# Circuit Design Using **EAGLE** Software

# 1.0 Introduction

EAGLE software is electronic design automation software created by AUTODESK. EAGLE stands for Easily Applicable Graphical Layout Editor. It enables printed circuit boards designers to easily create a layout of the board in the form of schematic diagram, component placement, PCB routing and comprehensive library content [1].

## 2.0 Features

Creating Schematic:Drawing circuit diagrams are very easy with EAGLE by using their schematic editor. Circuit diagrams can be populated with components and parts using an inbuilt component and device library. Parts and components can be placed on many sheets and connected together through ports. The schematics are stored in a file with extension .SCH. Components are connected through ports of the components and line. This line can be defined within the software. For example analog ground (AGND) can be defined and can be connected together. Similarly high potentials such as 3V and 5V lines can be defined which are connecting similar component ports or inputs.

Creating PCB: The printed circuit board (PCB) layout editor helps the designer with the layout of the circuit board which includes the placement of components and parts. EAGLE is a very interactive software and allows the designer to easily change the layout of the board. PCB editor stores the files with the extension .BRD.

Creating Gerber: Once the PCB layout is designed, it is ready to be printed on the circuit board printer or milling machine. Printer software usually requires gerber files which have all the information about the components, their connections with other components via traces, drill holes and other details and silk screen. These are standard file formats which are accepted by PCB fabrication companies. Some softwares save gerber file as extension .grb. Files that are generated by the EAGLE are as follows:

- 1. TOP copper layer (.cmp) or component layer
- 2. BOTTOM copper layer (.sol) or solder layer
- 3. TOP Silk Screen layer (.plc)
- 4. BOTTOM Silk Screen layer (.pls)
- 5. TOP Solder mask layer (.stc)
- 6. BOTTOM Solder mask layer (.sts)
- 7. Layer of internal cuts (.gko)
- 8. Perforation layer (.ncd)
- 9. Drilling configuration file (.drl)

Circuit boards can be milled at our department, Department of Electrical Computer and Biomedical Engineering (ECBE), Toronto Metropolitan University, Toronto. Most important files are .cmp, .sol and .drl.

# 3.0 Printer

ECBE uses LPKF s103 ProtoMat S series mill/drill plotter system. The plotter uses the motor speed of 40,000 to 100,000 revolutions per min (rpm) and moving at 150 mm/s speed. It prints high-end and complex PCBs very fast with professional results. It also provides high precision up to 0.26 micro-meter. This plotter/milling machine can provide double-sided circuit boards. With high grade of automation, this machine is capable of automatically exchanging tools, head illumination and camera detection of fiducials. It uses patented software called CircuitPro for PCBs. This software is able to import several CAD formats and is able to generate production data from these files.

# 4.0 Copper Boards

ECBE uses FR4 copper boards for the milling of PCBs. Epoxy resins are widely used in electronic application because they are virtually impervious to moisture [2]. It has outstanding mechanical and electrical strength as well as good dielectric properties. Some of the properties of this material are moisture resistance, excellent flammability rating, very good insulation and electrical properties.

# 5.0 Schematic and PCB Design

## 5.1 Creating a New Schematic Drawing

Follow these simple steps to draw a schematic using EAGLE software.

1. Click: Applications  $\rightarrow$  Engineering  $\rightarrow$  Eagle 6.5

Eagle Control Panel window will open, as shown in Figure 1..



Figure 1: Eagle software Control Panel used for designing schematic or PCB layout.

#### 2. Click: File $\rightarrow$ New $\rightarrow$ Schematic

This will open a new window for schematic drawing as shown in Figure 2.



Figure 2: Schematic window for drawing a new schematic.

3. Stretch the window to adjust viewing comfortability, as shown in Figure 3.



Figure 3: Schematic window stretched for better viewing and for ease of drawing.

#### 4. Click: View $\rightarrow$ Grid $\rightarrow$ Display On

Schematic window will create a grid which makes it easier for the designer for the placement of components. Schematic window with grid is shown in Figure 4.



Figure 4: Eagle schematic panel with grid.

Grid for the schematic window can be changed. Format can be changed between in dots and lines, mm and inches and even the distance between dots can be changed using the grid window as shown in Figure 5.

#### 5.2 Adding Components

Components can be added to the schematic drawing from the add menu which is on the side panel. Figure 6 shows the add component icon on the side panel. Once clicked it will open another window with all general and some specific components depending on the requirement..

Applications Places S	ystem 🗎 🌃							
ି GI	rid	×					1 Schem	natic -
Display On C Off	Style C Dots C	Lines	low <u>H</u> elp ■、 ■↓   い ∩	× 💷 🕌	? desig	link 🔻		
Size: 0.1	inch 💌	<u>F</u> inest						
Multiple: 1		Finant				\$		
Alt: 0.01	lincn 💌	Finest	_					
<u>D</u> efault	ОК	Cancel						
× ☆ ↓ ↓ ☆ ※ 器 端 / / 7 88 / / 7 0 つ ■ 』 1 1 1 1 1 1 1 1 1 1 1 1 1								

Figure5: Grid panel for adjusting the grid format and distance between the lines/dots.



Figure 6: Side panel icon to add components on schematic drawing.

 Click on Add menu icon on side panel or components can also be added using Eagle menu Edit → Add... This will open another Add window. These are the components which are present within the library of Eagle 6.5. The designer can also design their own components, but that topic is out of scope for this tutorial.



Figure 7: Add menu with the list of components.

- 2. Will try to add LM555 Timer chip with power and ground and some other components to our drawing. Select LM555 from the library as shown in Figure 8 and click **OK**. Figure 9 shows the component being selected from the library and placed on the drawing.
- Figure 10 shows 2 more components added to the drawing. More components can be added depending on the design. Figure 11 shown the connection between the components using wire from the side menu.

Applications Places System	m 🗟 🔟 💽 🛐 🔜 🖬 🖬 🖬 🖬 🖬 🖬	D	🔀 🚨 Tue May 16, 14:33 🗸 🐟 🗴
Name         //           10 Cold         10 Cold           14 Abox         44 Sox           14 Abox         44 Sox           17 Abox         74 Abox           17 Abox         74 Abox           17 Abox         10 Abox           17 Abox         10 Abox           17 Abox         10 Abox           18 Abox         10 Abox           19 Abox         10 Abox           10 Abox         10 Abox	Solid Parkets ADD Series     Solid Parkets     S	A A C C C C C C C C C C C C C	
UA555N     UA555N     UA555N     UA555N     UA555N     UA555N     UA555N     UA5     UA555N     UA5     UA555N     UA55     UA550	DL08-ROUND DIODE 3.5mm Terminal block 4-pin connector 2.1mm 5.5.5mm THM DC jack with internal switch. Digikey part #PJ-102A PMP Transistor PMP Transistor PMP Transistor PMP Stanson Contemporation (Contemporation (Contemporation)) PCL08-700002204/206	Package: DUC0-ROUND Dual In Line Package	
B 40-XX - 40PNFPC B 40PNFPC B 74:125 - 74:	OMRON SWITCH Ound bus BUFFER, 3-state DiL14 DiL14 ADDO Octal BUS TMARGEERVER, 3-state Postive VOITAGE REGULTOR gds ₽ Description ₽ Preview		
Attributes D	3	Cancel	يلم.

Figure 8: Selecting LM555 from the library and clicking OK.

incations Flat	ces System 🔙 📶	<b>Ç</b> 🔨 🙁				1 Sci	hematic	- /home	e/faculty	/nmalh	otr/unti	tled.sch	- EAGL	E 6.5.0	Profes	sional					2	🗿 Tue M	4ay 10
t <u>D</u> raw <u>V</u> iev	w <u>T</u> ools <u>L</u> ibrary <u>O</u>	ptions <u>W</u> indow <u>H</u> e	elp		dasīna —																		
🍩 🖙   ö  1	1/1 🗾 🗰 1021 1021	49998	<b>n</b> ~   <b>U</b>	8 3 .	dnk 🕇																		
iheet: 🗗 🗙 0.	1 inch (1.7 3.3)																 	 	 	 	 		
1																							
								-															
							IC1																
							2 TB	Q	3														
							400	DIC	7														
							5	+	6														
							° CV	THR															
							1 GN	ID V+	8														
							LM5	55N															
1																							

Figure 9: Component placed on the Eagle drawing.



Figure 10: Add more components depending on design requirement.



Figure 11: Connect the components using wire from the left menu.

#### 5.3 Save Schematic Drawing

This is a good time to save the schematic. The Eagle software saves the file with .sch extension. Click **File**  $\rightarrow$  **Save** and another window will open and click Save as shown in Figure 12.



Figure 12: Saving schematic drawing to home drive.

#### 5.4 Converting Schematic design to PCB

Figure 11 shows a very simple schematic (for demonstration purpose). It is believed the reader will have more components on the schematic drawing. Once the drawing is populated with required components, the next step is to convert the schematic into a PCB layout. This can be performed using **Generate/Switch to board** icon on the top bar as shown in Figure 13.



Figure 13: Show the top menu icon to convert the schematic into a PCB layout or to switch between schematic drawing and PCB layout.

Once the icon is clicked, a warning message will appear, this warning is because Eagle soft is generating a PCB based on schematic drawing for the first time. Usually, this icon is to switch between schematic drawing and PCB layout. Figure 14 shows the PCB layout.

- 1. It is recommended to turn on the grid, same way it was turned on for schematic design.
- Move components for the PCB design, when placed it will provide exactly same layout on the circuit board after printed on a milling machine. Figure 14 shows the layout of the components.
- Click on Route icon to make connection between the component pads. This will be the actual connection and will be printed on the PCB. It can be noticed that wires are converted into routes (yellow lines to blue lines) as shown in Figure 16..



Figure 14: Window showing PCB design layout area. Components are not placed and this is a raw layout.



Figure 15: Use Move icon to move the components to desired location.



Figure 16: Route is used to create the connection between different pads of different components or of the same component as needed.

**NOTE**: Remember wires can be routed and rerouted as the need arises, but wires cannot be deleted in the PCB layout, it has to be deleted in schematic.

# 6.0 Generating Gerber Files

Generally Eagle generates files such as schematic and board files (.sch and .brd respectively). These files are specific to Eagle software, however gerber files are required for PCB manufacturing [3]. This section will provide steps to generate gerber files for manufacturing.

#### 6.1 Generating Drill Files

To create a gerber file in Eagle, drillcfg command needs to be executed.

 Go to File → Run ULP → Drill configuration window will open up, click "drillcfg.ulp", and press Open, to generate corresponding configuration file, as shown in Figure 17.

🔊 Applications Places System 📄 🔟 🛛 🚺		🔣 🚨 Wed May 17, 13:40
•	2 Board - /home/faculty/nmalhotr/lpkf/manuals/Project.brd - EAGLE 6.5.0 Professional	× * ×
File Edit Draw View Tools Library Options Window Help		
_ = = = = : * * ™(≈) < < < < < < = * * * * * * * * * * * * *		
0.05 junk ( 0.30 4.00)		
i 💿		
• E		
<u>&gt;</u>	· Run 🛞	
X &	Look in: 🚔 /usr/local/eagle-6.5.0/ulp 🔍 🔾 🔾 🖓 🖬 🗐	
21) H4	nmalhotr designlink-inc.ulp editnext-lbr.ulp	
1	Computer designlink-lbr.ulp editnext-sheet.ulp	
ā r	dif40.ulp	
	dose-pro.up editprev-sheet.up	
սեր Գլ «Ն	drillegend-stack.ulp ex-dialogs.ulp	
	dxf.ulp extfile_copy.ulp	
	e-bauteil-erstellen.ulp ex-input-file.ulp	
	e-klemmenplan.ulp	
	e-makelist.ulp e-packages-aus-devices-pin-ist-padname.ulp exp-descr-script.ulp	
	edit-used-dev-pac-sym.ulp editnext-dev-sym.ac.ulp editnext-dev-sym.ac.ulp	
×#		
Q €	File name: drillcfg.ulp Open	
•	Eller of hunor Uncert and uncertainty (1 ulp)	
	rifes of type. Josef Language Programs (*.up/	
	<b>⋰</b> ⊈ີ∕_ <b>୲</b>	
· · · · · · · · · · · · · · · · · · ·		
		•
🔯 🌾 [Control Panel - EAGLE 🔛 Project.sch 🛛 🛃 Project.brd		

Figure 17: Open ULP to select drillcfg.ulp

2. Once you click **Open**, another window will appear, as shown in Figure 18, click

**OK**. A confirmation window will appear as shown in Figure 19, do not change anything and click **OK**.

3. Finally, its time to save the drill file. Figure 20 shows the file name and location of the drill file to be saved in the directory.

🔊 Applications Places System 📃 🔟 🚺 🚺		🛐 🚨 Wed May 17, 13:40
•	2 Board - /home/faculty/nmalhotr/lpkf/manuals/Project.brd - EAGLE 6.5.0 Professional	×
File Edit Draw View Tools Library Options Window Help	80	
	harr.	
0.05 inch (-0.30 4.00)		
÷ 11		
ф ф		
× 🐅		
10 ··1		
10 冬 花		
фr	Eagle: Drill Configuration	
	- Select unit for output file - OK	
N. N.		
/ T	Quit	
	N	
	IC1	
	─ <b>॔</b> ſ <u>×</u> <b>e</b>	
<u> </u>		
I I		2
Run: drilletg.ulp		
🖾 🖓 Fiojectoro		

Figure 18: Drill configuration file for units, mm or inch.

🔊 Applications Places System 📃 🔟 🛛 💽 🎦	2 Based theme for ultriamally beter field an analytic field and the CAPLE SEA Developed and	🔀 🚨 Wed May 17, 13:40
File Edit Draw View Tools Library Options Window Help	2 Board - /nome/racuity/nmainotr/ipkr/manuals/Project.brd - EAGLE 6.5.0 Professional	
0.05 inch (-0.30.4.00)		×
- ○ 上 - 中 - 中 - 中 - 中 - 中 - 中 - 中 - 中	• Eagle: Edit Drill Configuration         Edit only if you are sure what you do!         ID1 0 dlmm         TD2 1 02mm         Cox         Cancel	
Run: drillofg.ulp		•

Figure 19: Confirmation window to ensure the units.

② Applications Places System 교 및 문자 영상 다 문화 File Edit Draw View Tools Library Options Window Help 그 문자 중 중 위 및 또 문 및 및 및 수 및 및 수 ~ ● 및 ? ~~~~~ 및 및	2 Board - /home/faculty/nmalhotr/lpkf/manuals/Project.brd - EAGLE 6.5.0 Professional	🄀 🙆 Wed May 17, 13:41
<ul> <li>○ 0 os heh (0.304.00)</li> <li>○ 2 os heh (0.304.00)</li> <li>○ 2 os heh (0.304.00)</li> <li>○ 2 os heh (0.304.00)</li> <li>○ 3 be (0.304.00)</li> <li>○ 4 be (0.304.00)</li> <li>○ 5 be (0.304.00)</li> <li>○ 6 be (0.304.00)</li> <li>○ 6 be (0.304.00)</li> <li>○ 7 be (0.304.00)</li> <li< th=""><th>Save Configuration File</th><th></th></li<></ul>	Save Configuration File	
P C C C C C C C C C C C C C C C C C C C	Files of type: I dd	
📷 : 🦸 [Control Panel - EAGLE 🖫 Project.sch 🔛 Project.brd		

Figure 20: Window to save the drill file.

## 6.2 Generating Drill Files





Figure 21. Open CAM processor for more gerber files.

2. Go to  $\textbf{File} \rightarrow \textbf{Open} \rightarrow \textbf{Job}$  ..., click on excellon.cam and press OK, as shown in

Figure 22. Click Open, then click **Process job**, as shown in Figure 23.



Figure 22: Open CAM job, using excellon.cam file.



Figure 23: CAM processor, for generating corresponding drillfiles.

#### 6.2 Generating Gerber Files

- 1. From the top menu go to File  $\rightarrow$  CAM Processor, as shown in Figure 21.
- 2. Go to File  $\rightarrow$  Open  $\rightarrow$  Job ..., click on gerber274x.cam and press OK, as shown

in Figure 24. Click Open, then click **Process job**, as shown in Figure 25.



Figure 24: Open gerber274x.cam for generating gerber files.

3. Once the process is complete, all the drill files and gerber files can be seen in

your home directory. Files that are important for milling PCB board are as

follows:

- a. .cmp (Component side file)
- b. .dri (drill file)
- c. .drl (drill file)
- d. .sol (solder side file)

If it is a single sided board only solder side file is required, however if it is double sided board than component and solder sides are required for PCB.

Applications Places System	stem 🗃 🎹 📃 📆 🛃			🕺 🚨 Wed May 17, 15:48
			8 Board - /home/faculty/nmalhotr/lpkf/manuals/Project.brd - EAGLE 6.5.0 Professio	nal 🔍 🗟 🕺
3 CAM Processor - /u	ısr/local/eagle-6.5.0/cam/ge	erb274x.cam - 🗉 🕙 🙆 🛞		
File Layer Window Help				
Component side Solder	side Silk screen CMP Solder	r stop mask CMP Solder		
Job	Style	Nr 🛆 Layer 🔺		
Section Component side	□ <u>M</u> irror	1 Top 16 Bottom		
Prompt	□ Rotate	17 Pads 18 Vias		
Output	Upside down	1 19 Unrouted 20 Dimension		
Device GERBER_RS	274X Y Dos. Coord	21 tPlace		
		23 tOrigins		
File %N.cmp	Fill pads	25 tNames		
Offect		20 Divames 27 tValues		
V Disch		28 bValues 29 tStop		
X Dinch		30 bStop 31 tCream		
		32 bCream 33 tFinish		
_	6	34 bFinish 35 tGlue		
·		· · · · · · · · · · · · · · · · · · ·		
Proce	ess lob Process Section Descri	iption Add Del		
/home/faculty/nmalhotr/lpkf/ma	anuals/Project.brd	ll.		
ы <b>н</b>				
X#				
4 4				
•				
			ž 👷 👷	
		5 👸	😤 🏅 2 👸	
		0		
1				
📷 : 🦸 New_Project	Project.brd	Project.sch	CAM Processor	

Figure 25: CAM job for generating gerber files.

#### References

.

- 1. <u>https://www.autodesk.ca/en/products/eagle/overview?term=1-YEAR&tab=subscription#what-is-eagle-</u>
- 2. <u>https://dielectricmfg.com/resources/knowledge-base/glass-epoxy/</u>
- 3. https://www.pcbway.com/helpcenter/technical\_support/Generate\_Gerber\_files\_in \_Eagle.html