

Department of Electrical and Electronics Engineering

SSN College of Engineering



TABLE OF CONTENTS

EAGLE CADSOFT Professional	2
Getting Started	3
Toolbar quick reference	5
Creating the Schematic	6
Creating the Layout	10
Exporting the Image	12
Creating a custom library file	17

EAGLE CADSOFT PROFESSIONAL

EAGLE (Easily Applicable Graphical Layout Editor) is a schematic capture and PCB layout tool for hobbyists and DIY enthusiasts. EAGLE is a popular choice as it is an easy to use, powerful and affordable schematic capture and printed circuit board design package. Here you will learn how to get started, how to use the interface, and how to design a board file from a schematic and export it for hardware use.

GETTING STARTED

OPENING

Begin by opening EAGLE CADSOFT PRO. Click Start>Programs>EAGLE Layout Editor 4.16r2>EAGLE 4.16r2, or click on the shortcut on the desktop.

You will see the EAGLE control panel.

Name /	Description	Libraries
Name / Libranes Design Rules Scripts CAM Jobs Projects	Lescription Libraries Design Rules User Language Programs Script Files CAM Processor Jobs Project Folder	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. Additional new libraries, that have not yet been officially released, can be found on CadSoft's internet site at the download section of http://www.cadsoftusa.com. Use the ADD command in the Schematic Editor or Layout Editor window to search for a certain device or package! Information about defining your own libraries can be found in the file library.txt in the doc directory.

Now we need to create a Project which will contain the Schematic and the Board File. For

this click on File>New>Project

New 🕨	Project	
Open Open recent projects	<u>S</u> chematic <u>B</u> oard	
Save all	Library	
<u>C</u> lose project	CAM Job	
c <u>a</u> it Ait+A	ULP S <u>c</u> ript <u>I</u> ext	

Rename the Project to the desired name.



Now right click on the Project, select New>Schematic.

Name /		Descrip	tion	Empty Project
Name / Descri Descri Descripts Libraries Librarie Descripts Design CAM Jobs CAM P Projects Project Close Project New		Libraries Design User La Script Fi CAM Pr Project Example ect	s Rules nguage Programs les ocessor Jobs Folder es Folder Project	Use the context menu to create new schematic or boar within this project.
		Þ	<u>S</u> chematic	
	<u>R</u> ename Delete		<u>B</u> oard Library	
	Edit Descri	iption	CAM Job	
	<u>U</u> se all Use <u>n</u> one		<u>U</u> LP S <u>c</u> ript <u>T</u> ext	
C:\Program	Files\EAGLE-4.1	6r2\pro	_ Eolder Proiect	

You will be able to see the Schematic window which is used to draw the desired circuit.



Let us also look at the various Toolbar Buttons available in the Schematic editor as well as the Layout editor.

COM		OLBARS		Layout Edit	or
com		OEDANS	Info	+ 📀	Show
S	chematic E	ditor	Display	•	Mark
Info	•	Show	Move	\$ ₹ \$	Mirror
Display	1 40	Mark	Mirror	Eja 🕂	Rotate
Display	····	2	Group	1 1	Change
Move	♦ ³ / ₂	Сору	Cut	8 >	Paste
Mirror	Eja 🕂	Rotate	Delete	XA	Add
Group	[] Þ	Change	Pinswap	\$ - 0 (]_\$	Replace
Cut	F 🔪	Paste	Name	R2 83	Value
Delete	X	Add	Smash	= r	Miter
Pinswap	tt 83	Gateswap	Split	V 1.	Optimize
Name		Value	Route	25	Ripup
Smash	ä r	Miter	Wire	/ T	Text
Split	1 22	Invoke	Circle	6.	Are
	/ T	-	Circle		AIC
Wire		Text	Rectangle		Polygon
Circle	0.5	Arc	Via	0 \.	Signal
Rectangle		Polygon	Hole	¢	
Bus		Net		×#	Auto
Junction		Label	Ratsnest	A A	Auto
	A		ERC		DRC
ERC	C.		Errors	•	

ΤοοΙ	Usage	Tool	Usage
Info	Provides information when clicked on	Net	Used to draw wire connections
	something		between component pins
Display	Used to show/hide various details in	Wire	Same as above
	schematic and Layout		
Add	Displays the component library	Junction	Used to place junctions in wire
			connections
Gateswap	Used to change pin sequence in ICs	Route	Used for manual routing of tracks
Name	Used to change name of the component		
Value	Used to mark the value of the component	Ripup	Used to break established track
			connections
Miter	Used to change perpendicular lines to	Via	Used to place Via points
	smooth edges		
Split	Used to change net shapes	Auto	Autorouter feature
Invoke	Reveals Power Ports for select ICs	DRC	Used to Load and check design rules
Text	Used to insert text	Errors	Displays errors if any

CREATING THE SCHEMATIC

In this discussion, we will be creating the following circuit. It is a common LED sequencer circuit which lights up a series of 10 LEDs sequentially.



Source: http://www.allaboutcircuits.com/vol_6/chpt_7/6.html

Now in the schematic window click on the ADD button in the toolbar, a window pops up. In that use the search bar to choose the component that is required. Almost all the components in the market are available in this library but the names of devices have to be specific or else a result may not appear. Click on the OK button once the component has been chosen.

2 1 Schematic - C:\Program Files\EAGLE-4.1	16r2\projects\Led Flasher\untitled.sc	n - EAGLE 4.16r2 Professional		2 5 0
Eile Edit Draw View Tools Library O	E ADD		×	<u> </u>
日本 日本	Binch Sharch Sharch	Description 19-Inch Ste Eurocardi CMOS Logo Devices, CMOS Logo Devices, TTL Logo Devices, 74 TTL Devices, 74x Se TTL Device, 74x Se Alart Technologies Alarta Frogrammable L Advanced Moro Devic AMD MACH4/MACH5 Analog Device Comp AVR Device Lithum Batteries and h Bur-Brown Component Schoff Current Bus Be Sealors and Buzzen Sealors and Buzzen Sealors and Buzzen Sealors and Buzzen Sealors and Suzzen Sealors and Suzzen S		

CHOOSING A 555 TIMER

For example to get a 555 timer, type "timer" in the search bar and you will get multiple results for the IC. Choose LM555N if your IC is a dual in-line package (the most common ones) or if it's a surface mount device, choose LM555D.

DIL08 stands for Dual In-Line 8 pin IC, likewise SO08 for Small Outline 8 pin IC (SOIC : Small Outline ICs)

Name /	Description	-	
<u>⊨</u> 40xx	CMOS Logic Devices,		
	Quad precision TIMER	14	
i⊒ 45xx	CMOS Logic Devices,	>NAME	
⊕ 4536	Programmable TIMER	2 3	
⊕ 4541	OSCILLATOR/PROGE		0000
⊟ 74xx-eu	TTL Devices, 74xx Se	4 7	ᆝᅟᆜᆘᆋᆋᆋᆋᆋ
⊕ 74*292	Programmable frequen	GR DIS -	₹)>VALUE
	Programmable frequen	5 су тип 6	Z
🗐 74xx-us	TTL Devices, 74xx Se	- CV IRR	
⊕ 74*292	Programmable frequen	1 GND V+ 8	
	Programmable frequen	OILD. THE	
🗐 linear	Linear Devices	>VALUE	
□ *555	TIMER		
LM555D	SO08		
LM555N	DIL08		I
NE555D	SO08	TIMER	
NE555N	DILO8		
SE555D	SO08	Package: DIL08	
- SE555N	DILUS	Dual In Line Package	
	DILU8		
UA555D	SUUS	-	
verich V Smds V Descrir	ntion V Preview		
imer		1	

1 Schemade - C. (Flogram Hies/Exote-4.1012/projects/Led Hasher/unitide.sch - Exote 4.1012 Floressional	
Elle Edit Draw View Jools Library Options Window Help	
● ■ ● 第 1 2 1/1 ■ ● 話話 4, 9, 9, 9, 9, 9 9 9	
0.1inch (0.12.1)	
IC1 IC1 </td <td></td>	

Now drag the component onto the workspace and place it in the white area.

A few components like the 4017N decade counter may not show its Power pins (VCC and GND). For such ICs we have to reveal those connections. To do this click on Edit>Invoke.

Undo Alt+Backspace Redo Alt+Shift+Backspace	Q Q Q Q 0 ∽ ~ 0 1 ?	
Stop command		
ddd Corg Cog Odrte Gober Jonate More More More More Parte Parte Parte	C1 TR Q <u>3</u> R DIS <u>7</u> CV THR <u>6</u>	$ \begin{array}{c ccccccccccccccccccccccccccccccccccc$
Rotate Smash Spit FB Yalue Net classes	GND V+ 8 LM555N	13 C ENA Q9 15 RES CO 12

Click on the space inside the IC, the Invoke window will pop up. Select the field inside and click OK.

t gan praw yie	2 14 Clearly Sphere Window Rep	
0.1 inch (0.92	2	
12		100
XX		102
÷		3
~		2 Invoke IC2 (4017N)
-	101	Gate Symbol Add Swap Street Q1
83		A 4017 Net 0 1 02 4
*	2	P PWWN Hequid 0 0
2.8		Q3 10
T		Q4 10
2	4 P DIG 7	05 1
4		0.5
1	F + 0	Q6
~	CV THR	OK Cancel O7 0
	ov mit	08 9
	1 8	13
	- GND V+	ENA Q9
	And a set of the set o	
	1 M555N	15 550 00 12
	LIVIOODIA	RES CO
		4017N

Now the Power pins will be displayed, place it in any convenient location near the IC. All connections to VCC and GND of this IC should be given to these two pins.



Now in this way add the rest of the components required in the LED Sequencer circuit using the ADD Component Library.



Now we need to connect the components using wires or "nets". For this select the Net tool from the Toolbar. Now click on one component pin and stretch the wire to the other component pin and click again to fix the wire between the two pins.



After all the wires connections are established, the schematic should look like this. Neat perpendicular connects always makes sure that there is no confusion in the wiring or any accidental connections.



NOTE:

1. Care should be taken while connecting the wires to the components. You should connect the wire exactly to the edge of the terminal as shown below:



- 2. If you come across any junction you need to indicate the same using Junction ** button.
- 3. Make orthogonal connections that de-clutter the schematic and make it look organized.
- 4. After giving connections ensure that all required wiring is established. This can be checked by selecting the Move tool and by clicking and dragging the symbol to see if the pins/leads are connected to the wires.

With the above precautions connect the components.

This table will provide you the guideline to add some common components:

#	COMPONENT	KEYWORD	GUIDELINE	DESCRIPTION
1.	Berg strip/Connector/Push or Pull Switch	connector	Scroll down to MTA02-100	'MTA' signifies type of connector, '02' signifies number of pins in the connector and '100' signifies the size of the connector. If you want 3 pin select MTA03-100.
2.	Resistor	resistor	Expand R-EU_ and select R- EU_0207/10.	'EU' indicates European style resistors '0207' signifies resistor type 10 signifies grid length. If you want Vertical mounting style select REU_0207/5V or R-EU_0207/2V. 'V' indicates vertical mounting type resistor
3.	Capacitor	capacitor	Expand C , select C2.5/5	'2.5' signifies the area and '5' signifies the grid length
4.	Diode	diode	Select 1N4004	You can use the same for 1N4007 also
5.	Bridge rectifier	rectifier	Scroll down to RB1A and press OK	
6.	LED	led	Scroll down and expand LED. Select LED5MM	5MM LED is the usual LED available in market.
7.	Potential Divider/Trimm Pot	trimm	Expand R-TRIMM and select R- TRIMM64W	
8.	OP-AMP	op amp	Expand *741 and select LM741P	
9.	PIC Microcontroller 18F4550 (40 pin)	microchip	Scroll down and expand PIC18F4*_20 and select PIC18F4550_40	'40' signifies 40 Pin IC
10.	PIC Microcontroller 18F2550 (28 pin)	microchip	Scroll down and expand PIC18F2*_28 and select PIC18F2550_28DIP	'28' signifies 28 Pin IC and 'DIP' is abbreviation for Dual In Package

CREATING THE LAYOUT

After all the connections are given, we create the "board" file. This is done by selecting the "create board file" tool in the upper toolbar. An info box appears, Click YES to proceed.



This is the Layout window. The components and the workspace frame are shown. These components have to be optimally arranged inside the workspace frame which will be the actual placement of the devices in hardware.



Drag and drop the components onto the workspace frame as shown below.

2 Board - Dt\College Work\Eupraxia 2010\PCB_video - Copy.brd - EAGLE 4.16r2 Professional	
Elle Edit Draw View Tools Library Options Window Help	
▶ 월 월 달 8 ● 昭 昭 9, 9, 9, 9, 10 ● ● ● ?	
£ >	
	-R
· · · · · · · · · · · · · · · · · · ·	
ar	
	Cl and a second
07 Pro Pro Pro	+
C & Ditter Date Date Date	
· · · · · · · · · · · · · · · · · · ·	CT LED 1
	9
SHE	
la La dista si	

Now we need to load the standard DRC settings file which will determine various factors of the wires such as

- Track thickness
- Track spacing
 Track
 routing

Click on the DRC tool and the Load button in the DRC window. Load the DRC file (provided in the CD) and click open. Click on the Check button and ignore errors if any.

Eile Edit Draw View Tools Lib	ary <u>Options Window H</u> elp 號 육, 육, 오, 육, 월, 다 ~ 🚳 🎆 ?		
		Correction CEG ROB 2010 + A_Bot_Sch • 4 Search A_Bot_Sch	2
0.05 inch (0.55 1.05)		Organize 🕶 New folder 🔠 💌 🛅	0
● 世 秋 中 ◆ ○ 金 村 器 章 ▷ 『 ヽ ○ ■ ○ ◆ 、 ● ●	DRC (Vision_1) File Layers Clearance Datance Sizes Retiring Shapes Su EAGLE Design Rules The defaul Design Rules have been set to cover a wide range of applications. Your pi e please make the necessary adjustments and save your customized design rules und	Vightas ** Text House Pictures Pictures Pictures Pictures Videos Pictures Vision_1.dru 16-09-09 1:14 PM DRU File Computer Acer (C) Acer (C) College Notes College Notes College Notes College Notes Edit Description Select Apply	

In the next step, we need to lay the tracks or otherwise "route" it. To do this, select the Autoroute tool from the toolbar. Select the "|" option in both top and bottom fields since this is a double sided PCB. (For a single sided PCB select N/A in the top field and set only the bottom field). Press OK when done.

🛿 2 Board - D\College Work\Eupraxia 2010\PCB_video.brd - EAGLE 4.16r2 Professional 🔄	6 8
File Edit Draw View Iools Library Options Window Help	
┣━ ■ ● ■ 2 ● 2 ● 2 ● ● ● ●	
005 inch (0.55 1 20)	
Image: Constraint of the setup	
Central Busses Route Optimize1 Optimize2 Optimize3 Prefered Directoria Prefered Directoria Prefered Directoria Prefered Directoria Prefered Directoria Prefered Prefered Prefered Prefered Directoria Directoria Prefered Prefered	

After autoroute completes, the board file will look like this. The blue lines represent the bottom side tracks and the red lines are the top side tracks. The autorouting is a programmed feature in EAGLE which optimally selects the best path for a track (though sometimes the tracks may unnecessarily be elongated) using search optimization algorithms.



EXPORTING THE IMAGE

From this board file we can export images of the top side and bottom side separately for taking print. This can be done as follows.

Select the display tool and in the display window select the labels called "bottom", "pads" and "vias" and click OK.

2 Board - D:\College Work\Eupraxia 2010\PCB_video.brd - EAGLE 4.	L6r2 Professional	- ē X
File Edit Draw View Tools Library Options Window Help		
🕞 🖬 🚝 😤 🕷 뛢 뛢 텍 및 및 및 및 🗠 이	4 🚥 🖉 😚	
0.05 inch (0.60 /		
Display		
Layers:		
₩ XX Nr Name		
E[3 + 13 Route 13		
	5 0 0 0 0	
TIG Bottom		
17 Pads		
18 Vias		
20 Dimension		
7 / 21 Place		
22 bPlace		
✓ T	5 4017N +	
	00000000	
OK Cancel		
*		
X #		
4		

The board file now looks like this. To create an image of this bottom side track pattern click on File>Export>Image. In the Export Image dialog box

- set resolution to 150 dpi
- check monochrome
- Provide a file name ("bottom") and press OK.

2 Board - D:\College Work\Eu	praxia 2010/PCB_video.brd - EAGLE 4.16/2 Professional
<u>Eile</u> Edit <u>D</u> raw <u>V</u> iew <u>T</u> ools <u>L</u> i	jbrary Options Window Help
New	第 後 谷 古 部 0 ~ ● 第 3
<u>O</u> pen	
Open recent +	
Save	
Save <u>a</u> s	
Save a <u>l</u> l	
Print setup	a 6 6 0 a
Print	•
CAM Processor	0000
Switch to schematic	
Export	Ed Export Inage
Scrip <u>t</u>	Bie bottom Browse
<u>R</u> un Script	Deboard V Monochrome
<u>C</u> lose Directory	Besolution 150 b) dpi
Egit Netlist	Image Size 570 × 495 pixel
O N Partlist	OK Cancel
NetScript	
🚯 🖨 🛛 Image	a a a a a a a a a a a a a
•	
4	

Repeat the same for the Top side by deselecting "bottom" in the Display tool and selecting Top in its place. The exported top and bottom side images should look like this.



Use an image editor such as MS Paint to invert the colors and to flip the top side image laterally.



Take a printout of these two images in separate papers to get the final board file images for the two sides of the PCB. With the images we can proceed to the actual process of PCB fabrication.

Thus we have demonstrated the design of circuits using the EAGLE Schematic editor and the creation of board file images using the Layout editor which is used in the fabrication of printed circuit boards.



A finished single sided PCB that is ready to be used!



CREATING A CUSTOM LIBRARY FILE

In this section, we will show you how to create a user defined component that can be used in the EAGLE schematic and layout editor. The need to create a user defined component is due to absence of certain rare components in the otherwise extensive parts library of EAGLE CAD.

Begin by selecting File>New>Library.



You will notice 3 important buttons:

- 1. Device
- 2. Package
- 3. Symbol

These define the library file; the Package is the diagram that appears in the Board layout editor and the Symbol is the diagram that appears in the schematic editor. These two combined together forms the "Device".

2 /1	Library -	C:\Pro	gram Fil	es\EAGLE	-5.6.0\lbr\	untitled.lbr () - EAGLE 5	.6.0 Profess	sional		
File	Edit	Draw	View	Lib ? ary	Options	Window	Help				
	8	99 90	🛱 🔁		Q.A	<u>q</u> <u>q</u> <u>Q</u>	6 0	()	3		
			Device								
₽ O	0.1 inch	(-0. <mark>1</mark> .(1)								•
											÷
	4									+	
Edit	a device										

Begin by clicking on Symbol. A pop up window will appear where the name of the device has to be provided. In this tutorial we will be creating a 4 pin Transistor IC. Hence name the device as "TRANSISTOR".

🕼 Edit	
Package	
	🖉 Warning
	Create new package 'TRANSISTOR'?
New: TRANSISTOR	-
	Ver Ne

Click Yes in the Warning dialog box. The Symbol editor window now appears. Here we will be drawing the image of a standard NPN transistor.

<u>File E</u> dit <u>D</u> raw <u>V</u> iew <u>L</u> ib	rary <u>O</u> ptions <u>W</u> indow <u>H</u> elp	
🚘 🖬 🎒 🗱 🕸 😂 🦉		19 📓 🖓
94 Symbols 👻 🔟	//////////////////////////////////////	- C Width:
0.1 inch (0.5 0.6)	<i>J</i>	
∎ I. ² "		
t, 99		
± ∧∧ i∃ ∔		
······································		
* ≽	N	
×		
r 🛛		
∠)r		
ອາ		
× 1		

Use the Wire tool to draw the diagram as shown. For 45 degree lines select the angle button in the upper toolbar.

The completed image should look like this:



Now we need to add "Pins" to the diagram and for this select the Pin tool and add like this:



Additionally we can name the pins by selecting the "Names" tool and naming each pin separately.



Once this has been completed select the Package button to create the package for the Transistor.



Click OK when this dialog appears:

Package	~	
TRANSISTOR		
ew:		
ew:	Pac	Sym

We begin by fixing the pads for the component. This is the most important part of creating a Library file as the dimensions of the custom drawing should match exactly with the specifications provided in the component's datasheet. For example let's consider a 28 pin PIC18F2550 Plastic Dual In-Line Package (PDIP) IC.

Note the values of Pitch 'p' and the IC width 'E1'. These values would be the same for our 4 Pin Transistor IC, hence we can borrow them.



Use the grid to create these dimensions so that the pads can be placed precisely. By default the grid is at 0.1 inches of
2.54mm. We can change this to any value that is convenient.

.310

285

1 365

.325

.295

1 385

7.62

6.99

34 16

.300

275

1 345

Е

E1

D

Shoulder to Shoulder Width

Molded Package Width

Overall Length

7.87

7.24

34 67

8.26

7.49

35 18



Next would be to draw the package outline. Set grid to 0.025 inch as shown.

e <u>E</u> dit <u>D</u> ra	w 🧧 Grid	X	
2 🖬 🎒 🕄	🖇 🕯 Display	Style	
21 tPla	ace 💿 On 💿 Off	🔘 Dots 🛛 🔘 Lines	Vidth:
0.05 inch	n (-1	linch - Finant	
£°	Multiple: 1		
££			
4	Alt: 0.025	inch 🔻 Finest	
A-	Default	OK Cancel	
2	Dendar		
r			
Т			
2			
	والد بن کر اور کر کر کر در در این کر ای این کر کر این کر ای کر در این کر این کر ا		
Descrip	tion		

While placing the pads care should be taken to place it in the proper sequence. In this case the pads should be placed in anti-clockwise direction starting from the left bottom.

Use the Wire tool to draw the package outline. Leave a gap in the left which will be filled up with an arc that represents the "Notch" of the IC.



Drawing the Notch using the Arc tool:



At this stage we have created both the symbol and the package diagrams using the respective editors. Now the two files have to be merged in what is known as the Device editor. Click on the button and name the Device as "TRANSISTOR".

Eile Edit Draw View Libra	ry <u>o</u>	🖉 Edit 📃
₽ ₽ ₽ 8 83 ₽ € ₩ 1	K) (6	Device
0.025 inch (-0.450 0.250)		
×		Neve: TRANSISTOR
豊 ア		Dev Pac Sym
/ T // Warning		OK Cancel
	v device	TRANSISTOR?
• • • • • • • • • • • • • • • • • • • •		.]

Add the transistor symbol using the Add tool as shown.



Place Symbol in the given space.

📕 1 Li	brary - C:\Program Files\EAGLE-5.6.0\lbr\untitled.lbr (TRANSISTOR.dev) - EAGLE 5.6.0 Professional	
<u>F</u> ile	<u>Edit D</u> raw <u>V</u> iew Library <u>O</u> ptions <u>W</u> indow <u>H</u> elp	
	▋ ♣ 器 驫 & Щ Щ Q Q Q Q Q ∽ ~ መ ▮ ?	
1 1111		
: •••	0 1 inth (2 0 1 2)	
i 🤨	0.1101(0.0-1.3)	
🐂 Lž		
44		
1 1 A		
	Package	
82		
Tok	COLLECTOR	
a a a a a a a a a a a a a a a a a a a		
	EUTTER	
	- N	lew Connect
	* Pr	refix
	Description Technologies Attributes	0.0 % 0.00
	TRANSFERRE	le on On

Next click on New and add the transistor package:



The IC package diagram appears as shown. Click on Connect button to link the pins of the symbol with the pads of the package. This step completes the Device creation.

Now the package has 4 pins whereas the transistor has only 3 leads viz. Base, Collector and Emitter. So 3 pins can be connected and the last pin can be left unconnected.

We connect the pins in the following way:

PAD 1 $\leftarrow \rightarrow$ BASE PAD 2 $\leftarrow \rightarrow$ N/C PAD 3 $\leftarrow \rightarrow$ EMITTER PAD 4 $\leftarrow \rightarrow$ COLLECTOR



Now save the file at any preferred location in your hard drive. Before using this new library file in the schematic editor, the following actions are to be performed:

In the schematic editor,

- 1. Click on Library
- 2. Select Use: A window appears where the library file has to be located from hard drive and selected.
- 3. Select "Update All" which will add the new library file to Eagle's library database.



The new user defined component is now ready to be used in the schematic editor!

This completes the basic introductory tutorial of Eagle CAD and the rest of the finer details are left for the users' exploration!

HAPPY PCB MAKING 🙂