In some cases maybe you wish to tolerate an error. Use the *Approve* button for this. The error polygon of the approved error will be deleted then.

As soon as you have corrected an error in the layout, you can mark it as *Processed*. If you want to delete it, click the *Clear* button.

> *Objects that cannot be deleted in your board might be error polygons drawn by the DRC. You can erase them with ERRORS CLEAR.*

18 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

> *In case your EAGLE license has a single Schematic Editor or a single Layout Editor module, you nevertheless are able to define complete components in the Libraries.*
>
> *It has proved to be useful to create new components in your own library files. Thus, the library files supplied by CadSoft remain unchanged and can be easily replaced, for example, for an update. Your custom components will remain as they are in each case.*

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 *auto*. 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 objects 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*.

T 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.

> *We recommend to write these texts in vector font. So you can be sure that it looks exactly the same on the printed board as it looks in the Layout Editor.*

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 <u>right</u> mouse button while the *Ctrl* key is pressed.

Example: Use GROUP to define a group that contains both pads, then select CHANGE and SHAPE/SQUARE. Press the *Ctrl* key and 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 *Ctrl* key plus the right mouse button. See help function for further details.

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

placed.

T 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* with the 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. Layer 97 *Info* can be used for further information, for example, descriptive text.

Click onto *Description* at the bottom of the symbol editor if you would like to have a descriptive text for your symbol.

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.
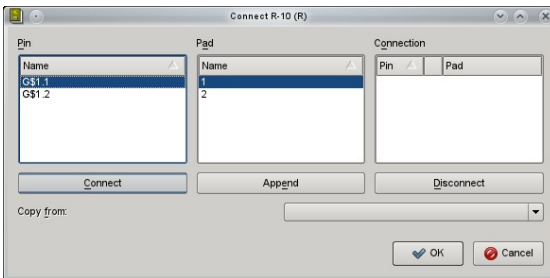
➢ *The Connect window*

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 this command and closes the window.

You can enter descriptive text for your Device 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.
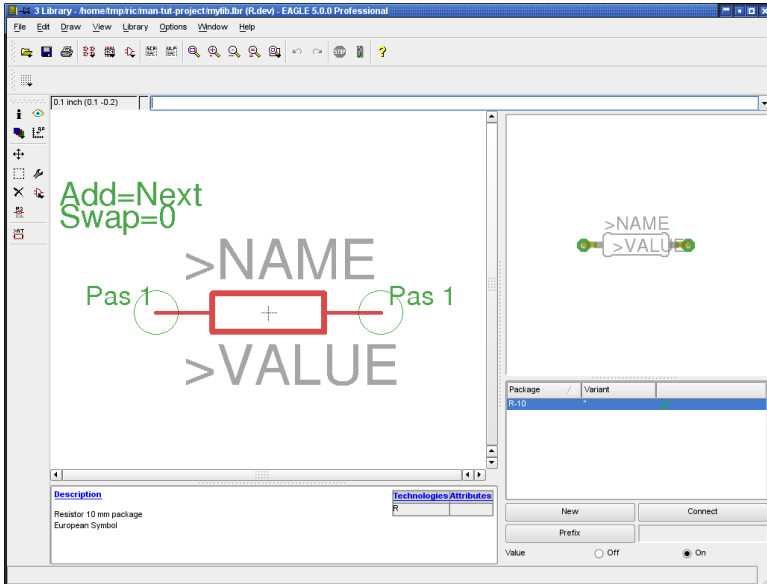
➢ *The Device Editor*

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.*

19 OUTPUT OF DRAWINGS AND MANUFACTURING DATA

EAGLE can output drawings, for example for documentation purposes, using the PRINT command which can be found in the *File* menus of the Schematic

or Layout Editor or initiated by clicking the printer icon 🖨 in the Action

toolbar. The printer drivers of your operating system are used for the printout. The drawing shown in the currently active Editor window will be printed. The layers that are actually visible in the Editor window (selectable with DISPLAY) will be visible on the printout.

The PRINT command allows generating PDF files, too.

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 printer 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*.

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

EAGLE can generate PDF files. Therefore select the *PDF..* button in the PRINT dialog and enter a file name for the PDF output file.

To generate image files use the EXPORT command (⇒ *File/Export..*) 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 *gerb274x.cam*, which can be found in the default subdirectory for CAM jobs, automates the output of the most common Extended Gerber data for double sided boards.

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 *gerb274x.cam* in the Control Panel's tree view (*CAM Jobs*), or by clicking the CAM Processor icon in the Layout Editor window and selecting *gerb274x.cam* in the file dialog (⇒ *File/Open/Job*).

In case you have started the CAM processor from the Control Panel, load the board file *demo3.brd*: ⇒ *File/Open/Board* and • • *demo3.brd*

Click the button *Process Job*. Now all the necessary files will be written into the directory where the board file is located.

The files have the following meanings:

demo3.cmp	Component side
demo3.sol	Solder side
demo3.plc	Silkscreen for component side
demo3.stc	Soldering mask for the component side
demo3.sts	Soldering mask for the solder side
demo3.gpi	Information file, not relevant here

The first five 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 and in the EAGLE manual.

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 board manufacturer (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.

20 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.

User Language Programs are started with the RUN icon 🔳 in the action toolbar.

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 to generate a Script file of this library. This is a good example to learn about the EAGLE command syntax.