

# **EAGLE 3.5**

**Schematic - Layout - Autorouter**

## **Tutorial**

**Copyright © 1999 CadSoft Computer, Inc.  
[www.cadsoftusa.com](http://www.cadsoftusa.com)**

Windows is a registered trademark of Microsoft Corporation

<b>1 What to expect from this Manual</b>	<b>6</b>
<b>2 System Requirements</b>	<b>6</b>
<b>3 Features of EAGLE (Professional Version)</b>	<b>6</b>
<b>General</b>	<b>6</b>
<b>Layout Editor</b>	<b>7</b>
<b>Schematic Module</b>	<b>7</b>
<b>Autorouter Module</b>	<b>7</b>
<b>4 EAGLE Freeware Limitations</b>	<b>7</b>
<b>5 How to Use the EAGLE Tutorial</b>	<b>7</b>
<b>6 Installation and Program Start</b>	<b>8</b>
<b>Windows 95/98/NT</b>	<b>8</b>
<b>Other Operating Systems</b>	<b>8</b>
<b>7 Individual EAGLE Setup</b>	<b>9</b>
<b>8 The Concept of the EAGLE User Interface</b>	<b>9</b>
<b>9 Typographic Conventions</b>	<b>10</b>
<b>Mouse Click</b>	<b>10</b>
<b>Several Input Alternatives</b>	<b>10</b>
<b>Use of Key Combinations</b>	<b>13</b>
<b>Command and Parameter Input via the Command Line</b>	<b>13</b>
<b>10 Control Panel</b>	<b>14</b>
<b>EAGLE Files</b>	<b>15</b>
<b>EAGLE Project File</b>	<b>15</b>
<b>11 Load File and select Monitor Zoom</b>	<b>15</b>
<b>12 Selecting Layers for Display</b>	<b>17</b>
<b>13 Setting up Grid and Unit</b>	<b>17</b>
<b>14 Wires, Circles, Arcs, Rectangles, and Text</b>	<b>17</b>
<b>The WIRE Command</b>	<b>18</b>
<b>Changing Line Width</b>	<b>19</b>
<b>Change Object to another Layer</b>	<b>19</b>
<b>Undo/Redo</b>	<b>19</b>
<b>The CIRCLE Command</b>	<b>20</b>
<b>The ARC Command</b>	<b>20</b>

---

<b>The RECT Command</b>	<b>21</b>
<b>The TEXT Command</b>	<b>21</b>
<b>Special Text Variables</b>	<b>22</b>
<b>15 Using Libraries</b>	<b>22</b>
<b>The USE Command</b>	<b>22</b>
<b>The ADD Command</b>	<b>23</b>
<b>The INVOKE Command</b>	<b>24</b>
<b>16 Drawing a Schematic</b>	<b>25</b>
<b>Grid</b>	<b>25</b>
<b>Adding a Frame to a Schematic</b>	<b>25</b>
<b>Adding and Changing Text</b>	<b>26</b>
<b>Entering a Schematic</b>	<b>28</b>
<b>The BUS Command</b>	<b>28</b>
<b>The DELETE Command</b>	<b>29</b>
<b>The NAME Command</b>	<b>29</b>
<b>The LABEL Command</b>	<b>30</b>
<b>The NET Command</b>	<b>30</b>
<b>The SHOW Command</b>	<b>31</b>
<b>The MOVE Command</b>	<b>31</b>
<b>History Function</b>	<b>31</b>
<b>Completing the Schematic</b>	<b>32</b>
<b>The JUNCTION Command</b>	<b>32</b>
<b>The SMASH Command</b>	<b>32</b>
<b>The VALUE Command</b>	<b>33</b>
<b>The Electrical Rule Check (ERC)</b>	<b>33</b>
<b>17 Function Keys</b>	<b>34</b>
<b>18 Automatic Forward&amp;Back Annotation</b>	<b>34</b>
<b>Generating a Board from a Schematic</b>	<b>34</b>
<b>19 Designing a PC Board</b>	<b>35</b>
<b>Designing a Board without a Schematic</b>	<b>35</b>
<b>Defining Board Shape</b>	<b>35</b>
<b>Placement Grid</b>	<b>36</b>

---

<b>Loading a Library for Using</b>	<b>36</b>
<b>Placing Components</b>	<b>36</b>
<b>Placing SMD Packages</b>	<b>37</b>
<b>Providing Names</b>	<b>37</b>
<b>Providing Values</b>	<b>37</b>
<b>Defining Signals</b>	<b>37</b>
<b>Creating a Board from a Schematic</b>	<b>38</b>
<b>Generating a Board File</b>	<b>38</b>
<b>Component Placement</b>	<b>39</b>
<b>Autorouter: A Brief Example</b>	<b>39</b>
<b>Routing Manually</b>	<b>40</b>
<b>Board Changes</b>	<b>40</b>
<b>Further Usage of the Layout Editor</b>	<b>41</b>
<b>DISPLAY Command</b>	<b>41</b>
<b>MOVE Command</b>	<b>41</b>
<b>GROUP Command</b>	<b>41</b>
<b>SPLIT Command</b>	<b>42</b>
<b>CHANGE Command</b>	<b>42</b>
<b>DELETE Command</b>	<b>43</b>
<b>ROUTE Command</b>	<b>43</b>
<b>RIPUP Command</b>	<b>43</b>
<b>SHOW Command</b>	<b>44</b>
<b>Refresh Screen</b>	<b>44</b>
<b>UNDO/REDO</b>	<b>44</b>
<b>Inner Layers</b>	<b>44</b>
<b>Supply Layers</b>	<b>44</b>
<b>Copper Pouring</b>	<b>45</b>
<b>20 Autorouter</b>	<b>46</b>
<b>21 Design Rule Check</b>	<b>47</b>
<b>22 Libraries</b>	<b>48</b>
<b>23 Output of Drawings and Manufacturing Data</b>	<b>48</b>
<b>Output a Schematic with the PRINT Command</b>	<b>49</b>

<b>Generating Gerber Data with the CAM Processor</b>	<b>50</b>
<b>24 EAGLE User Language - A Universal Output Tool</b>	<b>51</b>
<b>25 Script Files - Flexible Input Interface</b>	<b>51</b>

### **1 What to expect from this Manual**

---

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

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

### **2 System Requirements**

---

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

- an IBM-compatible computer (386 and above), Windows 95/98 or WindowsNT, Linux or DOS
- a harddisk with 30 Mbytes free memory (min.)

Output: printers, plotters, photo-plotters and drill machines, as well graphics files and user-programmableoutput via User Language.

### **3 Features of EAGLE (Professional Version)**

---

#### **General**

- maximum drawing area 64 x 64 inches
- resolution 1/10,000 mm (0.1 microns)
- mm or inch grid
- up to 255 layers, user definable colors
- command files
- C-like User Language
- simple library editing

## Layout Editor

- full SMD support
- full multilayer support (16 signal layers)
- Design Rule Check for board layouts
- copper pouring

## Schematic Module

- up to 99 sheets per schematic
- Online-Forward&Back Annotation between schematic and board
- automatic board generation
- automatic generation of supply signals (for IC's)
- Electrical Rule Check

## Autrouter Module

- fully integrated into basic program
- change between manual and automatic routing at any time
- ripup&retry algorithm
- user-definable strategy (by cost factors)
- routing grid down to 4 mil
- no placement restrictions
- up to 16 signal layers (with user definable preferred directions)
- up to 14 supply layers

## 4 EAGLE Freeware Limitations

---

The following restrictions apply to the EAGLE Light Version, which is available as Freeware:

- The board area is restricted to 4 x 3.2 inches.
- Only two signal layers can be used.
- A schematic can consist of only one single sheet.

## 5 How to Use the EAGLE Tutorial

---

This tutorial covers the use of the EAGLE Schematic Editor, Layout Editor, and Autorouter. This guide will lead you through the program in the natural order, starting with the schematic editor module and working

through to board design and autorouting. You will benefit most by going through the entire document.

Before you begin you should consult the README file and the files with the extension \*.DOC.

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.

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

## 6 Installation and Program Start

---

### Windows 95/98/NT

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

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

Follow the instructions on the screen. Select the *Standard* installation in order to make sure the subdirectories are installed as foreseen in this tutorial.

If you want to start EAGLE for the first time you have to license it. Keep ready your *User License Certificate*. You can find your individual installation code on it. Please remind that the license card should be stored savely and separately from the media.

For the freeware installation you do not need a *User License Certificate*.

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

### Other Operating Systems

Insert the media in your CD-ROM drive.

Choose the correspondig directory and read the installation notes in the *README* file.

If you want to start EAGLE for the first time you have to license it. Keep ready your *User License Certificate*. You can find your individual installation code on it. Please remind that the license card should be stored savely and separately from the media.

For the freeware installation you do not need a *User License Certificate*.

## **7 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. This topic would exceed the purpose of the tutorial and therefore is not discussed.

## **8 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 a clicking on menu items or toolbar icons. Values are normally entered into appropriate fields.

The knowledge of the internal command language is not necessary to successfully design schematics and boards with EAGLE. However, this concept offers further possibilities which make EAGLE a very flexible tool:

Any command, for instance, can be entered in text format via the command line or can be read from a file. Furthermore, command strings can be assigned individually to function keys. This enables the user e.g. to execute command sequences with a key stroke or a few mouse clicks (see SCRIPT command).

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 command syntax is described in the EAGLE help.

This tutorial is limited to the use of the EAGLE Windows interface. The traditional EAGLE interface can be used as before, but is not part of this tutorial.

## 9 Typographic Conventions

---

### Selecting Menu Items

The character  $\Rightarrow$  means, that a menu selection is to be made. For example

$\Rightarrow$  *File/Save*

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

### Mouse Click

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

- MOVE and F1

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

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

- DEMO.LBR

means: select DEMO.LBR with a double click of the left mouse button from the menu.

### Several Input Alternatives

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

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

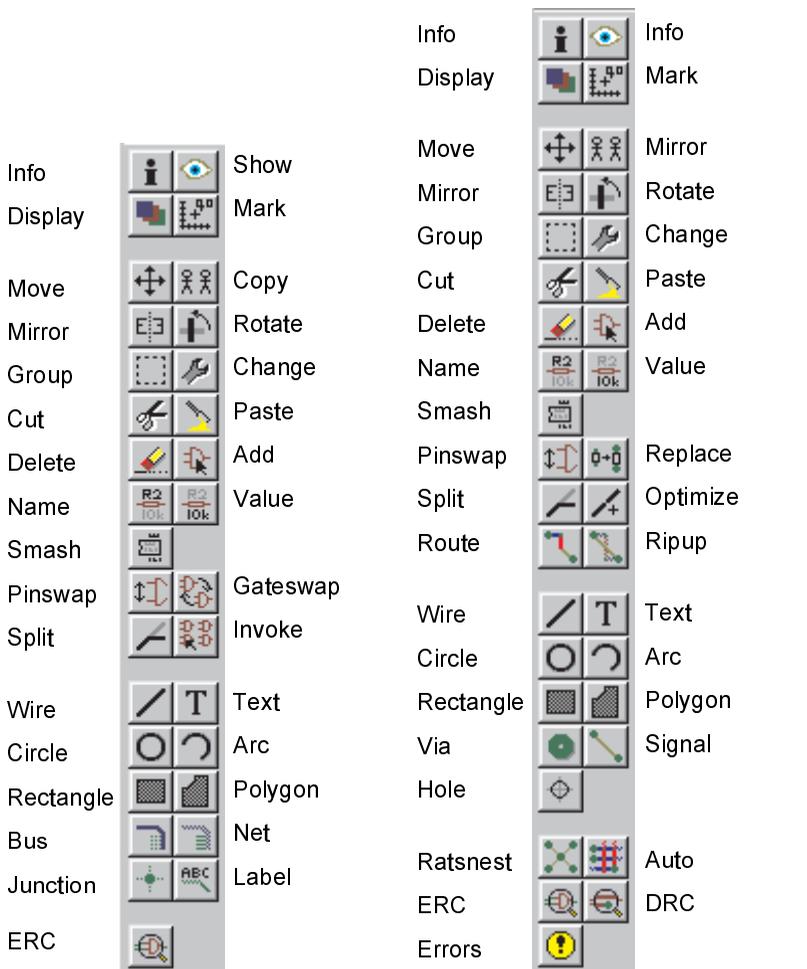
- Clicking the icon 
- Typing MOVE in the command line, followed by the enter key
- Pressing the function key F7 which is assigned to the MOVE command
- Selecting the menu item  $\Rightarrow$  *Edit/Move*

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

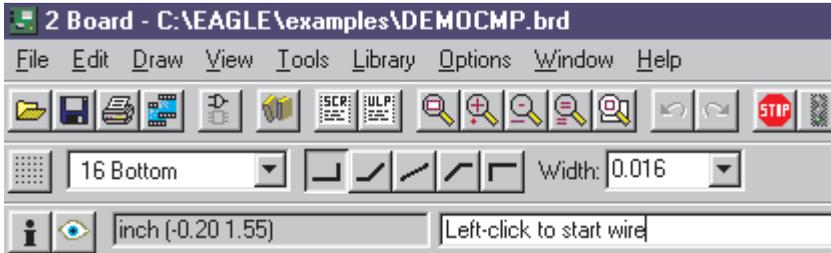
- MOVE

means: click the MOVE icon 

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



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



**From top to bottom: title, menu bar, action toolbar, dynamic parameter toolbar and coordinates display with command line**

## Use of Key Combinations

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

Alt+F1

The Alt key is held down while pressing F1, then release both keys.

## Command and Parameter Input via the Command Line

Actions which need to be terminated with the enter (i.e. return) key are symbolized with the character ←. For example

USE ←

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

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

CHANGE WIDTH 0.024 ←

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 ←

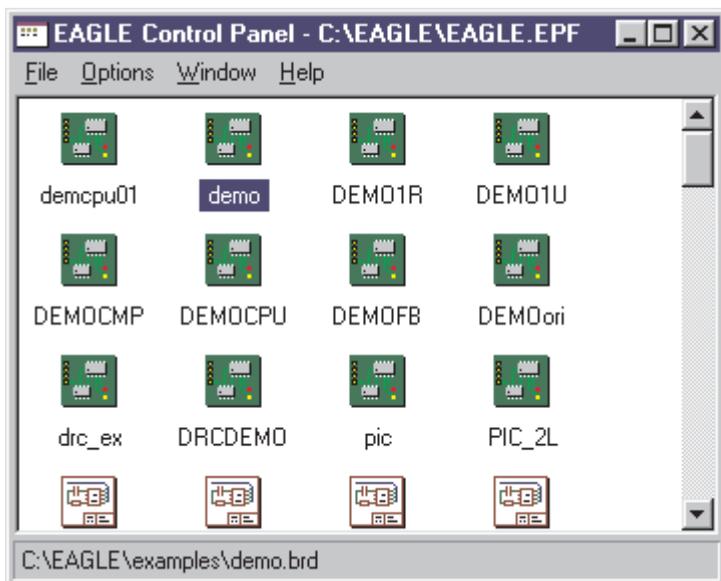
In this tutorial, however, the full commands are used.

## 10 Control Panel

After starting EAGLE, an information window will appear and the control panel, which allows you to load and save projects as well as to setup certain program parameters, will be opened.

The icons in the control panel show files that contain schematics, boards, component libraries, command files (script files), User Language Programs (ULP's), or CAM-Processor jobs (for the automated output of drawings and manufacturing data). They can be opened by ••. If such an icon is selected with the right hand mouse button, a menu will open, where you can choose from several options, like *Copy*, *Delete*, or *Rename*.

If you click an icon with the left mouse button, the full filename with path is shown in the lower area of the window



## EAGLE Files

The following file types can be edited with EAGLE:

Type	Window	Name
Board	Layout Editor	*.BRD
Schematic	Schematic Editor	*.SCH
Library	Library Editor	*.LBR
Script File	Text Editor	*.SCR
User Language Program	Text Editor	*.ULP
Any text file	Text Editor	*.*

*The Linux version only recognizes lower case letter file extensions!*

## EAGLE Project File

EAGLE stores the parameters set in the Options menu of the control panel along with some information on the active windows in a project file (\*.EPF). This file is used to restore the windows and the parameters after the next program-launch.

With  $\Rightarrow$  *File/Save project* you can store a certain user-defined setup under a name of your choice.

If you load this file with  $\Rightarrow$  *File/Open/Project* from the control panel you will get precisely this setup. This enables you to switch easily between different projects.

**For the exercises carried out in the following sections you must load the project `EXAMPLES.EPF`. Otherwise the files mentioned will not show up in the file dialog windows.**

When you start EAGLE the most recently used project will be loaded.

If you exit EAGLE with *Alt+X* and call up the program again, you will find all settings and windows in the same state as before leaving the program.

---

## 11 Load File and select Monitor Zoom

Now let us start doing some exercises. Start EAGLE, and wait until the control panel appears. Open the *Menu*  $\Rightarrow$  *Options/Directories* and you will see the preset paths for different EAGLE files. Click on the HELP

button to obtain detailed information. The `Libraries` field contains two paths as the library `DEMO`, used later on in this tutorial, is stored in the `..\EXAMPLES` directory. Since the settings should not be changed, please click the `Cancel` button.

Now load the `DEMOCMP` board file. You can do this either by selecting the board icon with the name `DEMOCMP` using a double click, or by selecting the file from the menu  $\Rightarrow$  `File/Open/Board`, again using a double click. The schematic with the same name will be loaded along with the board.

Enlarge the board editor window.

Now click the icon  to zoom into the drawing.

By clicking  the drawing will be shown in full size to fit your screen.

A click on  will zoom out.

The command controlled by the  icon is more versatile than in other programs. Click it, and then mark a rectangular area by dragging the mouse cursor while the left mouse button is pressed. Then release the mouse button, just as you are used to from other programs. The marked area will now be displayed.

If you want to select a new center with the same zoom factor, simply click the same icon, 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.

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  .

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 ←
```

## 12 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 DEMOCMP and click in the command toolbar on the icon for the DISPLAY command (look at the toolbar Layout on the previous pages). The marked layers are currently displayed. By clicking on the check boxes the display of each layer can be switched on or off. The *All* and *None* buttons switch on or off all layers.

By selecting/deselecting tPlace (silk screen upper side) the layers tOrigins, tNames, and tValues are selected/deselected, too. The same applies to bPlace (silk screen bottom side).

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

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

## 13 Setting up Grid and Unit

---

Schematics should always be drawn on a grid of 0.1 inches since the libraries are defined this way. The grid for boards is determined by the components used and by the complexity of the board.

Grid and unit are setup with the GRID command by clicking on the

GRID icon  in the parameter toolbar. All values are given in the currently selected unit. Please consult the help pages of the GRID command for detailed information.

## 14 Wires, Circles, Arcs, Rectangles, and Text

---

Wires, circles, arcs, rectangles, and text are created with the WIRE, CIRCLE, ARC, RECTANGLE and TEXT commands. On one hand

---

these objects serve as pure drawing elements for symbols, packages, frames etc., and on the other hand they can perform special functions, such as the definition of restricted areas.

First a new schematic file is to be created.

Close all of the editor windows and select

⇒ *File/New/Schematic*

from the control panel.

A new file with the name UNTITLED.SCH is now created. Normally you should never save a file with the name UNTITLED, but should use

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

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

Click the WIRE command in the command toolbar. All parameters for this command can be set up in the parameter toolbar. Next select layer 95 (names) from the layer-selection combo box. In this layer a rectangular line is to be drawn.



Define the starting point by a click on the left hand mouse button. Move up the cursor slightly to the right and press the right hand mouse button a few times. Please observe how the connection is shown in different diagonal and orthogonal modes. When the connection is displayed in a square angle, press the left hand button to fix its position. Now move the cursor to the starting point and ●● to drop the line. Now you should

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

## Changing Line Width

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

To change the line width of an existing object,

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

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

To change a line width to a value that is not shown the menu of the `CHANGE` command, you can use the command line to type in the value, for example:

```
CHANGE WIDTH 0.017 ←
```

and then click on the selected wire segment.

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

## Undo/Redo

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 canceled by undo.



## The CIRCLE Command

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

- CIRCLE

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

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

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 hand mouse button. Click several times on the right hand 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.

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

## 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. This color can be changed by using the SET command (see help).

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

## The TEXT Command

To activate the TEXT command, used for placing text,

- TEXT

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

Texts containing a spaces or a semicolon have to be enclosed in single quotes.

To change the size of a text:

- CHANGE
- SIZE
- Value in the menu

and • lower left corner or the text. At a rotated text the point of origin can move to its upper right corner. A Text is always displayed so that it can be read from the front or from the right.

To change a text

- CHANGE
- TEXT

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

Using

- CHANGE
- RATIO

you can change the line width in a text in relation to the height (see help page of the CHANGE command).

### Special Text Variables

If you place the text

>SHEET

this string will be substituted with the current sheet number, e.g. 1/1. EAGLE offers a number of similar text variables, e.g. for date/time which reflect the latest change in the file or the drawing output. Library parts are defined with fill-in texts for the name and the value of a component as the values of those parameters are assigned only when the schematic is drawn.

*Use Text Variables only in libraries.*

For more information about the TEXT command use help.

## 15 Using Libraries

---

EAGLE comes with numerous component 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.

To use schematic symbols or packages from a library, you must inform EAGLE beforehand which library you want to use. This is done with the USE command.

### The USE Command

Select the command with the icon



in the action toolbar. If *../examples* does not appear in the *Path* combo box in the window that appears, select this path before you select the DEMO library with

- DEMO.LBR

Then DEMO.LBR is ready for use.

The Path combo box allows the selection of the paths entered in the *Libraries* field of the  $\Rightarrow$  *Options/Directories* menu of the control panel. If you want to activate all of the libraries stored in those paths, type

```
USE *; ←
```

into the command line.

## The ADD Command

To select symbols from a library, • **ADD** in the command toolbar, and you can choose one of the libraries loaded with the USE command from a combo box. Select *DEMO* if this library is not yet selected. Now the devices stored in the DEMO library are shown below. In EAGLE, a Device is a complete component.

• • 7400, and a gate of the 7400 IC is attached to the cursor. By clicking the right mouse button it can be rotated, and it can be positioned at any location with the left mouse button.

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 U1A..U1D, whereas the fifth gate has been named U2A, since this gate requires a second IC.

If you now show the Pins layer, 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). Swaplevel 0 means that this pin cannot be swapped.

*The Pins layer is not usually either printed or plotted.*

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 unlock the gate symbol from the cursor and to call up the selection menu of the ADD command again. Select the IC 555 by • • 555, rotate it 180 degrees by two clicks with the right mouse button and place it somewhere on the

drawing area with the left mouse button. Repeat this with the other symbols of the DEMO library.

*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 different signals).*

*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 555 stored in the DEMO library. In such a case connect the power pins with the appropriate nets.*

### 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
- the gate U2A

A pop-up menu appears.

•• POWER and the power pins for U2 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, • U2A, and the pop-up menu appears. The asterisk assigned to gate A indicates that the gate has been used; those without an asterisk are available for use. If you want U2C to be placed before U2B, •• C in the pop-up menu. The menu closes, and U2C is attached to the cursor to be placed with a •. Once U2C is placed, EAGLE will use up the remaining gates in that package before assigning an additional package.

Don't hesitate to experiment with different libraries and with placing and rotating schematic symbols. • USE to activate an additional library. Use the ADD command to place the symbols.

*You can place devices in a drawing from as many libraries as you want. Devices are saved in the schematic or board files in their entirety.*

*When passing on a file, there is no need to supply the libraries with them.*

## **16 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.*

### **Adding a Frame to a Schematic**

As a start, select a drawing frame from the `FRAMES` library which contains predefined frames in miscellaneous formats. Type

```
USE *; ←
```

in the command line. This way you activate all the libraries stored in the *Libraries* paths (see *Options/Directories* menu of the control panel) for use them with the `ADD` command.

- `ADD` and select a frame from the menu, e.g. with
- `LETTER_P`

A frame which fits on a letter format page (portrait) is now attached to the cursor.

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

Now a further frame is attached to the cursor. Click on the icon with the stop sign to terminate the `ADD` command. Press:

```
Alt+F2
```

to show the frame in full size or click the `Zoom-to-fit` icon the action toolbar.

## Adding and Changing Text

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

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

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

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.

Practice manipulating texts by adding an address or a document number in the text field. `TITLE:` contains the file name in use. `DATE:` contains the date. 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 figure, which you can use for reference. If you don't want to enter the whole schematic you can use the file `DEMO.SCH` stored in the `..examples` directory.

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

If you have not already done so, select the `DEMO.LBR` with the `USE` command.

Using the `ADD` command, place a `6504`, `2716`, `6532`, and a `1X18` approximately as shown in the figure. You can toggle the grid on and off using `F6`, to help you locate 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.

When you have located the parts, start connecting them using the `BUS` and `NET` commands.

***Attention: Do not use the WIRE command!***

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

### The BUS Command

Let's start with the `BUS` command.

- `BUS`

To draw in the bus which connects IC1, IC2 and IC3, move the cursor a few grid lines to the left of IC1, on a level with IC1-13. Click on this point

with the left mouse button to define the starting point of the bus. Move the cursor upwards and two grid lines to the right, use the right mouse button to set the angle of the bus segment to 45 degrees, and **•** to fix the course of the bus so far. In the same way, draw the bus further to the position to the left of IC3-23, as can be seen in the diagram.

- • at the end point finishes the drawing of the bus connection.

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 canceled action once again.

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

*The cursor takes on the form of an arrow 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.*

### The **DELETE** Command

You delete objects with this command. If it is applied to nets, wires or busses, a single segment is deleted at a time. To use this command, **• DELETE** in the command toolbar, take the cursor to the object that is to be deleted, and **•**.

UNDO and REDO work here as well.

### The **NAME** Command

EAGLE automatically allocates names such as B\$.. for buses, P\$.. for pins and N\$.. for nets. In this exercise you give the bus a more meaningful name.

- **NAME** and then **•** on the bus shown in blue. A pop-up menu shows the pre-defined name of the bus. Type in

A[0..9]

and **• OK**. The bus immediately now has this name.

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

### The LABEL Command

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

- LABEL

locate the cursor on the bus and • on the bus. The name of the bus is attached to the cursor and you can place it in any location. You can also rotate the label with the right mouse button. Locate the label approximately as shown in the figure and • to fix its position.

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.

### The NET Command

Now that the bus is in place and labeled, let's drag out the signals A0..A9 from the bus. This is possible only if you have named the bus as described before.

Let's start by connecting IC1-5 to the bus.

- NET in the command toolbar, move the cursor over the bus one grid line above IC1-5 and • . The net connection to the bus must originate from the bus and be drawn to the component pin, if you want to use this convenient way to name it.
- to set the starting point of the net, and a pop-up menu will appear with the net names for this bus.
- A0 and then move the cursor to IC1-5, using the right mouse button to change the line until it appears as shown in the schematic.
- to terminate and fix the net line.

*Repeat this process to connect A1 through A9 as shown in the figure.*

*Attention: A net is only connected to a pin if it is placed on the connection point of the pin. Display the Pins layer with the DISPLAY command to locate these connection points. They are marked with a green circle.*

As mentioned before, nets with the same name define an electrical connection. In the example the nets attached to IC1-5, IC3-8, and IC2-7 share the same name. They define an electrical connection, although the net lines are not visibly connected.

## The SHOW Command

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

To show the complete path for the signal A0, • SHOW in the command toolbar then move the cursor onto IC1 pin 5 and •.

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 as

```
Net: A0
```

in the command line.

To show an object with a specific name, • SHOW and type the name (for example D0 ←) in the command line. You can subsequently type other names without the need to reactivate the SHOW command.

## The MOVE Command

In order to avoid mistakes when placing and moving nets you have to understand the following effects of the MOVE command:

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

## 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.

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

```
SHOW R1 ←
```

```
SHOW C1 ←
```

```
SHOW IC1 ←
```

Redraw the screen, e.g. with F2 and press the *up-arrow* and *down-arrow* keys several times. As you can see, you can scroll through

the list of the recently used commands. As soon as the desired command appears in the command line press the enter key.

### Completing the Schematic

Once you have completed the buses, use the `ADD` command to add the remaining components and the symbols for VCC and GND. Remember that you can use the `MOVE` command to move objects around.

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 and `••` to terminate a net line.

### The JUNCTION Command

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 SMASH Command

You will notice that when you rotate diodes and resistors from the horizontal to the vertical position, their reference designators (name) and value rotate with the part. EAGLE provides a `SMASH` command that allows you to change and rotate the names and values of symbols.

To activate the command

- `SMASH` in the command toolbar.

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

Note that the selection point of a text is usually at its lower left corner. After a 180 degree rotation it is at 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. You can do the same thing with the symbol's value which is currently `DIODE-10`

provided you have not loaded the schematic file `DEMO.SCH`. The value of a part used in a schematic defaults to the Device name used in the library. You can change the value with the `VALUE` command.

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 `CHANGE` icon and select `SIZE` from menu).

### The `VALUE` Command

EAGLE allows you to change the value of resistors, capacitors, ICs, etc. with the `VALUE` command. To activate the command

- `VALUE` in the command toolbar.

Locate the cursor on the diode symbol and •. Type the new value, `1N4148`, •OK, and the new value is now displayed.

You can use the `NAME` command to change the names of resistors, capacitors, ICs, 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 `DEMO.SCH`.

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

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

*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.

### 17 Function Keys

---

As indicated previously, several function keys are predefined with different commands. This layout can be changed by the user at any time. Only the Windows specific keys (like F1 for the help function) must not be redefined.

Please call up the help page for the ASSIGN command to get further information on the use of the function keys.

### 18 Automatic Forward&Back Annotation

---

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

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

### 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.

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

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

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

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

To monitor the Forward&Back Annotation load the `DEMOFB.SCH` file. The board `DEMOFB.BRD` will be loaded automatically into the Layout Editor.

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

---

## 19 Designing a PC Board

---

In this section you will create a small PCB design and modify an existing design using the Layout Editor. First, you will create a board without a schematic. This section is useful mainly for those users who have no Schematic Module. If you have the Schematic Module you would normally not have to deal with the steps described in the following section. You should, however, read through this section as it deals with some generally useful points.

### Designing a Board without a Schematic

Open a new file and enlarge the editor window.

#### Defining Board Shape

The first thing we'll 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 the Dimension layer : • `WIRE`, and select the layer `Dimension` from the

combo box in the parameter toolbar.

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

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

By double clicking the left mouse button you will terminate the WIRE command. The board outlines are now defined.

Using the `MOVE` command, the edges can be moved, or use `UNDO/REDO` to recall the previous actions and perhaps make changes.

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

### Placement Grid

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

### Loading a Library for Using

Click the `USE` icon in the action toolbar :



and select the library `DEMO.LBR`.

### Placing Components

**•** `ADD` in the command toolbar and **••** `DIL14`.

A 14-pin DIL package is now attached to the cursor. It can be rotated with the right mouse button and then placed with the left mouse button. Place two DIL-14 packages.

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

## Placing SMD Packages

Now use `ADD` to place two `1210` packages on the board. If you know the package name, you can type

```
ADD 1210 ←
```

in the command line.

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

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

## Providing Names

To assign a name to the packages just placed:

- `NAME` in the command toolbar.

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

```
IC1 ←
```

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

## Providing Values

To assign values to an element:

- `VALUE` in the command toolbar.

Move the cursor near the origin of `IC1` and `•`.

A pop-up 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` in the command toolbar and type

`GND` ←

- on pad 7 of IC1 (IC1-7)

move the cursor to IC2-7 and •• to terminate the GND airwire. The two pads are now connected to the GND signal.

Next we will connect VCC. Type

`VCC` ←

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

Define further signals using the same procedure.

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

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

Airwires can be deleted with the `DELETE` command if you don't work under Forward&Back Annotation control (in such a case you have to delete a signal by deleting the related nets in the schematic). Please note that `UNDO` and `REDO` function under the control of the Forward&Back Annotation, too.

## Creating a Board from a Schematic

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

## Generating a Board File

Load the file `DEMO.SCH` and click on the `BOARD` icon :



With this command you create a board file with the same name as the loaded schematic (`DEMO.BRD`). Answer the *Create file?* question with •

OK. Maximize the Layout Editor window.

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

- `MOVE`, and
- the right vertical edge of the board shape somewhere in the middle.

Move the cursor a little to the left and

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

### Component Placement

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

- `MOVE`,
- the biggest IC somewhere in its center and move the cursor inside the board outlines. The component and the airwires remain attached to the cursor. Press the right mouse button if you want to rotate the component.
- to fix the position of the component.

Place all of the components using the `MOVE` command.

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.).

### Autorouter: A Brief Example

If you would like to see a small demo of the Autorouter, click the icon for the `AUTO` command in the command toolbar. In the Autorouter menu click the `Start` button.

It should be finished in no time at all, provided the placement is not too bad (watch the command line). 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.

### Routing Manually

The `ROUTE` command changes the airwires into routed tracks.

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

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

*All measurements relate to the current unit selected with the `GRID` command.*

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

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

While the `ROUTE` command is active you can select the angle between two wire segments with the right hand mouse button.

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

### Board Changes

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

- move wire segments and components with `MOVE`,
- use the `DELETE` command to change routed tracks to airwires,
- use `RIPUP` to change all of the routed tracks to airwires.

You can start the Autorouter any time you want, whether manually routed tracks exist or not. The routed tracks will not be touched by the Autorouter. Typically, the power signal paths and other critical signals

are routed manually, before the board is passed on to the Autorouter.

## Further Usage of the Layout Editor

In this section you will modify a routed demo board. Load the file `DEMOCMP.BRD`, and enlarge the editor window.

Next a few important commands will be repeated.

### DISPLAY Command

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

- `DISPLAY`

and select with your mouse the layer 21 (`tPlace`). This layer contains information for the top side silkscreen of the board. By selecting or deselecting it, the layers `tOrigins`, `tNames` and `tValues` will be shown or hidden.

To activate this change, • `OK`.

### 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. Selecting the wire in the middle will move it in parallel.

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 the `tOrigins` layer has to be displayed. The same applies to components placed on the Bottom layer and the `bOrigins` layer.

As long as the `MOVE` command is active, you can rotate the object with the right mouse button.

### GROUP Command

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

- GROUP

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

*Please make sure that you only select objects which are in a visible layer. Packages on the Top layer can only be selected if the layer tOrigins is visible, and packages at the Bottom layer can only be selected if the layer bOrigins is visible.*

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

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

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

### **SPLIT Command**

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

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

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

### **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, type

```
CHANGE WIDTH .23 ←
```

in the command line 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.

## DELETE Command

Use the `DELETE` command to change a wire segment into an airwire.

## 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 with a •• until the airwire is completed.

## RIPUP Command

If you would like to turn 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.

### 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.

### Refresh Screen

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

### UNDO/REDO

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

### Inner Layers

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

### Supply Layers

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

```
SHOW GND ←
```

The GND signal will be highlighted.

Then type:

```
RIPUP GND ←
```

The GND-Signal will now be displayed as airwires only.

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

```
LAYER 2 $GND ←
```

The `LAYER` command is not available as an icon, since this does not make a lot of sense. Therefore it can only be entered in the command line.

Click the `RATSNEST` icon to make the airwires of the `GND`-Signal disappear.

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

```
DISPLAY NONE $GND ←
```

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

```
dis none $gnd ←
```

Now type

```
SHOW GND ←
```

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

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

## Copper Pouring

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 `DEMOCMP.BRD` once again, enlarge the window, and ripup the `GND` signal:

```
RIPUP GND ←
```

Use the `DISPLAY` command to switch on the *Layer Top, Pads, Vias, and Dimension*. Use the button `NONE`, in the menu, to switch off the display of all other layers.

Click the `POLYGON` icon in the command toolbar and type:

```
GND ←
```

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

Select the Layer `TOP` from the combo box in the parameter toolbar. Then:

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

The double click closes the polygon.

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

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

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

```
SHOW GND ←
```

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

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

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

## 20 Autorouter

---

No Autorouter on earth will lay your board out exactly as you would like. Many pc-board designers therefore reject the use of autorouters in general. In this section we will demonstrate that a good Autorouter can free you from a lot of boring work.

Load the board `DEMO1U.BRD`. Please note that this is not possible with the Standard or Light version of EAGLE.

Switch of the layer `tPlace` using the `DISPLAY` command, so that the components are not shown anymore.

This board contains manually routed supply signals. Circles and rectangles in the layers `tRestrict` and `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.

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

A pop-up menu appears where you can enter individual settings (see help). If you change the parameters the Autorouter will store them and use this information in future passes.

As we want to route all of the unrouted signals

- `Start`

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

```
AUTO; ←
```

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

Watch the status messages appearing in the command line. 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 sign icon.

A protocol of the routing run is stored in the file `DEMO1U.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.

---

## 21 Design Rule Check

---

The last step when designing a board is the Design Rule Check (DRC). The `DRC` command checks if the board corresponds with the user-defined design rules.

Load the file `DEMOCMP.BRD` and switch off the `tPlace` layer.

To run the test, click on the `DRC` icon in the command toolbar.

A menu appears which allows you to select several checks and to enter values for design rules.

- `HELP`

to find out more about this command and its parameters.

Turn off the checks for `OffGrid` and `Angle` with the mouse. They are not relevant here, as the SMD parts are not placed in the selected grid, and this would result in a number of error messages.

Now

- OK

The command line should show the message *No errors*. Now you can be sure the board matches your design rules.

Refresh the screen with `F2` or by clicking the Redraw icon.

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

```
DRC; ←
```

The semicolon prevents the DRC menu from appearing.

The command line should now show a number of errors.

Click the Zoom-to-fit icon and then the `ERRORS` icon in the command toolbar. A menu will open showing an error list. Press `F1`, to find out more about the `ERRORS` command.

As soon as you have edited the board clear the screen from the previous DRC results (shown as small polygons) by typing

```
DRC CLEAR ←
```

*If you find objects in your board which cannot be deleted with the `DELETE` command it might be the result of a DRC. Type `DRC CLEAR` to delete them.*

## 22 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.

## 23 Output of Drawings and Manufacturing Data

---

EAGLE can output drawings using the `PRINT` command. This command can be found in the *File* menus of the Schematic or Layout Editor. Or you can use the CAM Processor for generating output. It can be called up via the *File* menu of the control panel or, alternatively, via the CAM Processor icon from an editor window.

Usually the `PRINT` command is used only for drawings which are produced for control or documentation purposes, not for the output of layouts.

You can use the printers defined under Windows with this command.

The Linux version generates Postscript files that can be sent to `lpr` or to a file.

A printout of the drawing in the active editor window with the actual layer settings will be generated.

Film and manufacturing data are generated with the CAM Processor. It uses its own drivers, which can be defined or modified by the user (see `EAGLE.DEF` file).

***Attention: The DOS version does not support the `PRINT` command. Use the CAM processor for printing!***

## Output a Schematic with the `PRINT` Command

The schematic `DEMOCMP.SCH` is to be printed in black/white and full format on one page.

Load the file `DEMOCMP.SCH` and click the `PRINT` icon in the action toolbar. Check the boxes `Black`, `Solid` and `Rotate` (as the drawing is in landscape format). The boxes `Mirror` and `Upside down` are not marked.

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

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

For using a printer in the Linux version choose `lpr` in the `Printer` field. If you would like to print to a file, type in for example

```
cat > board.ps
```

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

## Generating Gerber Data with the CAM Processor

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

The File GERBER.CAM, which can be found in the default subdirectory for CAM jobs, automates the output of Gerber data for double sided boards. Attention: the process described can only be used for Gerber photoplotters with flexible aperture wheels, not for photoplotters with fixed aperture wheels.

Load the job into the CAM Processor, either by double clicking the icon with the name GERBER in the control panel, or by clicking the CAM Processor icon in an editor window and selecting GERBER.CAM in the file dialog.

Now load the DEMOCMP.BRD board file, for which the Gerber data is to be generated:

⇒ *File/Open/Board* and •• DEMOCMP.BRD

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

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

The files have the following meanings:

Democmp.cmp	Component side
Democmp.sol	Solder side
Democmp.plc	Silkscreen for component side
Democmp.stc	Soldering mask for the component side
Democmp.sts	Soldering mask for the solder side
Democmp.whl	Aperture wheel file
Democmp.gpi	Information file, not relevant, to be deleted
Democmp.***	Temporary file, to be deleted

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

Drilling data can be generated accordingly by using the job EXCELLON.CAM.

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

## **24 EAGLE User Language - A Universal Output Tool**

---

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

Please have a look at the files with the extension \*.ULP and the User Language section of the on-line help to get a better idea of the possibilities inherent in the program.

## **25 Script Files - Flexible Input Interface**

---

Script files are text files which can contain any EAGLE command (see SCRIPT command). Thus you have a flexible input interface defined by the EAGLE command syntax (see help or EAGLE Reference Manual).

Along with the User Language, script files enable the user to implement his own functions, e.g. a specific snap function dragging all of the components to the closest grid point. The principle is easy:

execute a User Language Program which creates a script file with the RUN command. The script file contains the MOVE commands to reposition the components. Then execute the script file using the SCRIPT command.

This is merely one example of the flexibility of EAGLE.

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

***USA and other countries:***

Phone +1 (561) 274 8355, USA also: 1-800-858-8355  
Fax +1 (561) 274 8218  
Internet: [www.CadSoftUSA.COM](http://www.CadSoftUSA.COM)  
Email: [Info@CadSoftUSA.COM](mailto: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](http://www.CadSoft.DE)  
Email: [Info@CadSoft.DE](mailto:Info@CadSoft.DE)

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