

# A Basic Introduction for Designing a Printed Circuit Board (PCB) with EAGLE eCAD/CAM Software

**by Kenneth Kwashnak** *SURVICE Engineering 4695 Millennium Drive Belcamp, MD* 21017

under contract W911QX-16-D-0014

Approved for public release; distribution is unlimited.

#### NOTICES

#### Disclaimers

The findings in this report are not to be construed as an official Department of the Army position unless so designated by other authorized documents.

Citation of manufacturer's or trade names does not constitute an official endorsement or approval of the use thereof.

The research reported in this document was performed in connection with contract/instrument W911QX-16-D-0014 with the DEVCOM Army Research Laboratory.

The findings in this report are not to be construed as an official Department of the Army position unless so designated by other authorized documents. The views and conclusions contained in this document are those of SURVICE Engineering Company and the DEVCOM Army Research Laboratory.

Destroy this report when it is no longer needed. Do not return it to the originator.





# A Basic Introduction for Designing a Printed Circuit Board (PCB) with EAGLE eCAD/CAM Software

by Kenneth Kwashnak SURVICE Engineering 4695 Millennium Drive Belcamp, MD 21017

under contract W911QX-16-D-0014

Approved for public release; distribution is unlimited.

	REPORT D	OCUMENTATIO	N PAGE		Form Approved OMB No. 0704-0188
Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data source data needed, and completing and reviewing the collection information. Send comments regarding this burden estimate or any other aspect of this collection of information, include burden, to Department of Defense, Washington Headquarters Services, Directorate for Information Operations and Reports (0704-0188), 1215 Jefferson Davis Highway, Suite 1 Respondents should be aware that notwithstanding any other provision of law, no person shall be subject to any penalty for failing to comply with a collection of information if i valid OMB control number.				structions, searching existing data sources, gathering and maintaining the et of this collection of information, including suggestions for reducing the , 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, mply with a collection of information if it does not display a currently	
1. REPORT DATE (DI	D-MM-YYYY)	2. REPORT TYPE			3. DATES COVERED (From - To)
December 2020		Contractor Report	t		September 2019–November 2020
4. TITLE AND SUBT	TLE ction for Designi	ing a Printed Circui	it Board (PCB) w	vith EAGLE	5a. CONTRACT NUMBER
eCAD/CAM So	ftware		( ) .		5b. GRANT NUMBER
					5c. PROGRAM ELEMENT NUMBER
Kenneth Kwash	nak				Su. PROJECT NOWIDER
					5e. TASK NUMBER
					5f. WORK UNIT NUMBER
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) SURVICE Engineering 4695 Millennium Drive Belcamp, MD 21017					8. PERFORMING ORGANIZATION REPORT NUMBER
9. SPONSORING/M	ONITORING AGENCY	(NAME(S) AND ADDRE	SS(ES)		10. SPONSOR/MONITOR'S ACRONYM(S)
DEVCOM Arm	y Research Labo	oratory	. ,		
ATTN: FCDD-RLW-PA					11. SPONSOR/MONITOR'S REPORT NUMBER(S)
Aberdeen Provin	ng Ground, MD	21005			ARL-CR-0855
<b>12. DISTRIBUTION</b> /Approved for pu	<b>AVAILABILITY STATE</b> ablic release; dist	<b>MENT</b> tribution is unlimite	ed.		
<b>13. SUPPLEMENTAR</b> ORCID ID(s): K	RY NOTES Kenneth Kwashna	ak, 0000-0002-120	3-6487		
<b>14. ABSTRACT</b> Printed circuit boards (PCBs) are ubiquitous elements of modern electronic systems. In DoD research, PCBs play a vital in creating reliable, economic, and deterministic circuits, apart from their bread-board or perf-board counterparts. The art transitioning a circuit's schematic to a physical manufacturable board can be challenging, but electrical computer-aided d (eCAD) and computer-aided manufacturing (CAM) software, such as the Easily Applicable Graphical Layout Editor (EAGLE), provides the creator with complete control of the circuit's design and streamlines the PCB fabrication process. report serves as a general introduction to formulate a custom circuit schematic and two-layer PCB, as well as to process necessary files for design production.				ns. In DoD research, PCBs play a vital role oard or perf-board counterparts. The art of enging, but electrical computer-aided design applicable Graphical Layout Editor reamlines the PCB fabrication process. This I two-layer PCB, as well as to process	
15. SUBJECT TERMS				manufacturing, PCB design	
			17. LIMITATION	18. NUMBER	19a. NAME OF RESPONSIBLE PERSON
16. SECURITY CLASSIFICATION OF:			OF	OF	Kenneth Kwashnak
a. REPORT	b. ABSTRACT	c. THIS PAGE		PAGES	19b. TELEPHONE NUMBER (Include area code)
Unclassified	Unclassified	Unclassified	00	203	(410) 278-5680

Standard Form 298 (Rev. 8/98) Prescribed by ANSI Std. Z39.18

# Contents

List	of Fig	gures		vi
List	of Ta	bles		xiv
List	of Co	de Bloc	:ks	xiv
Ack	nowl	edgmer	nts	xv
1.	Intr	oductio	n	1
	1.1	Prototy	yping	1
	1.2	eCAD a	and PCBs	3
	1.3	Purpos	se and Intent	4
	1.4	Equipn	nent	4
		1.4.1	Computer	4
		1.4.2	Circuit Programming and Testing Hardware	5
		1.4.3	PCB Component Hardware	6
		1.4.4	Software	7
		1.4.5	MPLAB X IDE and XC8 Compiler	10
	1.5	Unders	standing the Walkthrough	10
2.	EAG	iLE Quic	ck-Start	10
	2.1	Installi	ng the Software	10
	2.2	Creatir	ng a File Structure	16
	2.3	Using t	he EAGLE Control Panel	16
		2.3.1	File	17
		2.3.2	View	22
		2.3.3	Options	22
		2.3.4	Window	23
		2.3.5	Help	23
	2.4	Config	uring the Directory	24
	2.5	Creatir	ng a Schematic and Board	26
	2.6	Config	uring the Design Review Check (DRC)	32

3	Crea	ting a Component	39
	3.1	Creating a Library	39
	3.2	Creating a Symbol	42
	3.3	Package	51
	3.4	Device	58
	3.5	Activating the Library	63
4.	Two	-Layer PCB Design: General Design Cycle	65
	4.1	Step 1: Define OS File Structure	66
	4.2	Step 2: Configure Directories	66
	4.3	Step 3: Generate a New Project	68
	4.4	Step 4: Create a New Schematic	69
	4.5	Step 5: Design the Schematic	70
		4.5.1 Editing Imported Symbols	76
		4.5.2 Creating Grounded Mounting Hole Symbols	78
	_	4.5.3 Changing Component Designation	83
	4.6	Step 6: Map Nets and Connections	85
	4.7	Step 7: Run Electronic Review Check	88
	4.8	Step 8: Create a New Board Layout	90
	4.9	Step 9: Define PCB Settings	92
	4.10	Step 10: Organize Board Components	97
	4.11	Step 11: Create Copper Regions	99
	4.12	Step 12: Route Traces	104
	4.13	Step 13: Edit Silkscreen	107
	4.14	Step 14: DRC	109
	4.15	Step 15: Create Fabrication Files	115
		4.15.1 Creating the Drill List .drl	115
		4.15.2 Creating Drill Location Information .drd and .dri	117
		4.15.3 Creating Layer Information (Gerber274x.cam or Custom)	120
	4.16	Step 16: Formulate Board Instructions	137
	4.17	Step 17: Package Files	138
	4.18	Step 18: Send to Manufacturer	141

	4.18.1 Advanced Circuits	141		
	4.18.2 Oshpark	149		
5.	Reception and Testing of PCB	155		
	5.1 Reception of Oshpark Boards	155		
	5.2 Assembling and Testing PCB Power	156		
	5.3 Extended Testing	157		
6.	Conclusion	164		
7.	References	166		
Арр	pendix A. List of Manufacturing References	169		
Арр	Appendix B. Hypertext Markup Language (HTML) 171			
Арг	Appendix C. Sample Library Device Hypertext Markup Language (HTML) Description 175			
Арр	pendix D. Advanced Circuits Manufacturing Specifications	178		
Арр	pendix E. Oshpark Manufacturing Specifications	180		
Ado	ditional Reading	184		
List	of Symbols, Abbreviations, and Acronyms	185		
Dist	tribution List	186		

# List of Figures

Fig. 1	Example of a bread-board circuit	1
Fig. 2	Example of a perf-board circuit	1
Fig. 3	Example of an assembled PCB	1
Fig. 4	Example two-layer PCB, hexapod.brd	3
Fig. 5	CadSoft EAGLE v5.3.0 professional edition	7
Fig. 6	v5.3.0 control panel	8
Fig. 7	v9.6.2 control panel	9
Fig. 8	v5.3.0 board workspace	9
Fig. 9	v9.6.2 board workspace	9
Fig. 10	EAGLE 5.3.0 CD contents	11
Fig. 11	Installation executable	11
Fig. 12	File extraction splash screen	12
Fig. 13	Setup welcome screen	12
Fig. 14	CadSoft EULA	12
Fig. 15	Install directory location	13
Fig. 16	Setup finish	13
Fig. 17	Window's charm	14
Fig. 18	Opening EAGLE application	15
Fig. 19	EAGLE control panel	15
Fig. 20	File directory structure	16
Fig. 21	EAGLE control panel tabs	17
Fig. 22	EAGLE control panel, file tab	17
Fig. 23	Creating a new project	18
Fig. 24	Example of a schematic	18
Fig. 25	Example of a board	19
Fig. 26	Example of a schematic	19
Fig. 27	Example CAM processor	20
Fig. 28	Example ULP	20
Fig. 29	Example script	21
Fig. 30	Example text editor	21
Fig. 31	View tab	22

Fig. 32	Options tab	
Fig. 33	Window tab	
Fig. 34	Help tab	
Fig. 35	Directory, initial setup	
Fig. 36	Directory, sourced pathways	
Fig. 37	Projects directory structure	
Fig. 38	Creating a new project	
Fig. 39	Active project	
Fig. 40	Inactive project	
Fig. 41	Creating a schematic	
Fig. 42	Schematic editor	
Fig. 43	Save button	
Fig. 44	Display (layer) icon	
Fig. 45	Schematic layer view and values	
Fig. 46	Layer properties	
Fig. 47	Schematic/board icon	
Fig. 48	Warning message, create board file from schematic	
Fig. 49	Board editor	
Fig. 50	Board editor DRC	
Fig. 51	DRC	
Fig. 52	DRC description	
Fig. 53	DRC layers	
Fig. 54	DRC clearance	
Fig. 55	DRC distance	
Fig. 56	DRC sizes	
Fig. 57	DRC restring	
Fig. 58	DRC shapes	
Fig. 59	DRC supply	
Fig. 60	DRC masks	
Fig. 61	DRC misc	
Fig. 62	Creating a new library	
Fig. 63	Library editor	40
Fig. 64	Opening a library	
Fig. 65	Base library file directory	

Fig. 66	Libraries file structure	42
Fig. 67	Library edit button	43
Fig. 68	Edit window	43
Fig. 69	Symbol edit window	44
Fig. 70	Symbol editor	44
Fig. 71	View tab	45
Fig. 72	Grid settings	45
Fig. 73	Rectangle on 94 symbol layer	46
Fig. 74	Pin creation	46
Fig. 75	Group select tool with highlight area	47
Fig. 76	Moving multiple objects	47
Fig. 77	Edit pin information	48
Fig. 78	Completed symbol	49
Fig. 79	Symbol annotations	49
Fig. 80	Completed symbol	51
Fig. 81	Package creation	52
Fig. 82	Package editor	52
Fig. 83	Grid settings	53
Fig. 84	Component physical outline	53
Fig. 85	Pin 1 padded hole placement	54
Fig. 86	Pin 2 padded hole placement	54
Fig. 87	SMD pad information	55
Fig. 88	Silkscreen outline	55
Fig. 89	Package annotations	56
Fig. 90	<device> icon</device>	58
Fig. 91	Device creation	58
Fig. 92	Device configuration <add> tool</add>	59
Fig. 93	Select created symbol	59
Fig. 94	Symbol placement	60
Fig. 95	Adding a package to the symbol	60
Fig. 96	Package assignment	61
Fig. 97	Connecting pins	61
Fig. 98	Initial pin assignment	62
Fig. 99	Associate pin to pad	62

Fig. 100	Pin assignment complete	. 62
Fig. 101	Device creation complete	. 63
Fig. 102	New library and associated files	. 64
Fig. 103	Activate library	. 64
Fig. 104	PCB design cycle	. 65
Fig. 105	EAGLE control panel tree	. 66
Fig. 106	EAGLE control panel directories	. 67
Fig. 107	Directory locations	. 67
Fig. 108	Example EAGLE control panel file tree	. 68
Fig. 109	pcb_2layer project creation	. 68
Fig. 110	Active project	. 69
Fig. 111	Creating a new schematic	. 69
Fig. 112	Schematic file in tree	. 70
Fig. 113	Inactive default libraries	. 70
Fig. 114	Activating all libraries	. 71
Fig. 115	Adding a frame to schematic	. 71
Fig. 116	Frame placement	. 72
Fig. 117	Adding MICROTEST-DIP_8 to the schematic	. 72
Fig. 118	Frame and component in schematic editor	. 73
Fig. 119	<text> tool</text>	. 75
Fig. 120	Custom schematic text	. 76
Fig. 121	Raw imported symbol	. 76
Fig. 122	Edited symbol	. 76
Fig. 123	<smash> tool on object</smash>	. 77
Fig. 124	Moving smashed text	. 77
Fig. 125	mount_hole library	. 78
Fig. 126	MOUNT-HOLE symbol circle 1	. 79
Fig. 127	MOUNT-HOLE symbol circle 2	. 79
Fig. 128	MOUNT-HOLE symbol infill	. 80
Fig. 129	MOUNT-HOLE symbol text	. 80
Fig. 130	SYM-MOUNT-HOLE device creation	. 81
Fig. 131	SYM-MOUNT-HOLE device complete	. 81
Fig. 132	SYM-MOUNT-HOLE object	. 82
Fig. 133	Via used as plated through hole	. 82

Fig. 134	>NAME \$U1 component designation	. 83
Fig. 135	<prefix> Option</prefix>	. 83
Fig. 136	Defining line properties	. 84
Fig. 137	<label> Tool</label>	. 84
Fig. 138	Connecting component with <wire> tool</wire>	. 85
Fig. 139	Renaming nets and defining signals	. 85
Fig. 140	Merging nets	. 86
Fig. 141	Completed schematic	. 87
Fig. 142	<erc> tool</erc>	. 88
Fig. 143	ERC dialog	. 88
Fig. 144	Warning list	. 89
Fig. 145	Approved ERC	. 90
Fig. 146	<board> icon</board>	. 91
Fig. 147	Changing board workspace background color	. 91
Fig. 148	Design rule check on completed board	. 92
Fig. 149	DRC dialog	. 92
Fig. 150	Layers	. 93
Fig. 151	Clearance	. 93
Fig. 152	Distance	. 94
Fig. 153	Sizes	. 94
Fig. 154	Restring	. 95
Fig. 155	Shapes	. 95
Fig. 156	Supply	. 96
Fig. 157	Masks	. 96
Fig. 158	Misc	. 97
Fig. 159	Organizing components	. 97
Fig. 160	Board layout	. 98
Fig. 161	Placing component on bottom of board	. 98
Fig. 162	Recalculating airwire locations	. 99
Fig. 163	<polygon> tool</polygon>	. 99
Fig. 164	<polygon> top copper layer</polygon>	100
Fig. 165	<polygon> bottom copper layer</polygon>	100
Fig. 166	Polygon information	101
Fig. 167	Polygon signal name	101

Fig. 168	Renaming signal error
Fig. 169	<name> Tool on bottom layer</name>
Fig. 170	Formulating copper layers 103
Fig. 171	<ripup> tool, delete all signals 103</ripup>
Fig. 172	Cleared workspace
Fig. 173	Creating traces with the <route> tool 104</route>
Fig. 174	Trace properties
Fig. 175	Trace top layer
Fig. 176	Trace bottom layer 105
Fig. 177	Via
Fig. 178	<auto> Tool settings</auto>
Fig. 179	Auto-routed board
Fig. 180	Top silkscreen
Fig. 181	Updating libraries
Fig. 182	Completed board
Fig. 183	DRC check 110
Fig. 184	DRC errors 111
Fig. 185	<cut> tool 111</cut>
Fig. 186	Create a new package
Fig. 187	<paste> tool</paste>
Fig. 188	Device management
Fig. 189	Variant name
Fig. 190	Configure pins
Fig. 191	Final board 114
Fig. 192	<run> ULP option</run>
Fig. 193	drillefg.ulp
Fig. 194	Unit select 116
Fig. 195	Board drill sizes
Fig. 196	CAM processor 117
Fig. 197	CAM job 117
Fig. 198	excellon.cam 118
Fig. 199	Generate drill data
Fig. 200	Configuring file save location
Fig. 201	Process job generate drill data

Fig. 202	Creation of drd and dri files	120
Fig. 203	gerb274x.cam	121
Fig. 204	Gerber_RS274X top silkscreen section	122
Fig. 205	Gerber_RS274X top stop section	123
Fig. 206	Gerber_RS274X top copper section	124
Fig. 207	Gerber_RS274X bottom copper section	125
Fig. 208	Gerber_RS274X bottom stop section	126
Fig. 209	Gerber_RS274X bottom silkscreen section	127
Fig. 210	Gerber_RS274X board outline section	128
Fig. 211	Gerber_RS274X drills and holes section	129
Fig. 212	Gerber_274X description section	130
Fig. 213	Gerber_274X description information	130
Fig. 214	Save Gerber_274X job	131
Fig. 215	Processing Gerber files	132
Fig. 216	Gerber files complete	132
Fig. 217	Creating an instruction file	137
Fig. 218	Instructions text file	137
Fig. 219	Fab folder location	139
Fig. 220	Compressing to a .zip file	139
Fig. 221	Compressed fab file	140
Fig. 222	Renaming compressed file	140
Fig. 223	Advanced circuits welcome screen	142
Fig. 224	Advanced circuits main screen	142
Fig. 225	Advanced circuits FreeDFM	143
Fig. 226	FreeDFM sample results	143
Fig. 227	Advanced circuits quote specifications, page 1	144
Fig. 228	Advanced circuits quote specifications page 2	145
Fig. 229	Quote number	146
Fig. 230	Quote information	147
Fig. 231	Advanced circuits DFM, page 1	148
Fig. 232	Advanced circuits DFM, page 2	148
Fig. 233	Oshpark welcome screen	149
Fig. 234	Project board .brd file	149
Fig. 235	Oshpark instant quote	150

Fig. 236	Oshpark board top	151
Fig. 237	Oshpark board bottom	151
Fig. 238	Oshpark drills	152
Fig. 239	Oshpark top silkscreen	153
Fig. 240	Oshpark bottom layer	153
Fig. 241	Oshpark top solder mask	
Fig. 242	Oshpark top layer	
Fig. 243	Oshpark board outline	155
Fig. 244	Oshpark bottom solder mask	155
Fig. 245	Oshpark fabricated board	156
Fig. 246	Testing power on prototype PCB	157
Fig. 247	MPLAB X welcome screen	158
Fig. 248	Creating a standalone project	158
Fig. 249	Project creation	159
Fig. 250	Supported debug header	159
Fig. 251	Cross compiler toolchain	160
Fig. 252	Project name and location	160
Fig. 253	Creating main.c source file	161
Fig. 254	main.c code compilation	163
Fig. 255	PCB programming	
Fig. 256	Completed functional microcontroller PCB	165
Fig. C-1	Hypertext Markup Language (HTML) example	

# List of Tables

Table 1	Prototype method comparison2
Table 2	Virtual computer information
Table 3	Circuit programming and testing hardware
Table 4	PCB hardware
Table 5	Schematic layer settings
Table 6	EAGLE board editor settings
Table 7	DRC, layer tab, setup definitions
Table 8	Example component pin assignments
Table 9	Symbol element properties 50
Table 10	Package element properties
Table 11	Schematic custom two-layer board objects74
Table 12	Auto-router directions
Table 13	Filename alternatives
Table B-1	Basic HTML 172
Table B-2	HTML formatting
Table B-3	HTML color codes
Table D-1	Advanced circuits manufacturing specifications

# List of Code Blocks

Code block 1	HTML gerber_274X description	131
Code block 2	Custom CAM job for Excellon file	133
Code block 3	Custom CAM job for Gerber files	136
Code block 4	Instructions file content	138
Code block 5	PIC12F1501 code	163
Code block 6	HTML code for sample library device description	177

# Acknowledgments

I would like to thank Dr Thomas Kottke, Christopher Wolfe, and Steven Davis for reviewing this report. In addition, Dr Kottke has a wealth of electrical engineering knowledge, and I appreciate his insights on printed circuit board design and microcontroller programming.

#### 1. Introduction

#### 1.1 Prototyping

Bread- and perf-boards are great mediums for prototyping circuits. In prototyping hardware, the design process may evolve from a bread-board (Fig. 1) to a perfboard (Fig. 2) and then to a printed circuit board (PCB) (Fig. 3). The main reason to start with a bread-board is to plan a functional concept. Bread-boards are easy to work with and can expedite testing of a design. Next, a perf-board implementation provides a more rigid and responsive circuit. Finally, perf-board design is transformed into a PCB solution to finely tune and optimize the circuit. The goal of a PCB is to create a reliable, deterministic, and well-packaged device.



Fig. 1 Example of a bread-board circuit



Fig. 2 Example of a perf-board circuit



Fig. 3 Example of an assembled PCB

Table 1 provides the advantages and disadvantages of using each method.

Method	Pro	Con
Bread- Board	<ul> <li>Fast build for simple small circuits</li> <li>Relatively inexpensive</li> <li>Easy to swap out components</li> <li>No expensive soldering equipment, requires few, if at all, hand tools</li> <li>Inner rail system easy to connect components</li> </ul>	<ul> <li>Difficult to implement large scale systems</li> <li>Layout can get very messy</li> <li>Components not fully secured, fragile</li> <li>Difficult to control signal integrity</li> <li>Stuck with predefined hole pattern</li> <li>Limited on what type of parts to use without creating intricate adapters</li> <li>One sided design</li> <li>Difficult to reproduce many units</li> <li>Reused bread-board holes can wear out</li> </ul>
Perf- Board	<ul> <li>More secured and ridged design due to soldered / fastened components</li> <li>Incorporates a wider range of components, such as surface mount devices (SMDs)</li> <li>Two sided design</li> <li>Hobbyist look and quality</li> </ul>	<ul> <li>Layout can still get very messy</li> <li>Stuck with predefined hole pattern</li> <li>Difficult to disassemble</li> <li>Requires soldering</li> <li>Difficult to produce many units</li> </ul>
РСВ	<ul> <li>More secured and ridged design due to soldered / fastened components</li> <li>Incorporates an even wider range of components, since pad placement can be defined, such as ball grid arrays (BGAs)</li> <li>Add durable text, labels, and graphics to top and bottom layer</li> <li>Multi-layered design</li> <li>Precision Deterministic circuit behavior</li> <li>Can be rather large projects</li> <li>Professional look and quality</li> <li>Plug design files in CNC machine to fab boards</li> <li>Acid etching boards can produce prototypes</li> </ul>	<ul> <li>Requires soldering</li> <li>Usually requires eCAD software</li> <li>Send files out for manufacturing, requires time</li> <li>Boards, depending on complexity or requirements, can get costly for low quantity board count</li> </ul>

#### Table 1Prototype method comparison

## 1.2 eCAD and PCBs

The Easily Applicable Graphical Layout Editor (EAGLE) is an electrical computeraided design (eCAD) and computer-aided manufacturing (CAM) software. EAGLE supports the designer in transitioning prototype hardware to a more controlled precision design.

PCBs are electrical circuits that are etched on assorted laminates via conducting material, such as copper. The copper branches through various layers of insulating dielectric to form interconnected networks. Figure 4 represents a board layout from EAGLE's example files.



Fig. 4 Example two-layer PCB, hexapod.brd

The light blue and light green lines signify traces that connect the component's pins together. The light purple circles are the vias, which allow the traces from the top layer to connect with bottom layer. Various black outlines assist to define the overall shape of the board and components.

As seen in Fig. 4, a PCB can easily become crowded. Optimizing the overall placement and routing of components is critical in interconnecting hardware and ensuring component functionality. In some designs, multiple layers are required to control power or signal planes. Yet, adding multiple layers and creating additional vias will drive the overall cost of the board, as well as many other factors, such as number of drilled holes, different tool changes to drill those holes, size of the board, thickness of copper material, type of insulating material, and trace tolerances.

#### 1.3 Purpose and Intent

The layout of a PCB is an art form. The placement of traces is not usually as simple as drawing a straight line from one pin to another. When laying out traces for highprecision circuitry, line impedance, thermal expansion, layer capacitance, and signal integrity are just a few critical factors to consider in the design.

Unfortunately, electrical design, cost-reduction, and circuit optimization are beyond the scope of this paper. This report provides a basic introduction on how to compile a simple circuit, prepare a PCB for manufacturing, and recognize the fundamental controls of the EAGLE PCB software.

The designer of a modeled circuit wants the PCB to behave exactly the way it was intended to function. Using eCAD software, the circuit's design is systematically controlled to follow stringent PCB manufacturing requirements. This allows the designer to govern deterministic circuit behavior (i.e., signal integrity, timing of signals), stable board properties (i.e., mechanical rigidity, thermal relief, overall layer capacitance), ease of assembly (i.e., component spacing, clearance room, solder mask pad size), and to neatly and professionally package the product.

Circuit design may become very complex, depending on the application, interconnection of systems, and definitions of components. This walkthrough will guide the reader with a simple two-layer PCB design. This report 1) discusses material for resource creation (schematic symbols, board layout footprints, compiled devices, and component libraries), 2) reviews the methodology for fabricating a microcontroller board, and 3) presents the assembly and testing of a fabricated microcontroller PCB prototype.

## 1.4 Equipment

This section highlights the equipment for the design, assembly, and testing of the PCB.

## 1.4.1 Computer

This report refers to the computer equipment listed in Table 2, which includes an Ubuntu Linux 16.04 operating system (OS) laptop that utilizes VirtualBox (version 5.1.38) to run an isolated instance of Windows 7 Professional x64 OS. The virtual environment recognizes an external CD drive that is directly connected to the laptop (Kanguru, PN: U2-DVDRW-SL). The CD drive is necessary to install the software (Section 1.4.4) to the virtual Windows 7 OS.

Windows 7 professional x64 OS virtual computer				
Manufacturer	Dell			
Model Dell precision M6800				
Processor	Intel(R) Core (TM) i7-4910MQ @ 2.9GHz			
Installed memory 16 GB				

Table 2Virtual computer information

The reader should have a working proficiency with navigating and using the Windows 7 and the File Explorer interface. In addition, the reader should have general knowledge of computer hardware, resources, and troubleshooting.

The profile on the Windows 7 virtual machine has administrator rights in order to install and configure the EAGLE software.

#### 1.4.2 Circuit Programming and Testing Hardware

Equipment included in Table 3 describes the tools used to assemble and test the fabricated PCB. Assembly is required to test the functionality of the fully populated board, but not necessary for the initial design of the PCB. Yet, when creating a PCB, the designer must be aware of the assembly process and must incorporate viable pathways for tooling and physical component handling.

Line	Description	Manufacturer	PN	Q	Cost per item (\$)	Total (\$)
1	Lead free no clean solder wire 0.031 in Tin/Cpr Core 66	Kester <sup>1</sup>	24-9574- 7618	1	55.39ª	55.39
2	Soldering station, 35W	Aoyue <sup>2</sup>	936	1	42.75 <sup>b</sup>	42.75
3	No clean flux paste rosin	MG Chemicals <sup>3</sup>	8341-10ML	1	16.28 <sup>a</sup>	16.28
4	DC linear power supply, regulated, 0-30V, 0-5A	Korad <sup>4</sup>	KD3005D	1	85.00°	85.00
5	Banana to alligator clip cable (red and black wires)	B&K Precision <sup>5</sup>	TL 5A	1	10.10 <sup>a</sup>	10.10
6	Multimeter, true RMS	Fluke <sup>6</sup>	117	1	219.99ª	219.99
7	Desolder braid no clean, rosin, non activated (R), LF	Chemtronics <sup>7</sup>	60-5-5	1	6.55ª	6.55
8	Assorted hook-up wire	Sparkfun <sup>8</sup>	PRT-11375	1	16.95ª	16.95
9	Cable stripper and cutter, $20 \sim 30 \text{ AWG}$	Hakko <sup>9</sup>	CSP-30-1	1	14.06ª	14.06
10	Tweezer set	Wiha <sup>10</sup>	44593	1	207.96	207.96

 Table 3
 Circuit programming and testing hardware

<sup>a</sup> Data from Digikey on October 8, 2020

<sup>b</sup> Data from Aoyue3d on October 8, 2020

<sup>c</sup> Data from SRA Soldering Equipment on October 8, 2020

<sup>d</sup> Data from Amazon on October 8, 2020

Line	Description	Manufacturer	PN	Q	Cost per item (\$)	Total (\$)
11	Isopropyl, cleaning agent	MG Chemicals <sup>11</sup>	824-1L	1	16.75ª	16.75
12	Brush	MG Chemicals <sup>12</sup>	855-5	1	4.45 <sup>a</sup>	4.45
13	PICKit3	Microchip <sup>13</sup>	PG164130	1	25.00 <sup>d</sup>	25.00
14	PICKit3 programming cable	Digilent14	240-035	1	3.99ª	3.99
15	ESD caliper	Wiha <sup>15</sup>	41105	1	74.56 <sup>a</sup>	74.56

 Table 3
 Circuit programming and testing hardware (continued)

<sup>a</sup> Data from Digikey on October 8, 2020

<sup>b</sup> Data from Aoyue3d on October 8, 2020

<sup>c</sup> Data from SRA Soldering Equipment on October 8, 2020

<sup>d</sup> Data from Amazon on October 8, 2020

#### 1.4.3 PCB Component Hardware

Components listed in Table 4 describe the basis for formulating a functioning microcontroller demo circuit. If the board is to be assembled, variations of these components may be used, but the designer must consult the component datasheet to ensure compatibility. In addition, refer to Appendix A for a list of manufacturing resources for enclosures, panel overlays, and component distributors.

Line	Description	Manufacturer	PN	Q	Cost per item (\$) <sup>a</sup>	Total (\$)
1	Microcontroller, DIP8	Microchip <sup>16</sup>	PIC12LF1501	1	0.77000	0.77
2	IC DIP socket 8POS	Adam tech <sup>17</sup>	ICS-308-T	1	0.15000	0.15
3	Battery, CR2450, 3V	Energizer <sup>18</sup>	CR2450VP	1	1.19000	1.19
4	Battery holder, coin PC pin	MPD <sup>19</sup>	BS-2450	1	1.09000	1.09
5	Regulator linear 3V, TO92-3	Microchip <sup>20</sup>	MCP1700-3002E/TO	1	0.37000	0.37
6	Green SMD LED	Dialight <sup>21</sup>	5988170107F	4	0.39000	1.56
7	Tactile switch, NO	C&K <sup>22</sup>	PTS 647 SK38 SMTR2 LFS	1	0.13000	0.13
8	Res, 0805, 120Ω, 5%, 1/8W	Vishay dale <sup>23</sup>	CRCW0805120RJNEA	4	0.10000	0.40
9	Cap, 0805, 0.1uF, 25V, X7R	KEMET <sup>24</sup>	C0805C104M3RACTU	1	0.10000	0.10
10	0.100 in header, 5 pin, prog	Molex <sup>25</sup>	22-23-2051	1	0.34000	0.34

Table 4PCB hardware

<sup>a</sup> Price from Digikey as of September 18, 2020

Line	Description	Manufacturer	PN	Q	Cost per item (\$) <sup>a</sup>	Total (\$)
11	0.100 in, conn housing, 5POS	Molex <sup>26</sup>	0022012057	1	0.23000	0.23
12	0.100 in header, 2 pin, batt	Molex <sup>27</sup>	22-23-2021	1	0.17000	0.17
13	0.100 in, conn housing, 2POS	Molex <sup>28</sup>	0022012027	1	0.11000	0.11
14	Conn 22-30AWG crimp tin	Molex <sup>29</sup>	0008650804	7	0.10000	0.70

Table 4PCB hardware (continued)

<sup>a</sup> Price from Digikey as of September 18, 2020

#### 1.4.4 Software

EAGLE v5.3.0 Software CD Package (Fig. 5) is used for the eCAD tool in designing and creating the manufacturing files for the two-layer PCB.



Fig. 5 CadSoft EAGLE v5.3.0 professional edition

CadSoft used a stand-alone licensing model until around June 2016 until Autodesk acquired CadSoft - EAGLE. The software migrated to a subscription-based, online-authenticated service, which requires the user to periodically verify their credentials with the Autodesk license server.

EAGLE v5.3.0 uses a stand-alone license, while the most recent version (v9.6.2 at the time of publishing) uses a subscription-based model. Since the virtual computer does not have Internet access, it is impossible to download, install, verify, and run the software. Therefore, an older version of EAGLE is used to design a PCB.

Users may download the most recent version of EAGLE from the Autodesk website.<sup>30</sup> A free version of the software is available, but the user must log in through the Autodesk portal. After 14 days, the user must reverify their credentials.<sup>31</sup>

In addition, there are restrictions and limitations to the license type. The End User License Agreement (EULA) states that the "free" license shall only be used for personal learning.<sup>32</sup>

Fortunately, EAGLE version 5.3.0 was purchased before Autodesk acquired CadSoft and the sole purpose of the license is to perform research and development tasks on a stand-alone non-networked Windows-based machines.

**Quick comparison: v5.3.0 and v9.6.2.** The tutorial should transfer to the most recent version of EAGLE hosted by Autodesk. The most recent version of EAGLE will be loaded with new features and tools, but the overall premise remains the same (Figs. 6–9).



Fig. 6 v5.3.0 control panel



Fig. 7 v9.6.2 control panel



Fig. 8 v5.3.0 board workspace



Fig. 9 v9.6.2 board workspace

Over the years there have been improvements to EAGLE designer software, but the core features still remain. It appears the latest version has a few more tools, graphical user interface (GUI) developments, and system tweaks. One of the major advantages of the latest version is integration with Fusion 360, a CAD tool that allows the designer to export their PCB from EAGLE and import into Fusion 360 to review the 3-D design. Newer EAGLE software also supports 3-D models for all components, which ties in with the device creation.

#### 1.4.5 MPLAB X IDE and XC8 Compiler

Microchip's MPLAB X Integrated Development Environment (IDE) v5.4.0<sup>33</sup> and XC8 Compiler v2.30<sup>34</sup> are downloaded from a separate networked computer and burned to a CD. The programming of the microcontroller is briefly discussed in Section 5.3, but ultimately beyond the scope of this report.

#### 1.5 Understanding the Walkthrough

The following conventions are used to describe various elements of the text:

<options></options>	Element in EAGLE, such as a button, drop-down menu item, or interactive object.
<u>Library</u>	Source file(s) on Windows virtual computer
C:\Users	Source file location on Windows virtual computer
'Project'	Software property, such as a title on a panel, wording used on a program prompt.
"Hello"	Configurable text, such as a file name, folder, property, etc.
[CTRL] + [V]	Buttons on the physical keyboard. If two elements are conjoined by a "+" sign, simultaneously press both buttons on the keyboard.
World	Coding block for programming

## 2. EAGLE Quick-Start

#### 2.1 Installing the Software

Insert the EAGLE v5.3.0 CD into the CD drive. Navigate to the <u>Windows</u> folder (Fig. 10) and run <u>setup.exe</u> (Fig. 11).

🕞 🕞 🗢 🚺 🕨 Kwashnak 🕨	EAGL	.E 5.3.0 CD 🕨				Search E	4G 🔎
Organize 👻 🛜 Open	Inclu	ide in library 👻	Share with 🔹	<ul> <li>New folder</li> </ul>	8==	•	0
🔆 Favorites	-	Name		Date modified	Туре	Size	
		퉬 Linux		8/18/2020 3:59 PM	File folder		
🧮 Desktop		퉬 MacOSX		8/18/2020 3:59 PM	File folder		
🥽 Libraries		🌗 Windows		8/18/2020 3:59 PM	File folder		
<u> k</u> kwashnak	E	autorun	Data creat	10/07/000010.00	ر File		1 KB
💻 Computer		autorun.inf	Date creat	10/27/2008 12:03	Setup Information		1 KB
👊 Network		📄 Info.txt		1/16/2020 11:52 AM	Text Document		1 KB
📴 Control Panel		license.key		11/26/2008 2:16 PM	KEY File		3 KB
👿 Recycle Bin		LICENSE.TXT		10/27/2008 12:03	Text Document		8 KB
🍌 Kwashnak		📄 README.TXT		10/27/2008 12:03	Text Document		1 KB
🎉 Libraries							
Drojecte	Ψ.						
Windows Date mo File folder	odified	: 8/18/2020 3:59 PM					

Fig. 10 EAGLE 5.3.0 CD contents

🚱 🔍 🗣 📔 🕨 Kwashnak I	EAGLE	5.3.0 CD   Windows		<b>-</b> ↓	Search Win 👂
Organize 🔻 🖬 Open	Share	with 🔻 New folder		=	• •
🔆 Favorites	<b>^</b>	Name		Date modified	Туре
		README_de.txt		10/27/2008 12:03	Text Document
🧮 Desktop		README_en.txt		10/27/2008 12:03	Text Document
詞 Libraries		🧟 setup.exe		10/27/2008 12:03	Application
<ul> <li>kkwashnak</li> <li>Computer</li> <li>Network</li> <li>Control Panel</li> <li>Recycle Bin</li> </ul>	ш		Date created: 8/18/202 Size: 24.5 MB	20 3:59 PM	
Kwashnak  Libraries  Roject		e			•
setup.exe Date r Application	nodified Size	: 10/27/2008 12:03 AM : 24.5 MB	Date created: 8/18/202	20 3:59 PM	

Fig. 11 Installation executable

A CadSoft dialog box appears. Click on the <Setup> button (Fig. 12). Follow the prompted instructions to install the software in a desired location on the OS (Figs. 13–15). Once the installation is completed, click on the <Finish> button (Fig. 16).



Fig. 12 File extraction splash screen



Fig. 13 Setup welcome screen



Fig. 14 CadSoft EULA



Fig. 15 Install directory location



Fig. 16 Setup finish

Open the Windows Charm (Fig. 17) and search through the programs for the <u>EAGLE Layout Editor 5.3.0</u> folder (Figs. 18 and 19). Left-mouse-click on the app to launch or right-mouse-click on the app to see a list of operations.



Fig. 17 Window's charm



Fig. 18 Opening EAGLE application



Fig. 19 EAGLE control panel

# 2.2 Creating a File Structure

Switch to the Windows File Explorer and open a file directory on the computer. Create a main directory location to house Eagle Library (ex.: .lbr) and Project Files (ex: .epf, .brd, .sch) as shown in Fig. 20.

For example:

- Main Directory: C:\Users\kkwashnak\Desktop\Kwashnak
- Eagle Libraries: C:\Users\kkwashnak\Desktop\Kwashnak\Eagle Libraries
- Eagle Projects: C:\Users\kkwashnak\Desktop\Kwashnak\Projects



Fig. 20 File directory structure

Eagle Library files (.lbr) are a collection of symbols, footprints, and devices that are compiled into configurable components used in schematic and board files. The project file (.epf) assembles information about the project and associates the respective schematics (.sch) to board files (.brd).

## 2.3 Using the EAGLE Control Panel

Return to the EAGLE v5.3.0 application Control Panel and examine the main tabs: <File>, <View>, <Options>, <Window>, and <Help>. The Control Panel (Fig. 21) is the main interface to create, navigate, and configure project resources.

Control Panel - EAGLE 5.3.0 Professional					
File View Options Wind	ow Help				
Name	Description				
b Libraries	Libraries				
Design Rules	Design Rules				
> User Language Programs	User Language Programs				
Scripts	Script Files				
CAM Jobs	CAM Processor Jobs				
Projects					
2					

Fig. 21 EAGLE control panel tabs

#### 2.3.1 File

The <File> tab (Fig. 22) contains options to create or open projects, schematics, boards, libraries, CAM jobs, scripts, and text files.



Fig. 22 EAGLE control panel, file tab

Creating a new project generates a 'Project' folder inside a specified project directory that will contain necessary schematic and board files (Fig. 23).



Fig. 23 Creating a new project

A schematic file (.sch) is the layout design of a system portraying components and their connections with other elements, such as microcontrollers, resistors, capacitors, power supply lines, ground, and signals lines (Fig. 24).



Fig. 24 Example of a schematic

A board file (.brd) is the collection of schematic elements that are placed onto a physical PCB (Fig. 25). The board file defines the overall shape and dimension of the PCB. Components are rearranged, organized, and connected to other components via traces.



Fig. 25 Example of a board

Library files (.lbr) contain schematic symbols, board component footprints, and compiled devices (associates symbols to footprints (Fig. 26).

🛃 1 Library - C:\Program Files (x86)\EAGLE-5.3.0\lbr\microchip.lbr (PIC16F870.sym) - EAGL 🗖 🔳 🔀			
File Edit Draw View Library Options Window Help			
🚘 🖬 🎒 🗱 🜉 🔃 🔍 🔍 🍳 🔍 🖓 🗠 😓 🗢 🔷 🐂 🙆			
🖉 Edit 💽			
0.1 inch (0.0 1.3)			Symbol 🔷 🔽
I 🕑			PIC12F675
<b>4</b> I.i	_≥NAME Φ		PIC12F683
		PGD/RB	PIC16C55
€P ₽	RAD/AND	PGC/RB RB	PIC16C56
C) 🏄 🚽	RA1/AN1	PGM/RB	PIC16C71
F 🔰		RB	PIC16F62X-SSOP
×	*6 R/65/AN4	INT/RB	PIC16F676
		RX/RC	PIC16F870
$\nabla$	RCD/T10SD	D/RC	New:
/ Т	PC R62/CCP1	RG	Dev Pac Sym
$\hat{0}$	495 AA	>VALUE	
<b>°</b> '			OK Cancel +
* <			
		L	

Fig. 26 Example of a schematic

A CAM job creates required files necessary for PCB fabrication (Fig. 27).
Job Section Generate drill data Prompt Output Device EXCELLON	Style Mirror Rotate Usside down Ø pos. Coord Quidqolot Ø Optimize Ø Fill pads	Nr 1 1 16 17 18 19 20 20 21 22 23 24 25 26 27 28 29 30 31 32	Layer Top Bottom Pads Vias Unrouted Dimension tPlace bPlace tOrigins bOrigins tNames tNames tValues bValues tStop bStop tCream bCream	
---	--	---	---	--

Fig. 27 Example CAM processor

User Language Programs (ULPs) (.ulp) are code that automates tedious tasks and intricate processes (Fig. 28).

🔛 1 Text Editor - C:\Program Files (x86)\EAGLE-5.3.0\ulp\dxf.ulp - EAGLE 5.3.0 Professional
File Edit Window Help
#require 4.5702
<pre>#lequire 4.302 #usage "en: <b>Export DXF data</b>'n"     ""     "Converts a board or schematic into a DXF file."     ""     "Usage: RUN dxf [ -s <i>suffix</i> ] [ -u mm inch ] [ -a ] [ -     ""     "options:      ""     ""     """     ""<tb>"<tb>"<tb>"<tb>"<tb>"<tb>"<tb>"<tb>"<tb>"<tb>"<tb>"<tb>"<tb>"<tb>"<tb>"<tb>"<tb>"<tb>"<tb>"<tb>"<tb>"<tb>"<tb>"<tb>"<tb>"<tb>"<tb>"<tb>"<tb>"<b>" "&lt;"&gt;""" "&lt;"&gt;"" ""<br< td=""></br<></b></tb></tb></tb></tb></tb></tb></tb></tb></tb></tb></tb></tb></tb></tb></tb></tb></tb></tb></tb></tb></tb></tb></tb></tb></tb></tb></tb></tb></tb></pre>
"wires drawn with their real widths." ""
"DXF syntax generated according to the specifications given in -
1:1 Ins

Fig. 28 Example ULP

Script files (.scr) are code for configuring EAGLE default values, such as grid options, custom menus, shortcuts, and drill values (Fig. 29).



Fig. 29 Example script

Text files (.txt) are a workspace for creating ULPs, scripts, or general text-based files (Fig. 30).



Fig. 30 Example text editor

#### 2.3.2 View

The <View> tab covers two options, <Refresh> and <Sort> (Fig. 31). <Refresh> restarts the Control Panel GUI contents, which allows for file discovery. <Sort> reorganizes file contents, either by file alphabetical name or file program type.

File     View     Options     Window     Help       Name     Refresh     Description       ▷     Li     Sort →     by name       ▷     Desrugmmens     by type     e Programs       ▷     Scription     Scription	EAGLE Easily Applicable Graphical Layout Editor
Name         Refresh         Description           ▷         Li         Sort         by name           ▷         Desright name         by type         e Programs           ▷         Script Files         Script Files	EAGLE Easily Applicable Graphical Layout Editor
CAM Jobs CAM Processor Jobs Projects	Version 5.3.0 for Windows Professional Edition Copyright (c) 1988-2008 CadSoft All rights reserved worldwide Single User License #650536 1DB4-LSR- W-1CP <b>Registered to:</b> U.S. Amy Research Laboratory Thomas Kattike/AMSRL-WM-TE Building 120, Army Research Lab Aberdeen Proving Ground, MD 21005

Fig. 31 View tab

# 2.3.3 Options

The <Options> tab contains <Directories>, <Backup>, <User Interface>, and <Window positions> (Fig. 32). The <Directories> option allows configuration of source file locations. <Backup> configures the auto-save feature. <User Interface> configures EAGLE GUI components. <Window positions> saves or deletes the various screen coordinates for later use.

📕 Control Pa	Control Panel - EAGLE 5.3.0 Professional					
File View	Options Window Help	_				
Name > Libraries > Design Ru > User Lang > CaM Jobs > Projects	Directories Backup User interface Window positions CAM Proce	ge Programs ssoor Jobs	EAGLE Easily Applicable Graphical Layout Editor Version 5.3.0 for Windows Professional Edition Copyright (c) 1988-2008 CadSoft All rights reserved worldwide Single User License #650536 LDB4LSR- W-LCP Registered to: U.S. Army Research Laboratory Thomas Kotker (AMSRL-WM-TE Building 120, Army Research Lab Aberdeen Proving Ground, MD 21005			

Fig. 32 Options tab

#### 2.3.4 Window

The <Window> tab allows the user to navigate through the multiple open windows in the EAGLE program (Fig. 33).

🦊 Control Panel - EAGL	E 5.3.0 Professional	
File View Options	Window Help	
File View Options Name Design Rules Design Rules User Language Progr Scripts CAM Jobs Projects	Window Help Control Panel Alt+0 Libraries Design Rules ams User Language Programs Script Files CAM Processor Jobs	EAGLE Easily Applicable Graphical Layout Editor Version 5.3.0 for Windows Professional Edition Copyright (c) 1988-2008 CadSoft All rights reserved worldwide Single User License #6505361DB4+LSR- W-1CP Registered to: U.S. Army Research Laboratory Thomas Kottke/AMSRL-WM-TE Building 120, Army Research Lab Aberdeen Proving Ground, MD 21005

Fig. 33 Window tab

# 2.3.5 Help

The <Help> tab contains general information about the EAGLE program (Fig. 34). The <General> and <Context> options both yield a wealth of information on using the program.



Fig. 34 Help tab

# 2.4 Configuring the Directory

On the EAGLE Control Panel's top ribbon, go to <Options>, <Directories>. Add the recently created <u>Projects</u> folder to EAGLE's 'Projects' pathway and add the recently created <u>EAGLE Libraries</u> folder to EAGLE's 'Libraries' pathway (Fig. 35). Add a pathway to the textbox by entering ";" after the previous text and immediately paste the Windows address (do not leave a space between the ";" and the address). Afterward, click <OK> to exit.

Directories before revising folders are shown in Fig. 35.

Libraries: *\$EAGLEDIR\lbr* 

Projects:

\$HOME\pag	alo SE AGI E	DIRInroia	cts\oramplas
prionil leus	$(e, \varphi L A U L L)$	DINDIUE	lis exumples

📕 Directories	×
Libraries	\$EAGLEDIR\/br
Design Rules	\$EAGLEDIR\dru
User Language Programs	\$EAGLEDIR\ulp
Scripts	\$EAGLEDIR\scr
CAM Jobs	\$EAGLEDIR \cam
Projects	\$HOME\eagle; \$EAGLEDIR\projects\examples
	OK Cancel Browse

Fig. 35 Directory, initial setup

Directories after revising folders are shown in Fig. 36.

- Libraries: *\$EAGLEDIR\lbr;C:\Users\kkwashnak\Desktop\Kwashnak\Eagle Libraries*
- Projects: *\$HOME\eagle;\$EAGLEDIR\projects\examples;C:\Users\ kkwashnak\Desktop\Kwashnak\Projects*

📕 Directories	
Libraries	\$EAGLEDIR\br;C:\Users\kkwashnak\Desktop\Kwashnak\Eagle Libraries
Design Rules	\$EAGLEDIR\dru
User Language Programs	\$EAGLEDIR\ulp
Scripts	\$EAGLEDIR\scr
CAM Jobs	\$EAGLEDIR\cam
Projects	\$HOME\eagle; \$EAGLEDIR\projects\examples; C:\Users\kkwashnak\Desktop\Kwashnak\Projects
	OK Cancel Browse

Fig. 36 Directory, sourced pathways

Once the Project and Library folders are included into the EAGLE directory, click <OK>. The EAGLE Control Panel will automatically refresh. If not, use <View>, <Refresh>.

In the Control Panel's element tree, click the drop-down  $< \triangleright >$  on <Projects> toplevel heading and right-mouse-click on the <Projects> folder (Fig. 37). On the dropdown menu, click <New Project> and name the project. For instance, the new folder was named "Example" (Fig. 38).



Fig. 37 Projects directory structure



Fig. 38 Creating a new project

Upon creation of the project, a red folder icon will appear and there will be a green dot next to the description. The red folder indicates a project and the green dot signifies that the project is active. Only one project can be active per EAGLE instance (one may open multiple instances of EAGLE at a time). To activate/deactivate a project, click the dot; it will cycle for green-active (Fig. 39) and gray-inactive (Fig. 40). Active translates to working on the current project and linking the project schematic with the board files.

4	Projects	
	eagle	
	examples	Examples Folder
	🔺 🥁 Projects	
	4 🍓 Example 🛛	Empty Project
	E:~ 20	A adding any is ad
	Fig. 39	Active project
4	Projects	
4	Projects     Description     Description	
đ	<ul> <li>Projects</li> <li></li></ul>	Examples Folder
4	<ul> <li>Projects</li> <li> <ul> <li>eagle</li> <li>eamples</li> <li>Projects</li> </ul> </li> </ul>	Examples Folder
4	<ul> <li>Projects</li> <li>eagle</li> <li>examples</li> <li>Projects</li> <li>Example</li> </ul>	Examples Folder • Empty Project
a	<ul> <li>Projects</li> <li>eagle</li> <li>examples</li> <li>Projects</li> <li>Projects</li> <li>Example</li> </ul>	• Empty Project

Fig. 40 Inactive project

Note, a new folder appears in the file structure:

C:\Users\kkwashnak\Desktop\Kwashnak\Projects\Example

### 2.5 Creating a Schematic and Board

To start creating a schematic, go to the top ribbon of the Control Panel, select <File>, <New>, and click on <Schematic> (Fig. 41). Once the schematic editor screen opens (Fig. 42), save the file in the project directory (Fig. 43):

📕 Control Panel - C:\Users\kkwashnak\Desktop\Kwashnak\Projects\Example - EAGLE 5.3.0 ... 👝 📧 💌 File View Options Window Help Empty Project New ۶ 🦊 Project 🗁 Open Use the context menu to create new 😨 Schematic schematic or board files within this project. Open recent projects Board Save all Eibrary Close project CAM Job Alt+X Fxit 🔄 Projects 🔺 🎆 Exa Script Text

 $C: \verb|Users\kwashnak\Desktop\Kwashnak\Projects\Example$ 

Fig. 41 Creating a schematic



Fig. 42 Schematic editor



Fig. 43 Save button

Next, click on the <Display> icon, shown in Fig. 44.



Fig. 44 Display (layer) icon

The selected layers will be visible on the schematic's workspace (Fig. 45). Edit the layer's properties by pressing the <Change> button (Fig. 46). Follow Table 5 for general layer-setting guidelines.

🕄 Disp	lay 🔀
Layers	:
Nr	Name
91	Nets
92	Busses
93	Pins
94	Symbols
95	Names
96	Values
97	Info
98	Guide
	New Change Del
	All None
	OK Cancel

Fig. 45 Schematic layer view and values

🕄 Change layer properties 🛛 💌				
Number	91 👻			
Name	Nets			
Color		Displayed		
Fillstyle		Supply Layer		
	ОК	Cancel		

Fig. 46 Layer properties

Schematic editor					
No.	Name	No.	Color		Comment
1	Nets	91	Dark green		Connection lines
2	Buses	92	Dark purple		Consists of multiple nets
3	Pins	93	Dark green		Connection point
4	Symbols	94	Dark red		Component graphics
5	Names	95	Dark gray		Component's name
6	Values	96	Dark gray		Component's value
7	Info	97	Dark gray		Supplemental information
8	Guide	98	Dark yellow		Supplemental information

Table 5Schematic layer settings

Once all schematic layer values are set, click on the <Board> icon to swap to the EAGLE Board Editor (Fig. 47).



Fig. 47 Schematic/board icon

Since the board file is extracted from the schematic view, a prompt appears as shown in Fig. 48.



Fig. 48 Warning message, create board file from schematic

Click <Yes> to create the board file from the schematic and the Board Editor will appear (Fig. 49).



Fig. 49 Board editor

Use the <Display> button to configure layer settings for the EAGLE Board Editor. Table 6 indicates the general configuration settings for the main layers in EAGLE Board Editor. Note that the convention is lighter color on the top side of board.

After reviewing the board settings, save the file in the project directory:

*C*:\*Users*\*kkwashnak*\*Desktop*\*Kwashnak*\*Projects*\*Example*.

			В	Board Editor				
No.	Layer Group	Name	No.	Color Name	(	Color	Comment	Gerber
1	Top silkscreen	tPlace	21	Light gray				.tslk
2	Green top	tStop	29	Black with Line			Keep out region	.tstp
3	Top copper	Top, pads, vias	1, 17, 18	Red, dark green, dark green				.tcpr
4	Bottom copper	Bottom, pads,	16, 17, 18	Blue, dark green, dark				.bcpr
	Dettem copper	vias	10, 17, 10	green			•••	
5	Green bottom	bStop	30	Black with Line				.bstp
6	Bottom silkscreen	bPlace	22	Dark gray				.bslk
7	Board outline	Dimension	20	Light green				.dout
8	Drills, holes	Drills, holes	44, 45	Dark gray, dark gray				.ddrl
9	Top restriction zone	tRestrict	41	Red with dots				
10	Bottom restriction zone	bRestrict	42	Blue with dots				
11	Origins, top	tOrigins	23	Light purple			Part manipulation	
12	Origins, bottom	bOrigins	24	Dark purple			Part manipulation	
13	Name, top component	tNames	25	Light gray			Appears on silkscreen	
14	Name, bottom component	bNames	26	Dark gray			Appears on silkscreen	
15	Document notes, top	tDocu	51	Light yellow			PCB physical view	
16	Document notes, bottom	bDocu	52	Dark yellow			PCB physical view	

Table 6EAGLE board editor settings

### 2.6 Configuring the Design Review Check (DRC)

On the EAGLE's board editor, click on the Design Review Check <Drc> button (Fig. 50). This option configures the board's properties and creates rules to assist with the overall structure of the PCB. These rules should be reviewed with respect to a manufacturer's requirements.



Fig. 50 Board editor DRC

In Fig. 51, after clicking on the <DRC> button, the DRC (default) window appears (Fig. 52). Edit the project's Design Rules description with the <Edit Description...> button or use the <Load> / <Save as...> button (see Appendix B, HTML, and Appendix C, Sample Library Device HTML Description). Text entry follows basic HTML formatting.

🔡 DF	RC (d	default)										×
Fil	e	Layers	Clearance	Distance	Sizes	Restring	Shapes	Supply	Masks	Misc		
EA	AGLE	Design F	tules									
Th ne	e de cess	fault Desig ary adjusti	n Rules have b ments and save	een set to co e your custom	ver a wide ized desig	range of ap In rules under	plications. Ye	our particula e.	ar design m	ay have	different requirements, so please make the	
											Edit Description.	
											Load Save as	
									Cł	neck	Select Cancel Apply	,

Fig. 51 DRC

🔛 Description of Desi	ign Rules 'default'	×
Headline:	EAGLE Design Rules	
EAGLE Design Rules	i	
The default Design Ru applications. Your part requirements, so pleas save your customized	les have been set to cover a wide icular design may have different ie make the necessary adjustmen design rules under a new name.	range of ts and
kb>EAGLE Design Rul The default Design Rul a wide range of applic may have different re necessary adjustment design rules under a n	es les have been set to cover ations. Your particular design quirements, so please make the s and save your customized ew name.	
ОК	Cancel Undo	Redo

Fig. 52 DRC description

The 'Layers' tab controls each layer's copper thickness and isolation materials (Fig. 53). The setup field defines the board's construction (Table 7).

DRC (default)									
File Layers	Clearance	Distance	Sizes	Restring	Shapes	Supply	Masks	Misc	
Layers are combined and through the second s	ned through eit ugh vias are d fined by writing + ((2 + 3) + (4) are combined t	S her core or / lefined by wig g [t::1 *16)]] is a hrough a pre	Nr 1 0.03 16 0.03 etup (1* prepreg m riting ( p), which m multilayer apreg and	5mm 5mm 16) 16) aterial. <b>a*b</b> c .). defines a blinc defines a blinc setup with tu buried vias ar	Copper combines layed d via from to vo cores, co re produced	ers a and b o to layer t nbining laye through the	with a <i>core</i> and from b ers 2/3 and resulting s	e, while a bottom to 4/16, rer	Isolation
							_		

Fig. 53 DRC layers

Table 7DRC, layer tab, setup definitions

Convention	Definition	Example
*	Core material	FR4 or similar material
+	Prepreg	Isolation material
	Via	
t::b	Blind Via	t and b are layer numbers, top and bottom, respectively

The example board is set with (1\*16), meaning a two-layer board with layers 1 and 16 paired and vias going through both layers.

The following are examples from MIT archives<sup>35</sup>:

An example of a multilayered board would be ((1 \* 2) + (15 \* 16)):

- layer set 1, layers 1 and 2 are paired
- layer set 2, layers 15 and 16 are paired
- layer sets 1 and 2 combined via prepreg material in between
- vias are drilled through the entire board

The difference between buried and blind vias is the distinction on what layers they pass through. Vias that go through the entire board, not connecting the top and

bottom layer) are referred to as buried vias, while vias that go to and from select layers are called blind vias.

The following is an example of a multilayered board with blind vias, (2:1 + ((2\*3)+(14\*15)) + 15:16):

- layer set 1, layer 2 and 3 are paired
- layer set 2, layer 14 and 15 are paired
- layer set 1 and 2 are combined via prepreg material in between
- via from layer 2 to layer 1, thus connecting layer 1 to layer 2
- via from layer 15 to layer 16, thus connecting layer 15 to layer 16

Also, blind vias can be through inner layers, (2:1 + (3:2 + (3\*4) + 5:4) + 16:5):

- layer set 1, layer 3 and 4 are paired
- via from layer 3 to layer 2
- via from layer 5 to layer 4
- via from layer 16 to layer 5
- via from layer 2 to layer 1

The 'Clearance' tab describes the minimum distance between object signal layers (Fig. 54).

	Distance Siz	zes Restring	Shapes	Supply	Masks	Misc			
					Differ	ent Sian	als		
		Wire			Direr	circ Sign			
	Wire 8	Bmil				Pad			
	Pad 8	Bmil		8mil					Via
	Via 8	Bmil		8mil				8mil	
					Sam	e Signal	s		
	ond a	Smd		01		Pad		Orail	Via
	Siliu d	DITII		omi				omi	
linimum Clearance between the Same Signals check bet etting the values for the San	en objects in signal la ween <i>Smd</i> and <i>Via</i> de <b>ne Signals</b> checks t	ayers. loes not apply to <i>M</i> to 0 disables the re	<i>licro Vias.</i> spective che	ck.					

Fig. 54 DRC clearance

The 'Distance' tab describes the minimum distance between objects in signal layers, dimensions, and between drill holes (Fig. 55).

DRC (default)						×
File Layers Clearance Distance	Sizes Restrin	g Shapes	Supply	Masks	Misc	
Minimum Distance between objects in s Setting the value for the Copper/Dimen	Copper/Dimension 4 Drill/Hole 8 ignal layers (pads, sm sion check to 0 disabl	mil nil ds and any cop	per connecte	ed to them	and the board dimensions, and	d between drill holes.
				d	eck Select	Cancel Apply

Fig. 55 DRC distance

The 'Sizes' tab describes the minimum width of any objects in the signal layers and drill holes (Fig. 56).

BRC (default)	
File Layers Clearance Distance Sizes Re	string Shapes Supply Masks Misc
Minimum Width	10mil
Minimum Drill	24mil
Min. Micro Via	9.99mm
Min. Blind Via Rai	0.5
_ <b>-</b> +    <b></b> -	
Minimum Sizes of objects in signal layers and of drill holes	
Minimum Width and Minimum Drill may be overwritten	by larger values in the Net dasses for specific signals.
Min. Micro Via applies to <i>blind</i> vias that are exactly one la the default value of 9.99mm) means there are no micro vias	er deep. Typical values are in the range 50100 micron. A value larger than Minimum Drill (e.g.
Min. Blind Via Ratio defines the minimum drill diameter da	blind via must have if it goes through a layer of thickness t. Board manufacturers usually give this
"aspect ratio" in the form 1:0.5, where 0.5 would be the	value that has to be entered here.
	Check Select Cancel Apply

Fig. 56 DRC sizes

The 'Restring' tab describes the width of the copper ring that has to remain after the pad or via that has been drilled (Fig. 57).

File Layers Clearance	e Distance Sizes	Restring	Shapes Supply	Masks Misc		
			Min	%	Max	Diameter
	Pads	Тор	10mil	25	20mil	
		Inner	10mil	25	20mil	
		Bottom	10mil	25	20mil	
	Vias	Outer	Bmil	25	20mil	
		Inner	Bmil	25	20mil	
	Micro Vias	Outer	4mil	25	20mil	
J		Inner	4mil	25	20mil	
<b>Restrings</b> for pads and vias larger restring, that value will	are defined in percent of be used in the outer laye	the drill dia rs.	meter (limited by Min and	Max). If the diameter of an	actual pad or via would res	ult in a
If the <b>Diameter</b> option is ch <b>Micro Vias</b> are <i>blind</i> vias tha may be overwritten by a large	ecked the actual pad or v at are exactly one layer di er <b>Drill</b> value in the <i>Net c</i>	ia diameter eep and ha <i>(asses</i> ).	will be taken into account	in the inner layers, too. naller than the <b>Minimum D</b> i	rill value defined under <i>Siz</i> i	<i>es</i> (which

Fig. 57 DRC restring

The 'Shapes' tab describes the geometry of SMDs and pads (Fig. 58).

H	DRC	(default)										×
	File	Layers	Clearance	Distance	Sizes Restr	ng Shapes	Supply	Masks	Misc			
				Sn	nds	N	ſin			%	Max	
				/	Roundness	Omil		0			Omil	
				Pa	ds				sł	nape		
					Тор	As in library						•
					Bottom	As in library						•
					First	Not special						•
					Elongation %	100		100				
	Shap	es of pads	and smds.									
								C	neck	Select	Cancel	Apply

Fig. 58 DRC shapes

The 'Supply' tab describes the geometry of the thermal and annulus symbols used in the supply layers (Fig. 59).

DRC (	(default) Layers	Clearance	Distance	Sizes	Restring	Shapes	Supply	Mas	ks Misc	1			
						Min	]		%			Max	
			Gap	20	mil			50			100mil		
			Ther	mal 10	Is	olate			Restring				
$\left  \right\rangle$	X		Annu	ulus 20	mil								
Suppl If a Ro	ly symbols a	are generated for	for pads and via Annulus, the	Generat as in sup resultin	e thermals fo oply layers. g supply sym	bol will be a	fully filled	circle inst	tead of a rin	g.			
The T	ap is define	plate paramet	er will also be u	sed for	signal polygo	iu ridx). ns							
NOTE:	The actual	shape of suppl	ly symbols may	be diffe	rent when g	enerating ou	itput for j	photoplot	ters that use	specific ther	mal/annul	lus aperture	:s!
									Check	Select		Cancel	Apply

Fig. 59 DRC supply

The 'Masks' tab describes the dimensions of the solder stop and cream masks (Fig. 60).

File	Lavers	Clearance	Distance	Sizes	Pestring	Shanes	Supp	w Masks	Misc	1			
T IIC	Layers	Cicuranice	Distance	01203	resuring	onupes	Joupp	y	Plac				
					N	۹in			%			Max	
			J Ston	4mil				100			4mil		
					1								
			, Crea	am Omil				0			Omil		
			Limit	: Omil									
			-										
	11	1											
	11												
Masl	li k values are	defined in perc	:ent of the smal	ller dime	ension of smd	ls, pads and	vias (lim	ited by <b>Min</b> a	nd Max).				
Masl Stop	k values are	defined in pero	rent of the smal	ller dime those vi	ension of smd	ls, pads and a drill diame	vias (lim	ited by <b>Min</b> a	nd Max). iit.				
Masl Stop	n k values are masks are g	defined in pero	cent of the smal mds, pads and	ller dime those vi	ension of smd ias that have	ls, pads and e a drill diame	vias (lim eter that	ited by <b>Min</b> a exceeds Lim	nd Max). iit.				
Masl Stop Erea	k values are masks are g m masks are	edefined in pero generated for s e generated for	" cent of the smal mds, pads and r smds only.	ller dime those vi	ension of smd ias that have	ls, pads and e a drill diame	vias (lim eter that	ited by <b>Min</b> a exceeds <b>Lim</b>	nd Max). iit.				
Masl Stop Crea	n masks are masks are <u>o</u> m masks are	edefined in perce generated for s e generated for	" cent of the smal mds, pads and r smds only.	ller dime those vi	ension of smd ias that have	ls, pads and e a drill diam	vias (lim eter that	ited by Min a exceeds Lim	nd Max). iit.				
Masl Stop Crea	n masks are <u>c</u> n masks are <u>c</u>	defined in pero generated for s e generated for	in cent of the smal inds, pads and r smds only.	ller dime those vi	ension of smd	ls, pads and e a drill diam	vias (lim eter that	ited by Min a exceeds Lim	nd Max). iit.				
Masl Stop Crea	n masks are <u>o</u> masks are <u>o</u> m masks are	defined in percent generated for s e generated for	" cent of the smal mds, pads and r smds only.	ller dime those vi	ension of smd ias that have	ls, pads and e a drill diame	vias (lim eter that	ited by Min a exceeds Lim	nd Max). iit.				
Masl Stop Crea	li k values are i masks are <u>o</u> im masks are	defined in percongression of the second s	" cent of the smal mds, pads and r smds only.	ller dime those vi	ension of smd ias that have	ls, pads and a drill diam	vias (lim	ited by Min a exceeds Lim	nd Max). iit.				
Masl Stop Crea	li k values are i masks are <u>c</u> im masks are	defined in pero generated for s e generated for	" cent of the smain ands, pads and r smds only.	ller dime those vi	ension of smd ias that have	ls, pads and a drill diam	vias (lim eter that	ited by Min a exceeds Lim	nd Max). iit.				
Masi Stop Crea	II k values are masks are g m masks are	e defined in perc generated for s e generated fo	" cent of the smai mds, pads and r smds only.	ller dime those vi	ension of smd ias that have	is, pads and e a drill diam	vias (lim eter that	ited by Min a exceeds Lim	nd Max). iit.				

Fig. 60 DRC masks





Fig. 61 DRC misc

# 3 Creating a Component

# 3.1 Creating a Library

In order to populate a schematic and a board, one must create or use a library file, which contains component information such as symbols, footprints, and devices. The Library may contain multiple variations of a component, as well as several other devices related to the library. Typically a library file is dedicated to one manufacturer of a particular series of hardware. For example, a manufacturer may make an EAGLE library that is specific to their line of microcontroller devices.

On the EAGLE Control Panel's top ribbon, navigate to <File>, <New>, and <Library> (Figs. 62 and 63).



Fig. 62 Creating a new library



Fig. 63 Library editor

Save the library in the Eagle Libraries folder:

C:\Users\kkwashnak\Desktop\Kwashnak\Eagle Libraries

For this walkthrough, the library is named "ExampleManu".

If you need to open a library file, such as "ExampleManu", on the EAGLE Control Panel, go to <File>, <Open>, and <Library> (Fig. 64). Browse the file explorer for the specified file. Then select the .lbr and click <Open> (Fig. 65).

File	View Options Wind	low Hel	þ				
	New	•				Descriptio	in
D	Open	•	#	Project +		•	
	Open recent projects	+		Sch	ema	tic	
	Save all	Board.		rd			
	Close project		<i>[</i> ]	Library			ılder
N. I	Exit	Alt+X		CAN	/ Jo	b	isor Jobs
	iser Language Programs Jesign Rules ibraries		ULP SER	ULP	 nt		s
				Text			

Fig. 64 Opening a library

📕 Open			×
COO V 🌽 « EAGLE-5.	.3.0 ▶ lbr ▶	✓ Search lbr	Q
Organize 🔻 New fold	er	:= ▼	
Favorites	Name	Date modified	Туре 🔺
	퉬 elektro	8/18/2020 4:13 PM	File fol
🛛 📃 Desktop	19inch.lbr	4/21/2008 5:00 AM	LBR Fil
	40xx.lbr	4/21/2008 5:00 AM	LBR Fil
	41xx.lbr	4/21/2008 5:00 AM	LBR Fil
	45xx.lbr	8/18/2008 5:02 AM	LBR Fil
	74ac-logic.lbr	4/21/2008 5:00 AM	LBR Fil
	74ttl-din.lbr	4/21/2008 5:00 AM	LBR Fil
	74xx-eu.lbr	4/21/2008 5:00 AM	LBR Fil
	74xx-little-de.lbr	4/21/2008 5:00 AM	LBR Fil
	74xx-little-us.lbr	4/21/2008 5:00 AM	LBR Fil
	74xx-us.lbr	4/21/2008 5:00 AM	LBR Fil
	751xx.lbr	4/21/2008 5:00 AM	LBR Fil 👻
			E.
File n	ame:	✓ Libraries (*.lbr)	•
		Open 🔽 Ca	ancel

Fig. 65 Base library file directory

The base EAGLE directory for libraries was mounted upon installing the software as follows:

C:\Program Files (x86)\EAGLE-5.3.0\lbr

To open the recently created library file, "ExampleManu", navigate to

C:\Users\kkwashnak\Desktop\Kwashnak\Eagle Libraries

The Eagle Libraries were added to the directories in the previous steps (Fig. 66).



Fig. 66 Libraries file structure

The next steps will examine creating the Symbol (Sym), Package (Pac), and Device (Dev). A device is generated via a symbol and a package. Devices are incorporated into board and schematic designs.

# 3.2 Creating a Symbol

The Library Editor window will appear when creating or opening a Library as in Fig. 63.

Click on the <Edit> button on the left-hand tool bar (Fig. 67).



Fig. 67 Library edit button

Click on <Sym> button (Fig. 68) and type the component's name, such as "MICROTEST" (Fig. 69). This will appear on the schematic. When prompted to create a new symbol, click <YES>, and the Symbol Editor will open (Fig. 70).

Edit	×
Package	
New:	
Dev Pac Sym	
OK Canc	el

Fig. 68 Edit window

🖉 Edit 💽
Symbol
New: MICROTEST
Dev Pac Sym
OK Cancel

Fig. 69 Symbol edit window



Fig. 70 Symbol editor

Once the Symbol Editor is opened, click the <Save> icon.

Next, navigate to <View> (Fig. 71) and click on <Grid>. When finished selecting settings, click <OK> (Fig. 72). For designing multiple symbols, which will be combined on a schematic, follow a similar spacing scheme. This will assist in keeping the schematic clear and legible.

🗐 1 Librar	y - C:\Users	s\kkw	ashnak\Des	ktop\Kwas	hnak\Eagle	Librarie	s\Exa	mple	Manu	ı.lbr	(MI			×
File Edit	Draw	View	Library	Options	Window	Help								
i 🗠 🖬 é	🗃 💱		Grid			60	×   (	D	1	2				
	1 inch (1.9	•	Display/hid	e layers										<b>.</b>
i 📀		£.	Mark											
• E		۲	Show											
♣ 兆		i	Info											
		<u>Q</u>	Redraw		F2									
<i>4</i>		Q	Zoom to fit		Alt+F2									
×	······································	Ð,	Zoom in		FB									
₩ r		<u>Q</u>	Zoom out		F4									
V		<u>e</u> q	Zoom selec	t										
/т		-				-								
02														
														Ŧ
* *													Þ	
Change the	e grid settir	ngs												ы

Fig. 71 View tab

🧧 Grid			<b>X</b>
-Display	1	Style	
On	n 🔘 Off	O Dots	Lines
Size:	0.1	inch	▼ Finest
Multiple:	1		
Alt:	0.01	inch	▼ Finest
Defa	ault	ОК	Cancel

Fig. 72 Grid settings

Examine the component's datasheet and review the pin out specifications. A generic component is created for the following example.

Next, click on the <Line> tool, set the layer to <94 Symbols> and proceed to draw a rectangle in the design space (Fig. 73). While in mid-draw, right-clicking the mouse will cycle through different wire bending styles. Once completed, hit the [ESC] key on the keyboard. If the line is not placed in the correct location, use the <Move> or <Delete> tools. Exit the tools via [ESC].



Fig. 73 Rectangle on 94 symbol layer

After the rectangle is completed, click the <Pin> tool (Fig. 74). Pins denote connections to the chip. Right-click the mouse to reorient the pin placement. Exit the tool via [ESC].



Fig. 74 Pin creation

To move multiple objects at once, highlight an area of objects using the <Group> tool (Fig. 75). Then, change to the <Move> tool (Fig. 76) and right click on the highlighted objects and select 'Move: Group'.

- D - 199 (	-D- 1957 1957			2	
↓ <b>- </b>	L		}- ]4 }- [₽	d Direction: I/O → Swap	level: »
0.1 inch (-0.)	6 -0.6)				
↓ <b>∔</b> ` ] &	-Ca or	P\$1	P\$8	-CKO 0	
<u>&gt;</u>	-0 <sup>01</sup>	P\$2	P\$7		-
r	-0 <sup>01</sup>	P\$3	P\$6		
T D D	-Ca or	P\$4	P\$5	- CKO 0	

Fig. 75 Group select tool with highlight area



Fig. 76 Moving multiple objects

Click the <Info> tool and click on a pin. The <Info> tool enables changing of pin parameters (Fig. 77).



Fig. 77 Edit pin information

In the 'Properties' window, rename Pin 1, "P\$1", to "VDD". 'Position' states its coordinates in the Symbol Editor. Angle denotes the orientation of the object. Mirror creates a reflected image of the component. Direction denotes the type of pin function, such as NC (not connected), In (Input Only), Out (Output Only), I/O (either Input or Output), OC (Open Collector/Drain), Pwr (Power [i.e., VDD, GND]), Pas (Passive [i.e., resistor or capacitor]), Hiz (High Impedance), or Sup (General Supply). Continue to configure all pins as listed in Table 8 and shown in (Fig. 78).

Table 8	Example	component	pin	assignments
---------	---------	-----------	-----	-------------

Pin	Assignment
1	VDD
2	CP5
3	CP4
4	MCLR/VPP/CP3
5	CP2
6	CP1/PCLK
7	CP0/PDAT
8	VSS



Fig. 78 Completed symbol

To create the bar over Pin 4, "MCLR" enter: "!MCLR!".

Click the <Text> tool to add the following text below the symbol: ">Name" and ">Value"; set the text layer to '95 Names' and '96 Values', respectfully, using the <Info> tool (Fig.79). The ">" tells EAGLE to fetch the device's metadata when the symbol is displayed in schematic view. Save the file once the symbol is complete.



Fig. 79 Symbol annotations

Symbol element guidelines are listed in Table 9.

Parameter	Value
Grid settings: Adjust for ease of syr	nbol design and may correlate to
schematic grid settings	
Size	0.01
Multiple	1
Alt	0.005
Symbol, line: Adjust symbol outline	for schematic aesthetics
Width	0.01
Style	Continuous
Cap	Round
Layer	94 Symbols
Curve	0
Symbol, name: Text of ">NAME" is	assigned to the '95 Names' layer
near the symbol	-
Size	0.07
Ratio	8%
Font	Proportional
Layer	95 Names
Symbol, value: Text of ">VALUE" is	s assigned to the '96 Values' layer
near the symbol	
Size	0.07
Ratio	8%
Font	Proportional

Table 9Guidelines for symbol element properties

#### 3.3 Package

Click on the <Package> icon (Fig. 80).

1 Library - C:\Users	\kkwashnak\Deskto	op\Kwashnak\l	Eagle Libraries\Examplel	Manu.lbr (MI	- • •
File Edit Draw	View Library C	ptions Wind	dow Help		
		र ल ज ब	<b>u</b> n o i i i i i i i i i i i i i i i i i i	?	
	Package				
i 💿	0.5)				×
• <b>₩</b>	Pwr	VDD	VSS	Bur D	
	1/0 6	CP5	CP0/PDAT	Cio o	
<i>★</i>	1/0 80	CP4	CP1/PCLK		
Real F	1/0 G	MCERA	/PP/CP3 CP2 -		
/т 00		>NAME			
		>VALUE			
<b>∢</b>					4
Edit a package					

Fig. 80 Completed symbol

The <Package> icon will transfer the workspace to the overview screen of the library. Click on the <Pac> button, enter a Package name, for instance "DIP\_8", and click <OK> (Fig. 81). The Package name should denote the physical package of the component. When prompted to create a new package, select <Yes> and the Package Editor will appear (Fig. 82).

📕 Edit	×
Package	
New: DIP_8	
Dev Pac S	Sym
ОК	ancel

Fig. 81 Package creation



Fig. 82 Package editor

Navigate to <View>, <Grid>, and edit settings. The grid spacing should reflect the units and spacing as denoted by the component's datasheet (Fig. 83).

i⊈ ⊟ ⊜/⊧⊅⊋ ∰ +⊊ ! ∭		-/ \^   🖤 🏽 i
0.05 inch (-0.25 0.25)		
L2		
÷ %%	🧧 Grid	<b>X</b>
a <b>4</b> ≻	Display	Style
3 A	🖲 On 💿 Off	O Dots O Lines
← <u>&gt;</u> <	Size: 0.05	inch 👻 Finest
* r	Multiple: 1	
	Alt. 0.025	linch - Finant
ИТ Т	AIL: 0.025	
	Default	OK Cancel
Description		

Fig. 83 Grid settings

Click on the <Rect> tool and draw a rectangle using two left clicks: one to start the rectangle and one to end (Fig. 84). Then, using the <Info> tool, edit the rectangle's properties. Ensure the layer is set to '51 tDocu'. This layer will be used to denote the components physical body. Observe the use of symmetry to simply dimensioning.



Fig. 84 Component physical outline

To add connection mounting points, use the surface mounting pads <Smd> or padded hole <Pad> tool. For this chip, use the <Pad> tool (Figs. 85 and 86). Use the <Info> tool to configure pad properties. For pin 1 of the device, the pad's properties, 'Shape', is set to 'Square' in order to specify the orientation of the device.

🖉 1 Library - C:\Users\kkwashnak\Desktop\Kwashnak\Eagle Libraries\ExampleManu.lbr (DI 🗖 🛙 🖾						
File Edit Draw View Library Options Window	Help					
🔁 🖬 🚭 😫 📾 🕫 🖽 🖽 🔍 🔍 🕄 🕲	🗠 🗠 🎟	2				
IIII O O O Diameter: auto - Drill: 0.031496 - Angle: 0 -						
0.05 inch (-0.15 0.15)						
	Properties		<b>—</b> ———————————————————————————————————			
	Pad					
	Name	VDD				
	Position	-0.155	0.15			
<i>∉</i> <u>&gt;</u>	Angle	0				
×		Mirror				
	Diameter	auto	•			
	Drill	0.031496	•			
	Shape	Square	•			
	Thermals					
	Stop					
Description	First					
Use the DESCRIPTION command to enter a desc						
• Left-click to select object to get info for		OK	Cancel Apply			

Fig. 85 Pin 1 padded hole placement

🖅 1 Library - C:\Users\kkwashnak\Desktop\Kwashnak\Eagle Libraries\ExampleManu.lbr (DL 🗖 🔳 🖾						
File Edit Draw View Library Options Window Help						
🚘 🖩 🚭   33 🖏 北   🧱 🗒 🔍 🔍 🔍 🔍 💷   🗠 🎂 💼 📓 🥇						
IIII O O O Diameter: auto 🔻 Drill: 0.031496 🔻 Angle: 0 💌						
0.05 inch (-0.15 0.05)	0.05 inch (-0.15 0.05)					
1 P	Properties		<b>—</b> ———————————————————————————————————			
	Pad					
	Name	P\$2				
	Position	-0.155	0.05			
	Angle	0				
		Mirror				
	Diameter	auto	•			
	Drill	0.031496	•			
	Shape	Round	•			
	Thermals					
	Stop					
Description	First					
Use the DESCRIPTION command to enter a description of						
Left-click to select object to get info for		ОК	Cancel Apply			

Fig. 86 Pin 2 padded hole placement

If a <smd> pad is used, ensure the layer is set to '1 TOP' (Fig. 87). '1 TOP' is a copper trace on the top side of the board (assuming the package is made for a two-layer board).

ile Edit	Draw Vie ⊒,∣⊅⊅ (##, -	w Library Options Window Help N. [호텔 벨레 슈 슈 슈 슈 슈 슈
a 🗖 🖬	≝?≎≎∰.	<pre></pre>
	Properties	<b>EX</b>
[ 	Smd	
•	Name	P\$1
→ <sup>*</sup> <sup>*</sup> <sup>*</sup>	Position	0.55 0
∃ <b>∔</b> `	Angle	0
2 ×		Mirror Mirror
~	Smd Size	0.05 x 0.025
r	Laver	
-	Roundness	0%
T	Thermals	с <i>7</i> 8
22	Stop	
	Cream	· · · · · · · · · · · · · · · · · · ·
	Cream	
<b>`</b>		OK Cancel Apply ge.

Fig. 87 SMD pad information

Next, add an outline to the chip, using the <Line> tool (Fig. 88). Inspect the line properties and ensure layer is set to '21 tplace'. Also, some package silkscreens use a dot near the first pin of the device to help indicate the orientation of the device.

🕑 1 Library - C:\Users\kkwashnak\Desktop\Kwashnak\Eagle Libraries\ExampleManu.lbr (DI 🗖 🗉 🖾						
File Edit Draw View Library Options Window Help						
🚘 📕 🎒 33 🗱 42   🕮 📖 🔍 93 93 93 94 1 v 🖓 🌆 📱 孝						
Angle: 0 🔻 Abc 199 <sup>+</sup> 🖪 🖿						
0.025 inch (0.025 0.225)	0.025 inch (0.025 0.225)					
	Wire					
↔ ??	From	-0.15	0.225			
	То	0.15	0.225			
	Length	0.3				
<i>★</i>	Width	0.005	-			
	Style	Continuous	•			
	Сар	Round	<b></b>			
	Layer	21 tPlace	•			
	Curve	0				
		OK	Cancel Apply			
Use the DESCRIPTION command to enter a description of this package.						
◆ Left-click to select object to get info for						

Fig. 88 Silkscreen outline
Likewise to the symbol, use the <Text> tool to add ">NAME" and ">VALUE" to the package, where ">NAME" and ">VALUE" resides on the '25 tNames' and '27 tValues' layers, respectively (Fig. 89). When completed, save the package. The silkscreen on the top layer of the board will portray elements from the '21 tPlace' and '25 tNames'.



Fig. 89 Package annotations

Package element guidelines are listed in Table 10.

Parameter Value		
Grid settings: Adjust for ease of sym	bol design and may correlate to the	
component's datasheet		
Size	0.01	
Multiple	1	
Alt	0.005	
Silkscreen, circle: Pin 1 identification	marker	
Width	0.005	
Layer	21 tPlace	
Radius	0.009843	
Silkscreen, line: Chip outline		
Width	0.012	
Style	Continuous	
Cap	Round	
Layer	21 tPlace	
Curve	0	
Silkscreen, name: Text of ">NAME"	is assigned to the '25 tNames' layer	
near the footprint	-	
Size	0.05	
Ratio	8%	
Font	Proportional	
Layer	25 tNames	
Silkscreen, value: Text of ">VALUE'	' is assigned to the '27 tValues' layer	
near the footprint		
Size	0.05	
Ratio	8%	
Font	Proportional	
Layer	27 tValues	

# Table 10 Guidelines for package element properties

### 3.4 Device



After completing the package, navigate to and click on the <Device> icon (Fig. 90).

Fig. 90 <Device> icon

Create a new device with the chip's name and package type (Fig. 91), for example, "MICROTEST-DIP\_8".

🗐 1 Library - C:\Users\kkw	ashnak\Desktop\Kwashnak\Eagle Libraries\ExampleManu.lbr (DI 🗖 🔳 🔀
File Edit Draw View	Library Options Window Help
🚘 🗖 🎒 🗱 🗘	₩₩ �,₽,9,9,0, ∽ ~ 100 ₩ ?
1	🖉 Edit
0.05 inch (-0.50 0.1	· · · · · · · · · · · · · · · · · · ·
1 1 1 1	Device
· → ·····	
×	
B r	
$\overline{V}$	
<u>/</u> т	New: MICROTEST-DIP_8
	Dev Pac Sym
Description	OK Cancel
Use the DESCRIP	
USE UIE DESCRIP	Tow command to enter a description of dils package.

Fig. 91 Device creation

I Lib File E	rary - C:\Users\kkwashnak\Desktop\Kwashnak\Eagle Libraries\Examp dit Draw View Library Options Window Help   番 33 篇 む 罠 罠 間 間 鸟 免 気 気 図 トゥ マ መ	pleManu.lbr (MI 💼 📼 💌
	0.1 inch (-2.6 0.1)	Package
Add a p	Description     Technologies     Attribute       Use the DESCRIPTION command to enter a description of this device.     MICROTEST- DIP_8     DIP	Y New Connect Prefix Y Value  ⓐ Off ◎ On .:i

Inside the device configuration, navigate to and click on the <Add> tool (Fig. 92).

Fig. 92 Device configuration <Add> tool

Previously created symbols will appear in the selection window (Fig. 93). In this case, "MICROTEST" appears. Click on the symbol and click <OK>.

🕘 1 Library - C:\Users\kkwashnak\Desktop\Kwashnak\Eagle Libraries\ExampleManu.lbr (MI 👝 🔳 🔯
File Edit Draw View Library Options Window Help
🚘 🖩 🚭   33 🖏 tù   🕮 📖 🔍 🔍 🍳 🔍 🔍 🔍 🗠 🖓 🕘 🥊 🦉
Addlevel: Next 🗸 Swaplevel: 0 🗸
0.1 inch (-2.6 0.6)
Add 🔀
Symbol MICROTEST
X
R2 Tok
ackage
• • • • • • • • • • • • • • • • • • •
OK Cancel New Connect
Description Prefix Prefix
Use the DESCRIPTION command to enter a description of this device.

Fig. 93 Select created symbol

The MICROTEST symbol will float inside the workspace. Click to place and [ESC] to exit (Fig. 94).

1 Libra File Ed	ary - C:\Users\kk lit Draw Viev	washnak\Desktop\k w Library Optic	(washnak\Eagle L ons Window	.ibraries\Examp Help	ppleManu.lbr (MI 🗖 🔍 🗾	<b>X</b>
i 🚗 🖪	a 33 🕮 i			ທີດ 💷	2	
Ad	Idlevel: Next	<ul> <li>Swaplevel: 0</li> </ul>	•		· · · ·	
i ◎ ■ Ш 中 × 梁 ≋ 置	0.1 inch (0.0 0.0)	P NAME P NAME P VALUE			Package	•
	•				New Connect	
▲   eft_c	Description Use the DESCRI enter a descrip	IPTION command t tion of this device.	<ul> <li>Technologies</li> <li>MICROTEST-</li> <li>DIP_8</li> </ul>	<u>s</u> <u>Attributes</u>	S Prefix ▼ Value	1
Left-ci	ick to place MIC	NUTEST				н

Fig. 94 Symbol placement

Navigate to and click on the <New> button (Fig. 95).



Fig. 95 Adding a package to the symbol

All packages that are designed in the library appear in the 'Packages' window (Fig. 96), in this case, "DIP\_8". Click on the package and click <OK>. The 'Variant Name' bar may be used if you wish to assign multiple packages to a symbol.

Create new package variant for MICROTEST-DIP	_8
Packages DIP_8	>NAME
Variant name	OK Cancel

Fig. 96 Package assignment

Navigate and click on the <Connect> button (Fig. 97).

🖉 1 Libr File Er	ary - C:\Users\kkwas dit Draw View   🚭   33 🖽 42	hnak\Desktop\Kwashnak\Eag Library Options Windov 题 삕 이 옥 유 오 오 6	jle Libraries\ExampleN v Help 및   ⊷ ္ ( ᡂ 🎇	1anu.lbr (MI 👝 💷 🗾
i ● ■ ■ ■ ■ ■ ■ ■ ■ ■ ■	0.1 inch (0.9 -1.0)	VDD VSS CP5 CP0/PDAT CP4 CP1/PCLK MCLR/VPP/CP3 CP2 >NAME >VALUE	-O** -O** -O** -O** -O** 	Package DIP_8
	Contract of the second seco	ON command to of this device.	gies <u>Attributes</u>	New Connect Prefix Value @ Off O On

Fig. 97 Connecting pins

Match up connections from the 'Pin' and 'Pad' columns, one selection at a time. Once each Pin and Pad is selected, press the <Connect> button (Figs. 98–100). Configure all pins and pads. Afterward, click <OK>.

l	Connect '' (DIP_8)		
	Pin	Pad	Connection
	Name	Name	Pin
	G\$1.!MCLR!/VPP/CP3	IMCLRI/VPP/CP3	
	G\$1.CP0/PDAT	CP0/PDAT	
	G\$1.CP1/PCLK	CP1/PCLK	
	G\$1.CP2 G\$1.CP4	CP2 CP4	
	G\$1.CP5	CP5	
	G\$1.VDD	VDD	
	G\$1.VSS	VSS	
			۰ ( الس
	Cor	nect	Disconnect
	Copy from:		
			OK Cancel

Fig. 98 Initial pin assignment

Connect " (DIP_8)		×
Pin	Pad	Connection
Name	Name	Pin
G\$1.!MCLRI/VPP/CP3 G\$1.CPV/PDAT G\$1.CP1/PCLK G\$1.CP2 G\$1.CP4 G\$1.CP5 G\$1.VDD G\$1.VS5	IMCLRI/VPP/CP3 CP0/PDAT CP1/PCLK CP2 CP4 CP5 VDD VS5	< <u> </u>
	Connect	Disconnect
Copy from:		
		OK Cancel

Fig. 99 Associate pin to pad

Connect " (DIP_8)		
Pin	Pad	Connection
Name	Name	Pin
		G51.WCLRI/VPP/CP3 G51.CP0/PDAT G51.CP1/PCLK G51.CP2 G51.CP4 G51.CP5 G51.VDD G51.VS5
Con	nect	Disconnect
Copy from:		
		OK Cancel

Fig. 100 Pin assignment complete



🗐 1 Libr	rary - C:\Users\kkwashi	nak\Desktop\Kv	vashnak\Eagle	Libraries\Example	eManu.lbr (l	MI 🗖 🗖 🗾
File E	dit Draw View L	ibrary Optior	ns Window	Help		
🚘 🗖	😂 😫 🛱 🕹 🗎	U 🖳 🎯 🕀	<u>, q</u> 🔍 🚳	<b>∽</b> ∩ 💷	2	
	0.1 inch (0.3 -0.7)					-
∎ ¥₽	Add-Next Swap-D				<u>^</u>	
<b>→</b> 1	PWY	VDD	VSS	VSewo		
∏ ∦ X⊉	10 6 <u>5</u> 5	CP5	CP0/PDAT	CRAFDAT		>\ALUE
R2 10k	10 <u>6</u> <u>7</u> 4	CP4	CP1/PCLK	CRAFCLK	=	
ä	MCER/VPR/623	MCER/VPF	P/CP3 CP2	ငၕၣၗၜ	Packa	ge
					DIP_8	
		>NAME				
		>VALUE				4
	•			Þ	Ne	w Connect
	Description Use the DESCRIPTIO	N command to	Technologie MICROTEST-	<u>s</u> <u>Attributes</u>	Pre	:fix
	enter a description of	of this device.	D1F_0		▼ Value	● Off   On

Fig. 101 Device creation complete

# 3.5 Activating the Library

Move to the EAGLE Control Panel and find the "ExampleManu.lbr" library under 'Libraries'. Notice how there is a grayed out circle, signifying that the library file is not active (Fig. 102).

Control Panel - C:\Users\kkwashnak\Desi	ktop\Kwashnak\Projects\Example - EAGLE 5.3.0 👝 💷 📧
File View Options Window Help	
Name	Description
▲ Libraries	
<ul> <li>Ibr</li> <li>Eagle Libraries</li> <li>Image Among A</li></ul>	Libraries
第第 MICROTEST-DIP_8 型 DIP_8	
Design Rules     User Language Programs     Scripts	Design Rules User Language Programs Scrint Filer
CAM Jobs     CAM Jobs     Projects     Projects	CAM Processor Jobs
<ul> <li>examples</li> <li>Grojects</li> </ul>	Examples Folder
⊿ 🥁 Example ∰ untitled.brd ഈ untitled.sch	Empty Project

Fig. 102 New library and associated files

Click on the grayed out circle to enable the library (Fig. 103). The circle should turn green.

ዾ Control Panel - C:\Users\kkwashnak\Des	ktop\Kwashnak\Projects\Examp	ole - EAGLE 5.3.0 💼 🔳 📧
File View Options Window Help		
Name	Description	
▲ Libraries		
⊳ 🚞 lbr	Libraries	
A O ExampleManu lbr		
\$\$ MICROTEST-DIP_8		
DIP_8	-	
Design Rules	Design Rules	
User Language Programs     Security	User Language Programs	
Scripts	CAM Processor Jobs	
Projects	cran rocessory obs	
eagle		
examples	Examples Folder	
Projects	County Designat	
Example uptitled brd	<ul> <li>Empty Project</li> </ul>	
antiticalista R untitled.sch		
		.4

Fig. 103 Activate library

# 4. Two-Layer PCB Design: General Design Cycle

This section will use lessons learned from the previous sections to create the resources for the microcontroller board. Figure 104 depicts the general methodology for designing and manufacturing a two-layer PCB in EAGLE.



Fig. 104 PCB design cycle

- - -🕒 🕞 🛡 📙 C:\Users\kkwashnak\Desktop\Kwashnak 👻 🐓 Search Kw... 🔎 Organize 🔻 Include in library 🔻 Share with 🔻 New folder ..... ? Name Date modified Туре 쑦 Favorites EAGLE 5.3.0 CD 8/18/2020 3:59 PM File folder 🧮 Desktop 8/20/2020 8:17 PM 🌗 Eagle Libraries File folder 门 Libraries Projects 8/19/2020 1:53 PM File folder 🔋 kkwashnak 💻 Computer 📬 Network 🐖 Control Panel 👿 Recycle Bin 📗 Kwashnak 📗 VAULT 4 3 items

In Windows File Explorer, create a similar file structure as shown in Fig. 105.

#### Fig. 105 EAGLE control panel tree

C:\Users\kkwashnak\Desktop\Kwashnak

C:\Users\kkwashnak\Desktop\Kwashnak\Eagle Libraries

C:\Users\kkwashnak\Desktop\Kwashnak\Projects

# 4.2 Step 2: Configure Directories

In the EAGLE Control Panel, open <Options>, <Directories...>, and add the links to the OS file locations for both the Library and Project folders (Figs. 106 and 107).

Control Pa File View	anel - C:\Users\kkwashnak\Deskto	op\Kwashnak\Projects\Exan	nple - EAGLE 5.3.0 Pr 💼 🔳 💌
Name	Directories	Description	
<ul> <li>Libraries</li> <li>lib</li> <li>Ei</li> <li>Design RU</li> <li>User Lang</li> <li>Scripts</li> <li>CAM Jobs</li> <li>Projects</li> <li>Projects</li> <li>Ei</li> <li>Project</li> <li>Project</li> </ul>	Backup User interface Window positions uage Programs ; agle (amples rojects	Libraries Design Rules User Language Programs Script Files CAM Processor Jobs Examples Folder	

Fig. 106 EAGLE control panel directories

Directories	Image: State Sta
Libraries	\$EAGLEDIR \br;C: \Users \kwashnak \Desktop \Kwashnak \Eagle Libraries
Design Rules	\$EAGLEDIR\dru
User Language Programs	\$EAGLEDIR \ulp
Scripts	\$EAGLEDIR\scr
CAM Jobs	\$EAGLEDIR \cam
Projects	\$HOME\eagle;\$EAGLEDIR\projects\examples;C:\Users\kkwashnak\pesktop\Kwashnak\Projects
	OK Cancel Browse

Fig. 107 Directory locations

Libraries: \$*EAGLEDIR\lbr;C:\Users\kkwashnak\Desktop\Kwashnak\Eagle* Libraries

Projects: \$HOME\eagle;\$EAGLEDIR\projects\examples;C:\Users \kkwashnak\Desktop\Kwashnak\Projects

Note, the "Example" Project folder appears from the previous section (Fig. 108). In the tree, 'Libraries' should house custom and manufacturer libraries. Also, 'Projects' should encompass design files, such as schematics and board files. The 'Design Rules' folder contains edits to the board's specifications, which will be edited via the DRC in the Board Editor. Advanced users will use the other folders for custom scripts and manufacturing processes.



Fig. 108 Example EAGLE control panel file tree

## 4.3 Step 3: Generate a New Project

Create a new project in the 'Projects', "Projects" folder, rename file as "pcb 2layer" (Fig. 109).



Fig. 109 pcb\_2layer project creation

After creating the file, the "pcb\_2layer" project should become the active file (Fig. 110). If not, click on the circle next to the project name. Green means the file is active, gray means the file is not in use.

📕 Control Panel - C:\Users\kkwashnak\Des	ktop\Kwashnak\Projects\pcb	_2layer - EAGLE 5.3.0 💼 💿 💌
File View Options Window Help		
Name	Description	Empty Project
<ul> <li>Libraries</li> <li>Design Rules</li> <li>User Language Programs</li> <li>Scripts</li> <li>CAM Jobs</li> <li>Projects</li> <li>Mean</li> </ul>	Design Rules User Language Programs Script Files CAM Processor Jobs	Use the context menu to create new schematic or board files within this project.
<ul> <li>examples</li> <li>Projects</li> <li>Example</li> </ul>	Examples Folder	
b pcb_2layer	Empty Project	
C:\Users\kkwashnak\Desktop\Kwashnak\Pr	ojects\pcb_2layer	

Fig. 110 Active project

### 4.4 Step 4: Create a New Schematic

Add a schematic to the project folder, as shown in (Fig. 111).

ዾ Control Panel - C:\Users\kkwashnak\Des	sktop\Kwashnak\Pro	jects\pcb_2la	yer - EAGLE 5.3.0		
File View Options Window Help					
Name	Description	E	mpty Project		
<ul> <li>Libraries</li> <li>Design Rules</li> <li>User Language Programs</li> <li>Scripts</li> <li>CAM Jobs</li> <li>Projects</li> <li>image eagle</li> <li>image examples</li> <li>image eagle</li> <li>image examples</li> <li>image eagle</li> <li>image example</li> </ul>	Programs Jobs r	se the context men chematic or board fi	u to create new les within this project.		
a 👹 pcb_2layer	Empty Project	Close P	Close Project		
		New	•	Schematic	
		Renam Copy	e	Board Library	
		Delete Edit De	scription	CAM Job	
C:\Users\kkwashnak\Desktop\Kwashnak\Pr	rojects\pcb_2layer	Use all Use not	ne	ULP Script Text	

Fig. 111 Creating a new schematic

After creating a new schematic, the Schematic Editor will open. Save the file in the designated project folder location. In the EAGLE Control Panel, the schematic will appear in the tree (Fig. 112).



Fig. 112 Schematic file in tree

Save Schematic to folder:

C:\Users\kkwashnak\Desktop\Kwashnak\Projects\pcb 2layer

### 4.5 Step 5: Design the Schematic

In the EAGLE Control Panel, examine the 'Libraries' tree, 'lbr' folder (Fig. 113).

File View Options Window	Help			
Name		Description		Libraries
▲ Libraries			Ξ	The component libraries supplied with
🔺 🔄 Ibr		Libraries		EAGLE have been compiled with great care
elektro		Libraries for Electric		as an additional service to you, our
I9inch.lbr	9	19-Inch Slot Euroca		customer. However, the large number of
⊳ 🧾 40xx.lbr	0	CMOS Logic Device		these components means that the
41xx.lbr	9	41xx Series Devices		occasional discrepancy is unavoidable.
⊳ 🧾 45xx.lbr	0	CMOS Logic Device		Please note, therefore, that CadSoft take
Ø 24ac-logic.lbr	0	TTL Logic Devices,		no responsibility for the complete accurac
74ttl-din.lbr	0	TTL Devices with DI		of information included in library files.
⊳ 🗾 74xx-eu.lbr	0	TTL Devices, 74xx S		Additional new libraries, that have not ye
	0	Single and Dual Gat		been officially released, can be found on
74xx-little-us.lbr	0	Single and Dual Gat		CadSoft's internet site at the download
⊳ 🗾 74xx-us.lbr	9	TTL Devices, 74xx S		section of
	0	75xxx Series Devices		http://www.cadsoftusa.com.
Image: Provide the second s	lbr ∘	Agilent Technologies		Use the ADD command in the Schematic
allegro.lbr	0	Allegro MicroSyste		Editor or Layout Editor window to search
altera-cyclone-II.lbr	9	ALTERA Cyclone II F		for a certain device or package!
Image: Second	9	ALTERA Cyclone III		Information about defining your own
altera.lbr	9	Altera Programmab		libraries can be found in the file library.t
am29-memory.lbr	0	Advanced Micro De	+	in the doc directory.

Fig. 113 Inactive default libraries



Right click on the 'Libraries' tree main folder and select <Use All> (Fig. 114).

Fig. 114 Activating all libraries

Back in the Schematic Editor, navigate to and click on the  $\langle Add \rangle$  tool from the left-hand toolbar (Fig. 115). In the 'Search' bar, type "Frame" and hit [Enter]. Double click or select  $\langle OK \rangle$  on the desired frame size, in this case, 'FRAME\_A\_L FRAME A SIZE, 8 1/2 × 11 INCH, Landscape'.

File Ed	ematic - C:\Users\kkwashn lit Draw View Tools	ak\Desktop\Kwashnak\Projects\pcb_2la Library Options Window Help	layer\pcb_2layer.sch - EAGLE 5.3.0 Professional
i 🖙 🖬	😂 🗾 🖁 1/1 🗸 📢	I 🛱 🗮 🔍 🔍 🔍 🔍 🖉 🗠	ବ ଜା 🎟 📓 💡
i 📀	II ADD		<b>X</b>
▼中市□★×⇔◎を、10■	Name Alt-LOC A4-355C A4-355C A4-355C A4-353CP A4-55 DINA3_P PINA4_P PINA5_L DINA5_P FRAME B_L FRAME C_L FRAME C_L FRAME C_L FRAME C_L	Description Frames for Sheet and Layout FRAME FRAME FRAME FRAME FRAME FRAME FRAME FRAME FRAME FRAME FRAME FRAME FRAME FRAME FRAME FRAME FRAME Size, 11 × 12 INCH, Landsca FRAME C Size, 22 × 34 INCH, Landsca FRAME E Size, 22 × 34 INCH, Landsca FRAME E Size, 34 × 44 INCH, Landsca	Frame A Size , 8 1/2 x 11 INCH, Landscape
]]]] + ≪	LETTER_P RAHMEN3Q	FRAME	
a ∎ Ω	Search 📝 Smds	Description Preview	
€Q; (¶)	riame		CK Cancel Drop

Fig. 115 Adding a frame to schematic

Place the object at the origin of the workspace (Fig. 116). Left click once to place the frame and then hit [Esc], to exit placing additional frame objects.



Fig. 116 Frame placement

Just as the frame is placed, the microcontroller previously created in this tutorial will be placed in the schematic. Using the <Add> tool and 'Search' bar, type "MICROT\*". The "\*" is a wildcard that allows indexing to multiple instances. Find "MICROTEST-DIP\_8" and place into the schematic (Fig. 117).

I ADD	×								
Name         Description           ExampleManu         MICROTEST-DIP_8	VDD     VSS     VSS       CP5     CP0/PDAT       CP4     CP1/PCLK       MTLER/VPP/CP3     MTLER/VPP/CP3       >NAME       >VALUE								
Author: Ken Kwashnak         Contact: Kenneth.M.Kwashnak.ctr@mail.mil         Date: August 20, 2020         This is a sample description file for a Library Device component.         1 Testing Parameters         Sample Text									
MICROT*	A: This is an indented line item. This is <b>BOLD</b> text.      B: This is an indented line item. This is <i>IT41 ICIZED</i> text								
	OK Cancel Drop								

Fig. 117 Adding MICROTEST-DIP\_8 to the schematic

Left click to place component, press [ESC], and click <Cancel> or press [ESC] again (Fig. 118).



Fig. 118 Frame and component in schematic editor

Add components, as stated in Table 11, to the schematic using the <Add> tool; reference the Schematic Object for the object's name. Arbitrarily place the devices on the sheet. Use the <Move> tool to reorder and organize the devices. For the general schematic design, see Section 4.6.

Line	Schematic object	Description	Manufacturer	PN	Q
1	MICROTEST-DIP 8ª	Microcontroller, DIP8	Microchip	PIC12LF150116	1
2	No device <sup>b</sup>	IC DIP socket 8POS	Adam Tech	ICS-308-T <sup>17</sup>	1
3	No device <sup>b</sup>	Battery, CR2450, 3V	Energizer	CR2450VP <sup>18</sup>	1
4	BS-2450°	Battery holder, coin PC pin	MPD	BS-2450 <sup>19</sup>	1
5	MCP1702-5002E/TO <sup>c</sup>	Regulator linear 3V, TO92-3	Microchip	MCP1700-3002E/TO <sup>20</sup>	1
6	5988170107F_0805ª	Green SMD LED	Dialight	5988170107F <sup>21</sup>	4
7	PTS647SK38SMTR2L	Tactile switch, NO	C&K	PTS 647 SK38 SMTR2 LFS <sup>22</sup>	1
8	R-US_R0805	Res, 0805, 120Ω, 5%, 1/8W	Vishay	CRCW0805120RJNEA <sup>23</sup>	4
9	C-USC0805	Cap, 0805, 0.1uF, 25V, X7R	KEMET	C0805C104M3RACTU <sup>24</sup>	1
10	22-23-2051	0.100" header, 5 pin, Prog	Molex	22-23-2051 <sup>25</sup>	1
11	No device <sup>b</sup>	0.100", conn housing, 5POS	Molex	$0022012057^{26}$	1
12	22-23-2021	0.100" header, 2 pin, Batt	Molex	22-23-202127	1
13	No device <sup>b</sup>	0.100", conn housing, 2POS	Molex	0022012027 <sup>28</sup>	1
14	No device <sup>b</sup>	Conn 22-30AWG crimp tin	Molex	0008650804 <sup>29</sup>	7
15	SYM-MOUNT-HOLE <sup>d</sup>	0.125" plated mount hole			4
16	supply2, GND	Ground symbol, GND			5

#### Table 11 Schematic custom two-layer board objects

<sup>a</sup> Custom device.

<sup>b</sup> No device created, only need to reference component for assembly.

<sup>c</sup> Extracted symbol from Ultra Librarian. Edited device footprint.

<sup>d</sup> Custom symbol.

Either a battery, CR2450VP, or external power supply provides power to a two pin header, 22-23-2021 (The board must only be powered by a battery or an external power supply; never simultaneously hook up both inputs). The header connects to a linear regulator, MCP1700-3002E/TO. A PIC12LF1501 microcontroller receives power from a linear regulator. Pin 1 of the microcontroller is tied to the linear regulator output and Pin 2 of the 5-pin programming header. Pins 2, 3, and 5 of the microcontroller are tied to a resistor-led network (LED and resistor). Pin 4 of the microcontroller is tied to a reset switch and pin 1 of the 5-pin programming header. Pins 6 and 7 of the microcontroller are tied to the 5-pin programming header for the clock and data signals, PCLK and PDAT, respectively. Pin 8 of the microcontroller is tied to ground. The board has four plated mounting holes to assist with assembly and for placement into an enclosure.

The components listed in Table 11 are chosen to build a demo two-layer microcontroller PCB. Unfortunately, the circuit is not optimized or tuned but will function as desired. Always read the manufacturer's datasheets and compare specs of components to ensure compatibility and functionality, as well as applying safe and fundamental electrical engineering knowledge when designing a circuit.

It is always a good practice to make a detailed schematic, noting all voltages, connections, part numbers, signal profiles, and so on. But, for larger and more intricate projects, this method becomes too cumbersome and difficult. Documentation of expected circuit behavior is essential when testing and evaluating boards.

Using the <Text> tool, enter external connection information above the headers (Fig. 119). For example, the two-pin header 22-23-2021 connects the linear regulator to an external battery/power supply, while the five-pin header 22-23-2051 connects the microcontroller to a programming device. For each header, comment above the respective device: "Battery Connection" and "Programming Connection" (Fig. 120). Also, change the layer of the text to '97 – Info'.



Fig. 119 <Text> tool



Fig. 120 Custom schematic text

# 4.5.1 Editing Imported Symbols

If a device is imported into the main library and added to the design, the symbol, footprint, or the device may need to be adjusted to fit the design scheme. For example, examine Figs. 121 and 122.



Fig. 121 Raw imported symbol



Fig. 122 Edited symbol

Figure 121 illustrates the imported original symbol design and Fig. 122 is the updated symbol design that fits the schematic's current scheme. Each device may have numerous symbol variations in order to accommodate documentation requirements. If the symbol is added to the schematic and edited later, the symbol must be manually updated via <Library>, <Update...> or <Update All...>.

Symbols may represent the actual component layout, but others may be optimized for functionality. For example, on an integrated circuit (IC), there may be many ground (GND) lines. The symbol may have those GND lines grouped together to keep the schematic's routing clear and concise.

If a symbol is placed on the schematic, but the ">NAME" and/or ">VALUE" obstructs other objects, use the <Smash> tool and click on the obstructing object's center (Fig. 123). Then, use the <Move> tool to arrange selected text (Fig. 124). Always check the device, package, and symbol definitions with the datasheet when importing a library!



Fig. 123 <Smash> tool on object



Fig. 124 Moving smashed text

# 4.5.2 Creating Grounded Mounting Hole Symbols

One method to create a grounded mounting hole is to simply use a via and adjust the annulus (the ring of copper around the drill hole) and drill size on the board file. A different method would involve creating a highly detailed device. For this small project, the former is used due to its relative simplicity and ease of use.

It is essential to place all elements of the design on the schematic, even if there is no fully defined device (a device that has a paired package and symbol) placed on the board. Therefore, in addition to creating a custom via on the board, a symbol is placed on the schematic.

Create a new library, "holes\_mount" and place in the "Eagle Libraries" folder.

C:\Users\kkwashnak\Desktop\Kwashnak\Eagle Libraries\holes\_mount.lbr

Create a new symbol, "MOUNT-HOLE", in the library (Figs. 125-129).



Fig. 125 mount\_hole library











Fig. 128 MOUNT-HOLE symbol infill

🥑 1 File	Library - C Edit D	C:\Users\kkwasł Draw View	nnak\Deski	<mark>top∖Kwas</mark> Options	hnak\Eag Windo	gle Libraries\ w Help	holes_r	nount.lk	or (MOUN	IT-HOLE.sy	m) - EAGLE	5.3.0 Pro	ofessional		
-	: 🖪 🎒	88 🖾 🕀		Q, 🔍 🤉	a, 🔍 (	<b>ù</b>  ∽ ∾	<b>911</b>	8   ?							
i	<ul> <li>0.1 inc</li> </ul>	ch (0.10.0)													
•	z 💵 🖉 Prope	erties				×									
⊕ ⊨⊐	Text														
	Position	0.1		0.03							Λ	R			-
Ł	Angle	0				•						Ν	Λ		
×		Mirror								NZ		IV			_
	Size	0.07				•		-							
1	Ratio	8 %				•									
6	Font	Proportional				•			_				_		1
	Layer	95 Names				•				//	\ I			11	_
-0	Value	>NAME						>			$\Delta$ I		ų –		_
		ОК		Cancel		Apply									-
	•								III						Þ
♦ Le	eft-click to	select object to	get info f	or											

Fig. 129 MOUNT-HOLE symbol text

Once the symbol is completed, save the file.

Navigate to the <Device> and create a new device called "SYM-MOUNT-HOLE" (Figs. 130 and 131).



Fig. 130 SYM-MOUNT-HOLE device creation



Fig. 131 SYM-MOUNT-HOLE device complete

After completing the device, save the file and return to the schematic.

In the schematic, use the <Add> tool to insert the 'SYM-MOUNT-HOLE' object (Fig. 132).



Fig. 132 SYM-MOUNT-HOLE object

Use the <Text> tool to incorporate necessary information on the schematic, like "Plated Through, Mounting Holes". Next, navigate to the <Board> tool and use the <Via> tool to construct mounting holes (Fig. 133). Configure the object's information with the <Info> tool.



Fig. 133 Via used as plated through hole

Remember, the schematic only shows the symbol of the mounting holes, whereas on the board there is a via/mounting hole element. There is a disconnect between the schematic and the board with this method, but the user must understand the project's specifications upon designing and inspecting the project.

### 4.5.3 Changing Component Designation

Once a component is placed on the schematic, the ">NAME" tag displays the symbols name. In Fig. 134, the microcontroller is referred to as 'U\$1'. If multiple microcontrollers are added, the number increments +1, thus the schematic would state 'U\$2', 'U\$3', and so on. To change the designation 'U' to another letter or prefix, edit the device in the library and configure the <Prefix> option (Fig. 135).



Fig. 134 >NAME \$U1 component designation



Fig. 135 <Prefix> Option

Once a prefix is added and if a device is already on the schematic, delete the object and reinsert. The object designation will change to the next prefix.

Use the <Line> tool to connect component pins together (Fig. 136). Use the <Info> tool to change the line's name (also called a 'Net') (Fig. 137). A 'Bus' is simply a connection of 'Nets'.



Fig. 136 Defining line properties

I Schematic - C:\Users\kii         File       Edit       Draw       View         Image: Display the state of the st	wwshnak\Desktop\Kwashnak\Proje Tools Library Options Wind	tts\pcb_2layer\pcb_2layer.sch - EA ow Help Q Q I ← ← 1 1 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2	GLE 5.3.0 Professional	
	R1 R2	LED_RED	V <u>DD</u> CP5 CP4	VDD CP5 +
G 登録で ア ジ び の つ 日 同 間 (1) (1) (1) (1) (1) (1) (1) (1) (1) (1)	S1 TME1 	MCLR/	VPP/C <u>P3</u>	U\$1
	¢	m		

Fig. 137 <Label> Tool

## 4.6 Step 6: Map Nets and Connections

🔛 1 Sch	nematic - Ci	\Users\	kkwashni	ak\Desktop	\Kwashnak\l	Projects\pcb	_2layer\p	cb_2layer.s	sch - EAGLE 5	3.0 Professional	
File E	dit Draw	View	Tools	Library	Options	Window H	lelp				
i 🖙 🖥	8 📻	ő 🏼	1 🔹 📢		i 🔊 🕷 i	3 9 9	n Ci	<b></b>	?		
	91 Nets	•	<u> </u>	~~	$-\gamma$ .	S Miter: 0		- r r	Width: 0.0	06 👻 Style: Continuo	us 🔻
• •	Sheets	₽×	0.1 inch	(5.13.0)							-
🔍 🗠	t; į204∈	^			L			VDD	VDD	VSS	VSS *
+ 88 =	1/1					W	V	CP5	CP5	CP0/PDAT	CP0/PI
							∼	CP4	CP4	+ CP1/PCLK	CP1/P(
* >						MC	TRAP	P/CP3	MCERA	/PP/CP3 CP2	CP2
A 4€ 0.0											
83											'
<b>P2</b> 104 <b>P3</b> 104					3	8 4			U\$1		
۳ ۳					L				MICROT	EST-DIP_8	
<b>A</b>											=
					SV	/1					
					PT	S647Sł	<38S	MTR2	LFS		
הר											
+ es											
8									-	<b>F</b>	
et 🔒										GND	
		-	•								•
+ Left-	click to draw	wire (	double-cl	ick ends w	ire)						

Use the <Wire> tool to connect component pins (Fig. 138).

Fig. 138 Connecting component with <Wire> tool

Use the <Name> tool on wire elements to formulate signal names (Fig. 139). If a wire is renamed to an existing wire name, a prompt for merging two nets arises (Fig. 140). Wires may be renamed from their 'N\$1' or 'N\$2' to assist organizing signals and nets.

The fait Daw View Teels Library Options Window Help The fait of the state of the s	1 Schematic - C:\Users\kkwashnak\Desktop\Kwashnak\Projects\pcb_2layer\pcb_2layer.sch - EAGLE 5.3.0 Pro	fessional 👝 😐 🖾							
Pres     B     V	File Edit Draw View Tools Library Options Window Help								
Image: Section of x         D Indo (5.8.3.0)           Image: Section of x         D Indo	🏣 🔜 🍜 岩 🏨 🛶 🗰 💐 🔍 🍳 🔍 🔍 💷 🗠 😁 🌉 🤰 ?								
Plat Protection									
C Cancel Apply	Properties           Wire           From 5.55         3           From 5.55         4 <t< th=""><th>Plat</th></t<>	Plat							
the select object to get info for	SW1     SW1     PTS647S     Cancel Apply     SW1     Cancel Apply								

Fig. 139 Renaming nets and defining signals

I Schematic - C:\Users\kkwash	nak\Desktop\Kwashnak\Projects\pcb_2lay	er\pcb_2layer.sch	- EAGLE 5.3	.0 Professional 🛛 🖂	
🚘 🖬 🚳 🚅 🔒 1/1 🕞	* # # # <b>4 4 9 9 9</b> 0	a   💷 📱   1	3		
91Nets 👻 🗖	/// / 5 Miter: 0	· rr	Width: 0.00	6 👻 Style: Continuo	us 👻
i ⊙ Sheets ♂ × 0.1 inc	h (5.6 3.0)				• •
Ne Le		VDD	VDD	VSS	VSS
		CP5	CP5	CP0/PDAT	CP0/PI
中中 二 本	R2	CP4	CD4	+ CD1/DCLK	CP1/P(
* >	- VVV		014		CP1
× &		1117010	MCER/VI	PP/CP3 CP2	01.2
11) 0-0 P>		ך נ			]
다. 응 응	3	J	\$1		
۹r	Ca warning			8	
17 ¥8	A Merge	net segment 'NS8	into supply	rnet 'GND'?	=
	S)	Yes	No		
	P				
in the second se					
⊕ ( <b>)</b>				t.	
			(	JND	
					-
A Latte click to draw wire (double, click and wire)					
Left-click to draw wire (double-click ends wire)					

Fig. 140 Merging nets

In the end, the schematic may look like Fig. 141.



Fig. 141 Completed schematic

#### 4.7 Step 7: Run Electronic Review Check

The Electronic Rule Check, <Erc> tool, checks over the schematic for connection errors, such as improperly routed pins, missing connections, and paired pin state configurations (Fig. 142). Using this tool will jump to the Schematic Editor.



Fig. 142 <ERC> tool

The ERC reviews connections and related properties (Fig. 143). For example, if a pin is labeled as 'pwr', the ERC checks if the incoming connection is also labeled with 'pwr', so the signal type is in agreement.

Туре			Sheet
Board	and schematic are consister	ıt	
Erron	s (0)		
4 Warn	ings (41)		
9	Missing junction in net 1922		
2	Missing junction in net NS2	,	
2	Missing junction in net IVS2	,	
	Missing junction in net 1922	, ,	
2	Missing junction in net NS2	,	
	Missing junction in net NS2	,	
2	Missing junction in net NS2	,	
	Missing junction in net NS5	, ,	
2	Missing junction in net NS5		
2	No pipe on pet N\$16		
2	No pins on net N\$17		-
2	No nins on net N\$18		
2	No pins on net N\$19		-
2	No pins on net N\$20		
7	No pins on net N\$21		-
- d	No pins on net N\$22		
7	No pins on net N\$23		-
d	No pins on net N\$24		
- ă	No pins on net N\$25		
7	No pins on net N\$26		
- ă	No pins on net N\$27		
ð	No pins on net N\$28		
d	No pins on net N\$29		
ð	No pins on net N\$3		
đ	No pins on net N\$30		
ð	No pins on net N\$5		;
ā	No pins on net NS6		
Č.	POWER pin D1 ANNODE co	nnected to N\$11	
Č.	POWER pin D1 CATHODE co	onnected to GND	
0	POWER pin D2 ANNODE co	nnected to N\$10	
0	POWER pin D2 CATHODE co	onnected to GND	1
	POWER pin D3 ANNODE co	nnected to N\$7	
	POWER pin D3 CATHODE co	onnected to GND	1
(	POWER pin D4 ANNODE co	nnected to N\$12	
	POWER pin D4 CATHODE co	onnected to GND	1
	POWER pin U\$1 VDD conne	cted to 3V	1
	POWER pin U\$1 VSS connec	ted to GND	
(	POWER pin X3 + connected	to VBAT	
9	POWER pin X3 - connected	to GND	1
. (	Part FRAME1 has no value		
Appro	oved (0)		J
Center	ed	Clear all	
	Processed	Approve	

Fig. 143 ERC dialog

Although this design does not have any glaring errors, there are multiple warnings that could potentially be critical errors. Clicking on the warning moves the screen to the issue location (Fig. 144).



Fig. 144 Warning list

The top set of 'missing junctions' and 'no pins' makes sense; the <Wire> tool was used to create a diagrams. These diagrams have no connection to the PCB, so the warnings may safely be ignored or <Approved>. These floating wire segments will not affect the board file.

Next, after examining the 'POWER pin D1 ANNODE connected to N\$11', the LED object's pins are configured as both 'pwr' pins. Thus, the ERC is checking to make sure that the object is indeed hooked up correctly. This is for the designer to examine and approve, since the designer knows about the input and out connections of the device. In this case, the LED is configured correctly; the warning is <Approved>.

Following through, the remaining errors can be reviewed and <Approved> (Fig. 145).



Fig. 145 Approved ERC

# 4.8 Step 8: Create a New Board Layout

Click on the <Board> icon to switch views (Fig. 146). Navigate to <View>, <Grid>, and edit desired settings.



Fig. 146 <Board> icon

In addition, for this tutorial, the default black background for the Board workspace is changed to white. To mirror this step, switch to the EAGLE Control Panel, go to <Options>, and finally <User interface...> (Fig. 147). Under 'Layout', change 'Background' from <Black> to <White>.

📕 User interface	<b>—</b>
Controls V Pulldown menu Action toolbar V Parameter toolbar	Layout Background: O Blad O White Colored Cursor: O Small Large
Command buttons Command texts Sheet thumbnails	Schematic Background: () Black () White () Colored Cursor: () Small () Large
Always vector font          Image: Always vector font         Image: Always ve	<ul> <li>✓ Bubble help</li> <li>✓ User guidance</li> </ul>
	OK Cancel

Fig. 147 Changing board workspace background color
### 4.9 Step 9: Define PCB Settings

Use the <Drc> tool in the Board Editor to check and modify the standards for the PCB (Figs. 148–158). If errors are found, the board may need trace rework, components edits, and/or and device redesign. For this step, review PCB manufacturers and their manufacturing specifications.



Fig. 148 Design rule check on completed board



Fig. 149 DRC dialog

le Layers	Clearance	Distance	Sizes	Restring	Shapes	Supply	Masks	Misc	
			Nr 1 0.035 16 0.035	imm	Copper	·		1.5mm	Isolation
igers are combined in three indicates are determined with three indicates are determined with the completer (2:1) res. The cores yer 1 to 2.	hed through eit hugh vias are d fined by writing + ( (2 * 3) + (4' are combined t	Se ther <i>core</i> or <i>p</i> lefined by wri g [t::b *16))] is a hrough a pre	tup (1*) repreg ma ting ( J , which c multilayer preg and l	i6) iterial. <b>a*b</b> c ). lefines a blinc setup with tw puried vias ar	ombines laye d via from to vo cores, cor re produced i	ers <i>a</i> and <i>b</i> p to layer <i>t</i> mbining laye through the	with a <i>core</i> and from b ers 2/3 and e resulting s	, while <b>a</b> - ottom to l 4/16, res tack. Fina	+ <b>b</b> does the same with <i>prepreg</i> . layer <i>b</i> . spectively, with buried vias going through both ally layer 1 is added, with blind vias going from

Fig. 150 Layers

DRC (default *)												
File Layers	Clearance	Distance	Sizes	Restring	Shapes	Supply	Masks	Misc				
							Differe	ent Sign	als			
				Win	e							
		Wire	8mil					Pad				
		Pad	8mil			8mil					Via	
		Via	8mil			8mil				8mil		
							<i>c</i>	- C I				
	1			Sm	4		Sam	e Signai Pad	5		Via	
	•   • <b>•</b>	Smd	8mil	Din	-	8mil		- uu		8mil	10	
1		01110										
Minimum Clear	ance between	n objects in sign	al layers.									
The Same Signa	Is check betw	een <i>Smd</i> and <i>Vi</i>	a does not	t apply to A	Nicro Vias.							
Setting the values	for the Same	e Signals ched	ks to 0 disa	ables the re	espective ch	eck.						
							Cł	neck	Select		Cancel	Apply

Fig. 151 Clearance

BRC (default *)
File         Layers         Clearance         Distance         Sizes         Restring         Shapes         Supply         Masks         Misc
Copper/Dimension       40mil         Drill/Hole       8mil    Minum Distance between objects in signal layers (pads, smds and any copper connected to them) and the board dimensions, and between drill holes. Setting the value for the Copper/Dimension check to 0 disables that check.
Check Select Cancel Apply

Fig. 152 Distance

🛃 DRC (default *)		
File Layers Clearance Distance	Sizes Restr	ing Shapes Supply Masks Misc
Minimum Sizes of objects in signal layers and	mum Width mum Drill Micro Via Blind Via Ratio d of drill holes.	10mil       18mil       9.99mm       0.5
Min. Micro Via applies to <i>blind</i> vias that are en the default value of 9.99mm) means there are in Min. Blind Via. Batia defines the application drill	xactly one layer no micro vias.	deep. Typical values are in the range 50100 micron. A value larger than <b>Minimum Drill</b> (e.g.
"aspect ratio" in the form 1:0.5, where 0.5 v	would be the value	ue that has to be entered here.
		Check Select Cancel Apply

Fig. 153 Sizes

le Layers	Clearance	Distance Sizes	Restri	ng Shapes	Supply	Masks	Misc		
				Min			%	Max	Diamete
	-	Pads	Тор	10mil		25		20mil	
			Inner	10mil		25		20mil	
			Bottom	10mil		25		20mil	
_		Vias	Outer	8mil		25		20mil	
			Inner	8mil		25		20mil	
		Micro Via	s Outer	4mil		25		20mil	
			Inner	4mil		25		20mil	
Lestrings for pair arger restring, the f the Diameter ficro Vias are a may be overwritt	ads and vias are nat value will be option is checke <i>blind</i> vias that ar en by a larger <b>D</b>	defined in percent o used in the outer lay ed the actual pad or v e exactly one layer d will value in the <i>Net</i> o	f the drill o ers. via diamet leep and h <i>classes</i> ).	diameter (limited er will be taken ir nave a drill diame	by <b>Min</b> and to account ter that is s	I <b>Max</b> ). If t in the inner maller than	the diameter of r layers, too. the <b>Minimur</b>	f an actual pad or via would n Drill value defined under	l result in a <i>Sizes</i> (which

Fig. 154 Restring

DRC (default *)				<u>_</u>
File Layers Clearance	Distance Sizes Restri	ng Shapes Supply N	1asks Misc	
	Smds	Min	%	Max
	Roundness	Omil	0	Omil
	Pads		Shape	
	Тор	As in library		•
	Bottom	As in library		•
	First	Not special		<b></b> ]
	Elongation %	100	100	
Shapes of pads and smds.				
			Charles Calent	

Fig. 155 Shapes

🛃 DRC (default *)							
File Layers Clearance Distance	e Size	s Restring	Shapes	Supply	Masks	Misc	
					%	Max	
	Gap	20mil		5	)		100mil
			-1-1-				
	Thermal	10mil	olate			Restring	1
	Annulus	20mil					
						_	
	Gene	rate thermals fo	r vias				
Supply symbols are generated for pads a	and vias in	supply layers.	Laturation and				
The Gap is defined in percent of the drill	s, the resul	nited by <b>Min</b> ar	ool Will be a t	ully filled d	rcie instead	or a ring	ıg.
The Thermal Isolate parameter will also	be used f	for signal polygo	ns.				
NOTE: The actual shape of supply symbol	s may be d	ifferent when a	enerating ou	tput for ph	otoplotters	that use	e specific thermal/annulus apertures!
	,						
					_		
					Cł	neck	Select Cancel Apply

Fig. 156 Supply

File	Lawere	Classanca	Distance	Cizon	Destring	Chapter	Supply	Macke	Mine	1			
File	Layers	Clearance	Distance	Sizes	Resulting	Shapes	Suppry	Pidaka	MISC				
					Μ	lin			%			Max	c .
			Stop	4mil			1	00			4mil		
			Crea	am Omil			0				Omil		
			Limit	Omil									
1			. IMIT	1.000									
			Linic	Unit									
			-	Unit									
			-	Unin									
ļ				. Unin									
Mask	values are	defined in perc	ent of the smal	ller dimer	nsion of smds	s, pads and	vias (limit	ed by <b>Min</b> a	nd Max).				
Mask Stop	values are	defined in perc	ent of the smal	ller dimer	nsion of smds as that have	s, pads and a drill diame	vias (limit	ed by <b>Min</b> a	nd Max). it.				
Mask Stop	values are	defined in percented for s	ent of the smal	ller dimer	nsion of smd as that have	s, pads and a drill diame	vias (limit ter that e	ed by <b>Min</b> a	nd Max). it.				
Mask Stop Creat	values are masks are g	defined in perce generated for s e generated for	ent of the smal mds, pads and r smds only.	ller dimer	nsion of smd: as that have	s, pads and a drill diame	vias (limit ter that e	ed by <b>Min</b> a exceeds Lim	nd Max). it.				
Mask Stop Crea	values are masks are g	defined in pero generated for s e generated for	cent of the smal mds, pads and r smds only.	ller dimer	nsion of smd: as that have	s, pads and a drill diame	vias (limit ter that e	ed by <b>Min</b> a exceeds Lim	nd Max). it.				
Mask Stop Crea	values are masks are g m masks are	defined in pero generated for s e generated for	cent of the smal mds, pads and r smds only.	ller dimer	nsion of smd as that have	s, pads and a drill diame	vias (limit eter that e	ed by <b>Min</b> a exceeds Lim	nd Max). it.				
Mask Stop Crea	values are masks are g	defined in pero generated for s e generated for	ent of the smal mds, pads and r smds only.	ller dimer	nsion of smd: as that have	s, pads and a drill diame	vias (limit ter that e	ed by <b>Min a</b> exceeds Lim	nd Max). it.				
Mask Stop Crea	values are masks are g	defined in perce generated for s	ent of the smal mds, pads and r smds only.	ller dimer	nsion of smda	s, pads and a drill diame	vias (limit tter that e	ed by <b>Min</b> a exceeds Lim	nd Max). it.				
Mask Stop Crea	values are g masks are g	defined in perce generated for s	eent of the small	ller dimer	nsion of smd	s, pads and a drill diame	vias (limit ter that e	ed by Min a	nd Max). it.				
Mask Stop Crea	values are masks are ç m masks are	defined in perco generated for s e generated for	eent of the smal mds, pads and r smds only.	ller dimer	nsion of smd	s, pads and a drill diame	vias (limit ter that e	ed by <b>Min a</b> exceeds <b>Lim</b>	nd Max). it.				

Fig. 157 Masks



Fig. 158 Misc

## 4.10 Step 10: Organize Board Components

Moving back to the Board Editor, resize the board using <Info> or <Move> tool. The exact board size may not be known at this moment, but set the desired overall dimension. To assist with sizing via the <Move> tool, examine the cursor coordinates, circled in red in the Fig. 159. When creating a board's physical outline, use the layer '30 Dimension'.



Fig. 159 Organizing components

On the <Info> tool, notice the 'Layer' the object is residing. Using the <Move> tool, arrange the components within the board outline as in Fig. 160. The yellow airwires are the connections to each component, as in the schematic. If an object is missing these airwires, double-check your connections in the schematic. Upon clicking on an object, if a component overlaps another, right-click the mouse to cycle through the selection.



Fig. 160 Board layout

To move a device to the bottom of the board, use the <Info> tool on the object and select the 'Mirror' option (Fig. 161).



Fig. 161 Placing component on bottom of board

Use the  $\langle Ratsnest \rangle$  tool to reallocate airwires. The yellow lines (layer '19 – Unrouted') between objects are created via connections between elements in the schematic (Fig. 162).



Fig. 162 Recalculating airwire locations

# 4.11 Step 11: Create Copper Regions

Now, to create copper pours for the top and bottom layers, encapsulate the board's top copper layer via the <Polygon> tool (Fig. 163).



Fig. 163 <Polygon> tool

Draw a square inside the board, spaced away from the board's edge about 50 mils, on the '1 Top' layer (Fig. 164). Repeat for the '16 Bottom' layer (Fig. 165). After each polygon is drawn, the lines from the <Polygon> tool will appear dotted.



Fig. 164 <Polygon> top copper layer



Fig. 165 <Polygon> bottom copper layer\*

Double-check the information of the polygon to ensure the polygons are on their respective layers.

Using the <Info> tool and selecting one of the polygons under the 'Polygon' tab, 'Polygon Pour', check the 'Solid' option (Fig. 166). This will generate a copper layer on the specified layer.

<sup>\* &#</sup>x27;1 Top' layer was hidden using the <Display> tool.

Ele Edit Draw View Iools Library Options Window Help	Properties	5.3.0 Professional	
□ ノ / / □ / / S Miter: 0	Wire		>
0.05 inch (0.40 1.45)	From	0.05	1.45
■ E	То	1.45	1.45
+ 88	Length	1.4	
La Securation and La Constant	Width	0.016	-
	Сар	Round	
* >	Layer	1 Top	•
	Curve	0	
	Polygon	·	
	Polygon Pour	Solid	
är 📔 🔤 + 🔭	Spacing	0.05	
	Isolate	0	
<u>ъъ</u>	Rank	1	
/ T	Orphans		
	Thermals		
	Signal		
	Name	N\$13	
	Net Class	0 default	
	Airwires hidden		
⊕ €			-
€t €t			

Fig. 166 Polygon information

Rename the bottom layer (blue) polygon to GND, using the 'Signal' area in the 'Properties' window (Fig. 167).

🛃 Properties		
Wire		
From	0.05	1.45
То	1.45	1.45
Length	1.4	
Width	0.016	-
Сар	Round	· · · ·
Layer	16 Bottom	•
Curve	0	
Polygon		
Polygon Pour	Solid	•
Spacing	0.05	•
Isolate	0	•
Rank	1	•
Orphans		
V Thermals		
Signal		
Name	GND	
Net Class	0 default	•
Airwires hidden		
	ОК	Cancel Apply

Fig. 167 Polygon signal name

If an error is received, in the board workspace, use the <Name> tool to edit the object's Signal name (Figs. 168 and 169).

Wire		
From	0.05	1.45
То	1.45	1.45
Length	1.4	
Width	0.016	•
Сар	Round	
Layer	16 Bottom	
Curve	0	
Polygon		
Polygon Pour	Solid	<b>•</b>
Spacing	0.05	r 🗳
Isolate	0	Circul across (CND) already mintal
Rank	1	Use the NAME command to combine signal
Orphans		
Thermals		ОК
Signal		
Name	GND	
Net Class	0 default	•

Fig. 168 Renaming signal error



Fig. 169 <Name> Tool on bottom layer

Afterward, use the <Ratsnest> tool once again to formulate copper layers and airwires (Fig. 170).



Fig. 170 Formulating copper layers

If the in-filled polygons obstruct the view of airwires and routing work, use the <Ripup> tool on the drawn polygons (Figs. 171 and 172). The infill should revert back to the dotted lines. After clicking on the <Ratsnest> tool, click on the <Go> icon to remove all signals on the board. This will also remove any vias, even the custom vias for the mounting holes.



Fig. 171 <Ripup> tool, delete all signals



Fig. 172 Cleared workspace

## 4.12 Step 12: Route Traces

Use the <Route> tool to place trace connections between components (Fig. 173). Follow the airwires to connect components or create custom net designs.



Fig. 173 Creating traces with the <Route> tool

Using the <Info> tool on a wire, examine the 'Wire' and 'Signal' properties of the trace. Due to the simple nature of this PCB, routing traces at a default width will suffice, but for more complicated circuits, trace properties become incredibly important. One reference for trace calculation is the "PCB Trace Width Calculator" from Advanced Circuits.<sup>36</sup> Continue to route all traces on the board. Right-clicking while placing a trace adjusts the type of line used, and the options for the line may be found in the property bar circled in Fig. 174.



Fig. 174 Trace properties

If the trace is improperly placed, use the <Ripup> tool to delete the trace and use the <Ratsnest> tool to auto-replace the airwire. Switching from the '1 Top' to '1 Bottom' layer mid-route will create a via, which can travel from the top of the board to the bottom (Figs. 175–177).



Fig. 175 Trace top layer



Fig. 176 Trace bottom layer



Fig. 177 Via

The <Auto> tool is a trace auto-router (Fig. 178), which may be useful in guiding board design or for quickly producing trace layouts. The '41 tRestrict', '42bRestrict', and '43 vRestrict' layers control the auto-router layout behavior. For example, if a polygon (square) was placed over a region, the auto-router will not set traces within the specified area.

📓 2 Board - C/Users\\kwashnak\Desktop\Kwashnak\Projects\pcb_2layer\pcb_2layer.brd - EAGLE 5.3.0 Professional	83
The bat Daw yew joos parary Options Window Hep	
	_
1 · · · · · · · · · · · · · · · · · · ·	-
NE	
🕂 👯 📓 Autorouter Setup	
Ela 🏠 General Busses Route Optimize1 Optimize2 Optimize3 Optimize4	
Freferred Directions Routing Grid 50 mil	
🗙 🗞 1 Top / 🖛 9 Route9 N/A 💌 Via Shape Round 🗸	
10 00 2 Route2 N/A + 10 Route10 N/A +	
3 Route3 N/A w 11 Route11 N/A w	
1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	
V / S RouteS N/A + 13 Route13 N/A +	
6 Route6 N/A + 14 Route14 N/A +	
/ T 7 Route7 N/A + 15 Route15 N/A +	
0 7 8 Route8 N/A v 16 Bottom / v	
Load Save as	
• \	
♦ 🖑 OK Select Cancel	
XIII	
€ €	
• · · · · · · · · · · · · · · · · · · ·	E.
Ratsnest:	

Fig. 178 <Auto> Tool settings

Following the autorouter conventions defined in Table 12, the output mirrors Fig. 179. The infill of the PCB now has traces routed between components.

Symbol	Preferred Direction
-	Left to right
	Up and down
/	45 deg, bottom left to top right
\	45 deg, top left to bottom right
*	No preference
Auto	EAGLE will access and optimize

Table 12 Auto-router directions



Fig. 179 Auto-routed board

### 4.13 Step 13: Edit Silkscreen

Each device's package may have a soldermask (silkscreen) that references text from the '21 tPlace', '22 bPlace', '25 tNames', and '26 bNames' layers. The soldermask dictates the component's outline, description, and/or value on the surface of the PCB. Using the <Display> tool, turn off all layers except layer '21 tPlace'. Use the <Text> tool to create "Top Silkscreen" and place it next to the board (Fig. 180). Ensure the text is on layer '21 tPlace'. This note is used as a reference and will not be printed on the board.



Fig. 180 Top silkscreen

It is evident that some of the imported component's silkscreen will need a few edits because the text is too long and it overlaps other objects. Find the device in the Library and edit the text. For example, the MCP1702-5002E/TO's ">NAME" and ">VALUE" text layers are changed from layer '25 tPlace', to layer '51 tDocu'. Continue to edit all necessary component text. Later, custom text will be written adjacent to their respective component.

After editing all the devices' packages, update all libraries with <Library> tab, <Update all...> (Fig. 181).



Fig. 181 Updating libraries

Since there are only a handful of parts, each component will be manually labeled to assist with PCB assembly. Use the <Text> tool to label components, the designer's name, date, board version, and any other necessary details (Fig. 182). Text is assigned to either layer '21 tPlace' or '22 bPlace'. It may be helpful to turn on/off other layers to help with character placement, such as '1 Top', '16 Bottom', '17 Pads', '18 Vias', '20 Dimension', '21 tPlace', '22 bPlace', '29 tStop', '30 bStop', '41 tRestrict', '42 bRestrict', '44 Drills', and '45 Holes'.



Fig. 182 Completed board

## 4.14 Step 14: DRC

The <DRC> tool checks the board for various issues, such as component overlap, layer interference, open traces, and so on (Fig. 183).



Fig. 183 DRC check

After examining the errors (Fig. 184), it looks like a majority of the silkscreening interferes with pads. There are multiple solutions:

- 1) Rearrange components and/or traces
- 2) Increase the size of the board and space out components
- 3) Adjust silkscreen on device packages or create package variants
- 4) Add restricted zones to the board and re-run auto-route



Fig. 184 DRC errors

Creating a variant to a device requires using the <Cut> and <Paste> tools. First, open the original device from the Library Editor, then use the <Group> tool to highlight the entire object, and finally use the <Cut> tool to add the object to EAGLE's clipboard buffer (Fig. 185).



Fig. 185 <Cut> tool

Create a new package (Fig. 186).

🖉 Edit 📃 💌
Package
CR2450
CR2450-2
New:
Dev Pac Sym
OK Cancel

Fig. 186 Create a new package

Paste object using the <Paste> tool (Fig. 187).



Fig. 187 <Paste> tool

Edit the component. To overcome the error, the silkscreen circle's outline was reduced to a smaller diameter. Notice the original diameter interfered with the power input header. If the circle diameter is too small, it may now interfere with the through-holes from the microcontroller.

After editing the duplicate symbol, open the original device. Click <New> associated with the updated symbol (Fig. 188).



Fig. 188 Device management

Enter a variant name and click <OK> (Fig. 189).

Create new package variant for BS-2450	<b>X</b>
Packages CR2450 CR2450-2	NAME >NAME
Variant name	
REDUCED_SILK	
	OK Cancel
	h.

Fig. 189 Variant name

Name	Pin
2	G\$1.+
	< <u> </u>
Connect	Disconnect
	Connect

Go to <Connect> and configure the pins (Fig. 190).

Fig. 190 Configure pins

New symbol is ready for use. Return to the Schematic Editor and use the <Delete> tool to remove the original BS-2450 object. <Add> the updated variant. Switch to the Board Editor and rearrange the object. Use the <Ratsnest> to update the copper pours and airwires. Re-run the <Auto> auto-router. Afterward, check the DRC again and configure the rest of the errors (Fig. 191).



Fig. 191 Final board

### 4.15.1 Creating the Drill List .drl

To start creating the fabrication files, such as drill information, navigate to 'File' >> 'Run' >> 'Drillcfg.ulp', and select <Open> (Figs. 192 and 193).



Fig. 192 <Run> ULP option

🔄 Run			<b>×</b>
C:\Program	Files (x86)\EAGLE-5.3.0\ulp	🛨 😽 Search ulp	Q
Organize 🔻 New fold	er	8==	- 🗌 🔞
🔆 Favorites	Name	Date modified	Туре 🖍
	del-devices.ulp	3/28/2002 2:09 AM	ULP File
Nesktop	del-empty-devices.ulp	3/28/2002 2:09 AM	ULP File
	del-pack-sym.ulp	3/28/2002 2:09 AM	ULP File 🗉
	dif40.ulp	3/29/2005 2:14 AM	ULP File
	dose-pro.ulp	3/28/2002 2:09 AM	ULP File
	drill-aid.ulp	10/27/2008 5:03 AM	ULP File
	drillcfg.ulp	3/28/2002 2:09 AM	ULP File
	drillegend.ulp	10/27/2008 5:03 AM	ULP File
	drill-legend.ulp	8/4/2004 4:13 AM	ULP File
	dxf.ulp	4/21/2008 5:00 AM	ULP File
	e-attributverwaltung.ulp	10/27/2008 5:03 AM	ULP File
	e-bauteil-erstellen.ulp	4/21/2008 5:00 AM	ULP File
	✓ III		•
File n	ame: drillcfg.ulp	✓ User Language Pro	grams (*.ulp 🔻
		Open	Cancel

Fig. 193 drillcfg.ulp



When prompted for output units, select either <inch> or <mm> (Fig. 194).

Fig. 194 Unit select

Review and edit drill sizes. Minimizing tool changes may save on the total board cost.

Notice in Fig. 195, "T02 0.031in" is pretty close in diameter to "T03 0.039". It is possible to configure either of those lines to a similar drill size, but double-check that the part used will be able to handle the change. Click on <Ok> to proceed and save the file in a new folder, called "Fab":

 $C: \label{eq:layer} C: \$ 

🔄 Eagle: Edit Drill Configuration 🛛 💽
Edit only if you are sure what you do!
ф01 0.018in T02 0.031in T03 0.039in T04 0.125in
Ok
Cancel

Fig. 195 Board drill sizes

### 4.15.2 Creating Drill Location Information .drd and .dri

Navigate to 'File', 'CAM Processor' (Fig. 196).



Fig. 196 CAM processor

File	CAM Processor Layer Wind	r - EAGLE 5.3.0 Pro dow Help	ofessional				
	Open Open recent Save job Close Exit Output Device File	Ctrl+S Ctrl+F4 Alt+X	Board Schematic Drill rack Wheel Job	Mirror Rotate Upside down pos. Coord Quidxplot Quidxplot Optmize Fill pads	Nr 1 1 2 3 4 4 5 6 6 7 8 9 100 111 12 13 14 4 15 16 16 177 18 9 9	Layer Top Route3 Route4 Route5 Route7 Route7 Route8 Route9 Route10 Route11 Route11 Route12 Route13 Route13 Route15 Bottom Pads Vias Unrouted	E
C:\Us	;ers\kkwashnak\	Pr \Desktop\Kwashn	rocess Job Process	Section Descript	tion	Add	Del

Inside CAM Processor window, navigate to 'File', 'Open', 'Job' (Fig. 197).

Fig. 197 CAM job

Dpen CAM Job						×
COO - 🎉 « EAGL	E-5.3	.0 ▶ cam	<b>- - i i j</b>		Search cam	Q
Organize 👻 New f	older				!≡ ▼ [	1 0
🔆 Favorites	n CAM Job					
		excellon.cam			12/13/2006 2:16 AM	CAM File
🧮 Desktop		gerb274x.cam			12/13/2006 2:16 AM	CAM File
🕞 Libraries		gerb274x-4layer.cam			4/21/2008 5:00 AM	CAM File
😹 kkwashnak	=	gerber.cam			12/13/2006 2:16 AM	CAM File
🖳 Computer	-	layout2.cam			12/13/2006 2:16 AM	CAM File
🏭 Local Disk (C:)		schematic.cam			12/13/2006 2:16 AM	CAM File
🔮 CD Drive (D:)						
Open CAM Job       Search cam         Organize        New folder         # Favorites       Image: Computer in the second in the se						
🗣 Network						
Control Panel						
Organize     New folder       Pavorites     Date modified       Type     Image: Computer       Image: Computer     Image: Computer       Image: Computer<						
🌗 Kwashnak						
ul_MCP17025002	Ŧ	•	111			۰.
Fi	le na	me: excellon.cam	•		CAM Processor Job Files (*	.cam 🔻
					• •	

Select the 'excellon.cam' file and click on <Open> (Fig. 198).

Fig. 198 excellon.cam

The CAM Processor will open with a configured "Generate drill data" tab (Fig. 199). Configure section as follows:

- 1) 'Section' "Generate drill data".
- 2) 'Style' Check 'pos. Coord.' and 'Optimize'. Fill pads will be checked, but grayed out.
- 3) 'Layer' Select layers '44 Drills' and '45 Holes' only.
- 4) 'Device' EXCELLON
- 5) 'File' "%N.dhd", (notation indicates filename-drill hole data), click on <File> and save in "Fab" folder (Fig. 200).

ienerate drill data	Style	Nr	Layer	
Section Generate drill data Prompt Output Device EXCELLON  File %Al.drd	Mirror Rotate Upside down Ø pos. Coord Quidqolot Ø Optimize Fill pads	33 34 35 36 37 38 39 40 41 42 43	tFinish bFinish tGlue bGlue tTest bTest tKeepout tRestrict bRestrict vRestrict vRestrict	
Offset X Onch Y Onch		44 45 46 47 48 49 51 52	Drills Holes Milling Measures Document Reference tDocu bDocu	E

Fig. 199 Generate drill data

Job Section Generate drill data Prompt Output Device EXCELLON • File /Projects/pcb_2layer/Fab/%44.drd Offset X Oinch Y Oinch	Style Mirror Rotate Uppide down Pois Coord Quidqholt Optimize Fill pads	Nr 33 34 35 36 37 38 39 40 41 42 43 44 45 46 47 48 49 51	Layer třinish břinish tGlue bGlue tTest bTest tKeepout bKeepout tRestrict vRestrict vRestrict Drills Holes Milling Measures Document Reference tDocu	Ē
--	--	---	--	---

Fig. 200 Configuring file save location

Click on <Process Job> and review "Fab" folder (Fig. 201). The folder is populated with the .drd and .dri files (Fig. 202).

Job Secton Generate dril data Prompt Output Device EXCELLON File %4N.drd Offset V Onch	Style Mirror Rotate Updid down Optic Coord Quidqiot V Gomize V Fill pads	Nr 33 34 35 36 37 38 39 40 41 42 43 44 45 46 47 48 49 51	Layer trinish brinish brinish bollue bollue tTest tKeepou	E
---	---	---	--	---

Fig. 201 Process job generate drill data

3 CAM Processor - C:\Program ile Layer Window Help	Files (x86)\EAGLE-5.	3.0\cam\excellon.ca	m - EAGLE	5.3.0		8
Generate drill data						
Job		Style	Nr	Laver		
Section Generate drill data Prompt		Mirror Rotate	33 34 35	tFinish bFinish tGlue		
Output Device EXCELLON	•	pos. Coord     Quickplot     Optimize	37	tTest bTest tKeepout		
File /Projects/pcb_2	ayer/Fab/%N.drd	I Fill pads	40 41 42 43	bKeepout tRestrict bRestrict vRestrict		
Offset X Dinch			44 45 46	Drills Holes Milling		
Y 0inch			47 48 49 51 52	Measures Document Reference tDocu bDocu		•
(\Users\kkwashnak\Desktop\Kwas	Process Job Proce	ess Section Descrip 2layer\pcb_2layer.br	tion d	Add	Del	
						×
♥ 🕒 C:\Users\kkwashr	ak\Desktop\Kwash	nak\Projects\pcb_2l	yer\Fab	<b>- 4</b> ∳ Se	earch Fal	b 🖇
Organize 🔻 📄 Open	Share with 🔻 🛛 🛚	lew folder		•		0
🔆 Favorites	Name	Date	modi	Туре	Size	
	pcb_2lay	ver.drl 9/21/	2020	DRL File		1 1
🧮 Desktop	pcb_2lay	rer.dri 9/21/	2020	DRI File		1 1
	pcb_2lay	/er.ard 9/21/	2020	UKU File		1 K
	٩					

Fig. 202 Creation of drd and dri files

When exiting the CAM Job, EAGLE prompts to save the job. Create a new folder named "Jobs" in the "pcb\_2layer" project folder and save the updated excellon.cam:

 $\label{eq:c:Userskwashnak} C: Users \ kwashnak \ Desktop \ Kwashnak \ Projects \ pcb_2 layer \ Jobs \ excellon \ .cam$ 

#### 4.15.3 Creating Layer Information (Gerber274x.cam or Custom)

Open CAM Processor, 'File', 'Open', 'Job', 'gerb274x.cam' and click on <Open>.

The directory may have updated due to the previous .cam file save. Check the default install directory for the original cam files (Fig. 203), such as

C:\Program Files (x86)\EAGLE-5.3.0\cam

📟 Open CAM Job			×
C:\Program	n Files (x86)\EAGLE-5.3.0\cam	🛨 🍫 Search cam	٩
Organize 🔻 New fol	der		:= • 🔳 🔞
🔆 Favorites	Name	Date modified	Туре
	excellon.cam	12/13/2006 2:16 AM	CAM File
🧮 Desktop	gerb274x.cam	12/13/2006 2:16 AM	CAM File
	gerb274x-4layer.cam	4/21/2008 5:00 AM	CAM File
	gerber.cam	12/13/2006 2:16 AM	CAM File
	layout2.cam	12/13/2006 2:16 AM	CAM File
	schematic.cam	12/13/2006 2:16 AM	CAM File
		111	
Eile -	1.074	CAMProven	a lah Files (* assa a
File	name: gerb2/4x.cam	CAM Processo     Open	Cancel

Fig. 203 gerb274x.cam

The default standard will have multiple sections, such as 'Component side', 'Solder side', Silk screen CMP', and so on. These sections will be altered for the custom two-layer PCB. When a section tab is selected, click on the <Del> button to remove the page and <Add> to insert a new page.

Section details are described in the following modules.

#### 4.15.3.1 Top Silkscreen .tslk

The Top Silkscreen section instructions are as follows (Fig. 204):

- 1) 'Section' "Top Silkscreen"
- 2) 'Style' Check 'pos. Coord.' and 'Optimize'. Fill pads will be checked, but grayed out.
- 3) 'Layer' Select layer '21 tPlace' and '25 tNames'.
- 4) 'Device' GERBER\_RS274X
- 5) 'File' "%N.tslk", click on <File> and save in "Fab" folder.

Job Section Top Silkscreen Prompt Output Device GERBER_RS274X  File %N.tslk  Style Nr Layer 13 Route13 14 Route14 15 Route15 16 Bottom 17 Pads 18 Vias Quickplot V Optimize V Fill pads  Nr Layer 13 Route13 14 Route14 15 Route15 16 Bottom 17 Pads 18 Vias 19 Unrouted 20 Dimension 21 [tPlace 23 tOrigins	
Offset 24 bOrigins 25 tHames 26 bNames 27 tValues 28 bValues 29 tStop 30 bStop	

Fig. 204 Gerber\_RS274X top silkscreen section

#### 4.15.3.2 Top Stop .tstp

The Top Stop section instructions are as follows (Fig. 205):

- 1) 'Section' "Top Stop"
- 2) 'Style' Check 'pos. Coord.' and 'Optimize'. Fill pads will be checked, but grayed out.
- 3) 'Layer' Select layer '29 tStop'.
- 4) 'Device' GERBER\_RS274X
- 5) 'File' "%N.tstp", click on <File> and save in "Fab" folder.

op Silkscreen	Top Stop	Top Copper	Bottom Copper	Bottom Stop	Bot	tom Silksa	reen	Board Outline	Drills and Hol
Job Section Top S Prompt Output Device File Offset X Oinch Y Oinch	GERBER_R	S274X		Style Mirror Rotate Upside dowr Ø pos. Coord Quickplot Ø Optimize Fill pads	1	Nr 19 20 21 22 23 24 25 26 27 28 29 30 31 31 32 33 34 35 36 27	Layer Unrout Dimens tPlace bPlace tOrigin bOrigin tNames bValues tValues tValues tValues tValues tStop tCream bCream tFinish tGlue bGlue	ed iion s is s s	
			Proce	ess Job Proces	s Sect	tion	escriptior	Add	Del

Fig. 205 Gerber\_RS274X top stop section

#### 4.15.3.3 Top Copper .tcpr

The Top Copper section instructions are as follows (Fig. 206):

- 1) 'Section' "Top Copper"
- 2) 'Style' Check 'pos. Coord.' and 'Optimize'. Fill pads will be checked, but grayed out.
- 3) 'Layer' Select layers '1 Top', '17 Pads', and '18 Vias'.
- 4) 'Device' GERBER\_RS274X
- 5) 'File' "%N.tcpr", click on <File> and save in "Fab" folder.

op Silkscreen	Top Stop	Top Copper	Bottom Copper	Bottom Stop	Bottom S	ilkscreen	Board Outline	Drills and Hole
Job Section Top C Prompt Output Device File Offset X Oinch Y Oinch	GERBER_F	RS274X		Style Mirror Rotate Upside down Ø pos. Coord Quidqolot Ø Optimize Ø Fill pads		Layer 1 Top 2 Route 3 Route 4 Route 5 Route 6 Route 7 Route 10 Route 10 Route 11 Route 11 Route 13 Route 14 Route 15 Route 15 Route 16 Botto 17 Pads 18 Vias 19 Unro	22 23 24 25 26 27 28 29 210 211 212 213 214 215 m	E
						(		

Fig. 206 Gerber\_RS274X top copper section

#### 4.15.3.4 Bottom Copper .bcpr

The Bottom Copper section instructions are as follows (Fig. 207):

- 1) 'Section' "Bottom Copper"
- 2) 'Style' Check 'pos. Coord.' and 'Optimize'. Fill pads will be checked, but grayed out.
- 3) 'Layer' Select layers '16 Bottom', '17 Pads', and '18 Vias'.
- 4) 'Device' GERBER\_RS274X
- 5) 'File' "%N.bcpr", click on <File> and save in "Fab" folder.

op Silkscreen	Top Stop	Top Copper	Bottom Copper	Bottom Stop	Bot	tom Silkscr	een	Board Outline	Drills and Hole
Job Section Botto Prompt Output Device File Offset X 0inch Y 0inch	m Copper	S274X		Style Mirror Rotate Upside down Ø pos. Coord Quickplot Ø Optimize Ø Fill pads	1	Nr 9   10   11   12   13   14   15   16   17   18   19   20   20   21   22   23   24   23   24   25   26   27   26   27   28   28	Layer Route9 Route10 Route11 Route12 Route13 Route14 Bottom Pads Vias Unroute Dimens tPlace bPlace tOrigins bOrigins	) 2 3 4 5 s s s	
			Proc	ess Job Proces	s Sect	27 t	tValues scription	Add	Del

Fig. 207 Gerber\_RS274X bottom copper section

#### 4.15.3.5 Bottom Stop .bstp

The Bottom Stop section instructions are as follows (Fig. 208):

- 1) 'Section' "Bottom Stop"
- 2) 'Style' Check 'pos. Coord.' and 'Optimize'. Fill pads will be checked, but grayed out.
- 3) 'Layer' Select layer '30 bStop'.
- 4) 'Device' GERBER\_RS274X
- 5) 'File' "%N.bstp", click on <File> and save in "Fab" folder.

Job Section Bottom Sta					
Prompt Output Device G File % Offset X Oinch Y Oinch	op ERBER_RS274X	Style Mirror Rotate Upside down Pos. Coord Quidplot Potmize Fill pads	Nr 22 23 24 25 26 27 28 29 30 31 32 33 34 35 36 37 38 39 40	Layer bPlace tOrigins bOrigins tNames bNames tValues bValues tStop bStop tCream bCream tFinish bFinish tGlue bGlue tTest bTest tKeepout bKeepout	E

Fig. 208 Gerber\_RS274X bottom stop section

#### 4.15.3.6 Bottom Silkscreen .bslk

The Bottom Silkscreen section instructions are as follows (Fig. 209):

- 1) 'Section' "Bottom Silkscreen"
- 2) 'Style' Check 'pos. Coord.' and 'Optimize'. Fill pads will be checked, but grayed out.
- 3) 'Layer' Select layer '22 bPlace'.
- 4) 'Device' GERBER RS274X
- 5) 'File' "%N.bslk", click on <File> and save in "Fab" folder.

Top Silkscreen	Top Stop	Top Copper	Bottom Copper	Bottom Stop	Bot	ttom Silksc	reen	Board Outline	Drills and Hole
Job Section Botto Prompt	om Silkscreen			Style Mirror Rotate		Nr 13 14 15	Layer Route13 Route14 Route15	1	· · · · · · · · · · · · · · · · · · ·
Output Device	GERBER_R	!S274X	<b></b>	Upside dowr  pos. Coord  Quidxplot  Optimize  Fill pads	ו	16 17 18 19 20 21	Bottom Pads Vias Unroute Dimens tPlace	ed ion	Ŧ
File Offset X Oinch Y Oinch	%N.bslk					22 23 24 25 26 27 28 29 30 31	bPlace tOrigins bOrigin tNames bNames tValues bValues tStop bStop tCream	5	

Fig. 209 Gerber\_RS274X bottom silkscreen section
### 4.15.3.7 Board Outline .dout

The Board Outline section instructions are as follows (Fig. 210):

- 1) 'Section' "Board Outline"
- 2) 'Style' Check 'pos. Coord.' and 'Optimize'. Fill pads will bechecked, but grayed out.
- 3) 'Layer' Select layer '20 Dimension'.
- 4) 'Device' GERBER\_RS274X
- 5) 'File' "%N.dout", click on <File> and save in "Fab" folder.

Job         Section       Board Outline         Prompt       Mirror         Output       Upside down         Output       Ø pos. Coord         Quickplot       Quickplot         File       %N.dout         Offset       %N.dout         X       Oinch         Y       Oinch	op Silkscreen	Top Stop	Top Copper	Bottom Copper	Bottom Stop	Botto	om Silksa	reen	Board Outline	Drills and Hole
30 bStop 31 tCream	Job Section Boar Prompt Output Device File Offset X 0inch Y 0inch	d Outline GERBER_R	.S274X		Style Mirror Rotate Upside down Ops. Coord Quickplot Optimize Fill pads		Vr 13 14 15 16 17 18 19 20 21 22 23 24 25 26 27 28 29 30 31	Layer Routel Routel Bottor Pads Vias Unrou Dimen tPlace bPlace tOrigin tName bOrigi tName bValue tStop bStop tCrean	13 14 15 15 n ted sion 15 ns 15 15 15 15 15 15 15 15 15 15 15 15 15	

Fig. 210 Gerber\_RS274X board outline section

### 4.15.3.8 Drills and Holes .ddrl

The Drills and Holes section instructions are as follows (Fig. 211):

- 1) 'Section' "Drills and Holes"
- 2) 'Style' Check 'pos. Coord.' and 'Optimize'. Fill pads will be checked, but grayed out.
- 3) 'Layer' Select layers '44 Drills' and '45 Holes'.
- 4) 'Device' GERBER\_RS274X
- 5) 'File' "%N.ddrl", click on <File> and save in "Fab" folder.

	Top Stop	Top Copper	Bottom Copper	Bottom Stop	Bottom Silkso	reen Board Outline	Drills and Holes
Job				Style	Nr	Layer	
Section Drills	and Holes			Mirror	33	tFinish	
Descent				Rotate	34	bFinish	
Prompt				Upside down	35	tGlue	
Output				Dos Coord	36	bGlue	
ouput					3/	t lest	
Device	GERBER_R	S274X	<b></b> ]	Quickplot	38	b lest	
				Optimize		bKeepout	
				Fill pads	40	tRestrict	
File	%N.ddrl				42	bRestrict	
					43	vRestrict	
Offset					44	Drills	
					45	Holes	
X Oinch					46	Milling	
V O'					47	Measures	E
Y UINCN					48	Document	
YUINCH					49	Reference	
Y UINCH					51	tuocu	
Y UIICH					51	hDaau	

Fig. 211 Gerber\_RS274X drills and holes section

### 4.15.3.9 Description

The description (Fig. 212) of the custom Gerber\_274x can file provides information (Fig. 213) to the manufacturer about how to handle the Gerber files.

Top Silkscreen	Top Stop	Top Copper	Bottom Copper	Bottom Stop	Bottom Silks	creen	Board Outline	Drills and Hole
Job Section Top S Prompt Output Device File Offset X Oinch Y Oinch	GERBER_F	15274X		Style Mirror Rotate Upside down Ø pos. Coord Quidxplot Ø Optimize Fill pads	Nr 11 14 14 15 16 17 17 16 17 17 16 17 17 16 17 17 16 17 17 16 17 17 16 17 17 16 17 17 17 17 17 17 17 17 17 17 17 17 17	Layer 3 Route1 4 Route1 5 Route1 5 Bottom 7 Pads 9 Vias 9 Vias 9 Unrout 9 Unr	3 4 5 1 ed sion s s s s s	E
					31	tCream		-

Fig. 212 Gerber\_274X description section

E Description of CAM Job 'gerb274x.cam'	<b>.</b>
Headline:	Generates Extended Gerber Format
Generates Extended Gerber Format	
\nThis CAM job consists of eight sections that	t generates data for a two layer board.
\nYou will get eight gerber files that contain d \nTop Silkscreen *.tslk \nTop Sop *.tstp \nTop Copper *.tcpr \nBottom Copper *.bcpr \nBottom Silkscreen *.bslp \nBottom Silkscreen *.bslk \nBoard Outline *.dout \nDrill and Hole *.ddrl	lata for:
kb>Generates Extended Gerber Format \nThis CAM job consists of eight sections \nThis CAM job consists of eight sections (p> \nYnou will get eight gerber files that cont \nTop Silkscreen *.tsik \nTop Copper *.tsp \nYnot Copper *.tsp \nYnot \nBottom Copper *.bsp \nPottom Silkscreen *.bsik \nYnBottom Silkscreen *.dout \nYnBottom Hole *.dout \nYnBottom Hole *.ddrl	> : that generates data for a two layer board. ain data for:
OK	Cancel Undo Redo

Fig. 213 Gerber\_274X description information

```
<b>Generates Extended Gerber Format</b>
\nThis CAM job consists of eight sections that generate data for a two layer board.
\nYou will get eight gerber files that contain data for:
<br>\nTop Silkscreen *.tslk
<br>\nTop Stop *.tstp
<br>\nTop Copper *.tcpr
<br>\nBottom Copper *.bstp
<br>\nBottom Stop *.bstp
<br>\nBottom Silkscreen *.bslk
<br>\nBottom Silkscreen *.bslk
<br>\nDrill and Hole *.ddrl
<br>
```

Code block 1 HTML gerber\_274X description

Finally, navigate to <File>>> <Save Job>, and save the CAM file in the Jobs folder (Fig. 214):

*C:\Users\kkwashnak\Desktop\Kwashnak\Projects\pcb\_2layer\Jobs\gerb27* 4*x.cam* 

	Open Open recent Save job	Ctrl+S	Copper	Bottom Copper	Bottom Stop Style Mirror	Bottom Silksc	Route13	Drills and Hole
D	Close Exit Jutput evice	Ctrl+F4 Alt+X	4X	▼	Rotate     Upside down     pos. Coord     Quickplot     Optimize     Fill pads	14 15 16 17 18 19 20	Route14 Route15 Bottom Pads Vias Unrouted Dimension	E
V V	File Offset Oinch Oinch	vashnak/Project	s/pcb_2laye	er/Fab/%N.tsk		21 22 23 24 25 26 27 28 29 20	tPlace bPlace tOrigins bOrigins tNames bNames tValues bValues tStop bCare	
				Proc	ess Job Proces	s Section D	escription Add	Del

Fig. 214 Save Gerber\_274X job

Click on <Process Job> (Fig. 215).

p Silkscreen	Top Stop	Top Copper	Bottom Copper	Bottom Stop	Bottom Silkso	reen Board Outline	e Drills and Hol
Job Section Top S Prompt Output Device File Offset X Oinch Y Oinch	GERBER_F vashnak/Pr	lS274X ojects/pcb_2lay	▼ er/Fab/%N.tsk	Style Mirror Rotate Upside down Ø pos. Coord Quidxplot Ø Optimize Ø Fill pads	Nr 13 13 14 15 16 17 18 19 20 20 21 22 23 24 25 26 27 28 29 30	Layer Route13 Route14 Route15 Bottom Pads Vias Unrouted Dimension tPlace bPlace tOrigins bOrigins tNames bNames tValues bValues bValues bStop bStop	
						tCream	

Fig. 215 Processing Gerber files

Review all files in Windows Explorer (Fig. 216).

					- • ×
<	C:\Users\kkwashnak\D	esktop\Kwashnak\Pro	jects\pcb_2layer\Fab	<b>-</b> ≁	Search Fab 👂
Organize 🔻	Include in library 🔻	Share with 🔻	New folder	:=	0
쑦 Favorites		Name	Date modi	Туре	Size
		pcb_2layer.bcpr	9/21/2020	BCPR File	23 KB
📃 Desktop		pcb_2layer.bslk	9/21/2020	BSLK File	2 KB
		pcb_2layer.bstp	9/21/2020	BSTP File	1 KB
		pcb_2layer.ddrl	9/21/2020	DDRL File	6 KB
		pcb_2layer.dout	9/21/2020	DOUT File	3 KB
		pcb_2layer.drd	9/21/2020	DRD File	1 KB
		pcb_2layer.dri	9/21/2020	DRI File	1 KB
		pcb_2layer.drl	9/21/2020	DRL File	1 KB
		pcb_2layer.gpi	9/21/2020	GPI File	2 KB
		pcb_2layer.tcpr	9/21/2020	TCPR File	38 KB
		pcb_2layer.tsk	9/21/2020	TSK File	9 KB
		pcb_2layer.tstp	9/21/2020	TSTP File	3 KB
12	items				

Fig. 216 Gerber files complete

#### 4.15.3.10 excellon.cam

File contents of the excellon.cam are stated in code block 2.

```
[CAM Processor Job]
Description[de]="<b>Erzeugt Bohrdaten im Excellon-Format</b>\nDieser CAM-Job kann
verwendet werden, um Bohrdaten im Excellon-Format für numerisch gesteuerte Bohrmaschinen
zu erzeugen.\nDer EXCELLON-Treiber erzeugt eine Bohrdatendatei, die bereits
die\nBohrertabelle im Dateikopf enthält."
Description[en]="<b>Generates Excellon Drill Data </b>\nThis CAM job can be used to
generate data for drilling machines in Excellon format.\nThe EXCELLON device
generates a drill data file which contains the drill table in the header of the file.\n"
Section=Sec_1
[Sec_1]
Name[de]="Bohrdaten erzeugen"
Name[en]="Generate drill data"
Prompt[en]=""
Device="EXCELLON"
Wheel=".whl"
Rack=""
Scale=1
Output="C:/Users/kkwashnak/Desktop/Kwashnak/Projects/pcb_2layer/Fab/%N.drd"
Flags="0 0 0 1 0 1 1"
Emulate="0 0 0"
Offset="0.0mil 0.0mil"
Sheet=1
Tolerance="0 0 0 0 0.025 0.025"
Pen="0.0mil 0"
Page="12000.0mil 8000.0mil"
Lavers=" 44 45"
```

#### Code block 2 Custom CAM job for Excellon file

#### 4.15.3.11 gerb274x.cam

File contents of gerb274x.cam are stated in code block 3.

```
[CAM Processor Job]
Description[de]="<b>Erzeugt Extended-Gerber-Format (RS274X)</b>\nDieser CAM-Job
besteht aus fünf Arbeitsschritten und erzeugt Fertigungsdaten für eine zweilagige
Platine.\nDie fünf Gerberdateien enthalten Daten für:<br><\nBestückungsseite</p>
*.cmp<br>\nLötseite
                       *.sol<br>\nBestückungsdruck
                                                    oben
                                                               *.plc<br>\nLötstoplack
Bestückungsseite *.stc<br>\nLötstoplack Lötseite *.sts<br>"
Description[en]="<b>Generates Extended Gerber Format</b>\n\n\nThis CAM job consists
of eight sections that generates data for a two layer board.\n\n\nYou will get
eight gerber files that contain data for:\n<br>\nTop Silkscreen *.tslk\n<br>\nTop Stop
                            *.tcpr\n<br>\nBottom Copper
                                                          *.bcpr\n<br>\nBottom Stop
*.tstp\n<br>\nTop Copper
*.bstp\n<br>\nBottom Silkscreen
                                *.bslk\n<br>\nBoard Outline *.dout\n<br>\nDrill and
Hole *.ddrl\n<br>"
Section=Sec 1
Section=Sec_2
Section=Sec_3
Section=Sec_4
Section=Sec 5
Section=Sec_6
Section=Sec_7
Section=Sec 8
```

```
[Sec 1]
Name[de]="Bestückungsseite"
Name[en]="Top Silkscreen"
Prompt[en]=
Device="GERBER RS274X"
Wheel=".whl"
Rack=""
Scale=1
Output="C:/Users/kkwashnak/Desktop/Kwashnak/Projects/pcb_2layer/Fab/%N.tsk"
Flags="0 0 0 1 0 1 1"
Emulate="0 0 0"
Offset="0.0mil 0.0mil"
Sheet=1
Tolerance="0 0 0 0 0 0"
Pen="0.0mil 0"
Page="12000.0mil 8000.0mil"
Layers=" 21"
[Sec_2]
Name[de]="Bestückungsseite"
Name[en]="Top Stop"
Prompt[en]="
Device="GERBER_RS274X"
Wheel=".whl"
Rack=""
Scale=1
Output="C:/Users/kkwashnak/Desktop/Kwashnak/Projects/pcb_2layer/Fab/%N.tstp"
Flags="0 0 0 1 0 1 1"
Emulate="0 0 0"
Offset="0.0mil 0.0mil"
Sheet=1
Tolerance="0 0 0 0 0 0"
Pen="0.0mil 0"
Page="12000.0mil 8000.0mil"
Layers=" 29"
[Sec_3]
Name[de]="Bestückungsseite"
Name[en]="Top Copper'
Prompt[en]="
Device="GERBER_RS274X"
Wheel=".whl"
Rack=""
Scale=1
Output="C:/Users/kkwashnak/Desktop/Kwashnak/Projects/pcb 2layer/Fab/%N.tcpr"
Flags="0 0 0 1 0 1 1"
Emulate="0 0 0"
Offset="0.0mil 0.0mil"
Sheet=1
Tolerance="0 0 0 0 0 0"
Pen="0.0mil 0"
Page="12000.0mil 8000.0mil"
Layers=" 1 17 18"
```

```
[Sec_4]
Name[de]="Bestückungsseite"
Name[en]="Bottom Copper"
Prompt[en]="
Device="GERBER RS274X"
Wheel=".whl"
Rack=""
Scale=1
Output="C:/Users/kkwashnak/Desktop/Kwashnak/Projects/pcb 2layer/Fab/%N.bcpr"
Flags="0 0 0 1 0 1 1"
Emulate="0 0 0"
Offset="0.0mil 0.0mil"
Sheet=1
Tolerance="0 0 0 0 0 0"
Pen="0.0mil 0"
Page="12000.0mil 8000.0mil"
Layers=" 16 17 18"
[Sec 5]
Name[de]="Bestückungsseite"
Name[en]="Bottom Stop"
Prompt[en]=""
Device="GERBER RS274X"
Wheel=".whl"
Rack="'
Scale=1
Output="C:/Users/kkwashnak/Desktop/Kwashnak/Projects/pcb_2layer/Fab/%N.bstp"
Flags="0 0 0 1 0 1 1"
Emulate="0 0 0"
Offset="0.0mil 0.0mil"
Sheet=1
Tolerance="0 0 0 0 0 0"
Pen="0.0mil 0"
Page="12000.0mil 8000.0mil"
Layers=" 30"
[Sec_6]
Name[de]="Bestückungsseite"
Name[en]="Bottom Silkscreen"
Prompt[en]="
Device="GERBER_RS274X"
Wheel=".whl"
Rack=""
Scale=1
Output="C:/Users/kkwashnak/Desktop/Kwashnak/Projects/pcb 2layer/Fab/%N.bslk"
Flags="0 0 0 1 0 1 1"
Emulate="0 0 0"
Offset="0.0mil 0.0mil"
Sheet=1
Tolerance="0 0 0 0 0 0"
Pen="0.0mil 0"
Page="12000.0mil 8000.0mil"
Layers=" 22"
```

```
[Sec 7]
Name[de]="Bestückungsseite"
Name[en]="Board Outline"
Prompt[en]="
Device="GERBER RS274X"
Wheel=".whl"
Rack=""
Scale=1
Output="C:/Users/kkwashnak/Desktop/Kwashnak/Projects/pcb 2layer/Fab/%N.dout"
Flags="0 0 0 1 0 1 1"
Emulate="0 0 0"
Offset="0.0mil 0.0mil"
Sheet=1
Tolerance="0 0 0 0 0 0"
Pen="0.0mil 0"
Page="12000.0mil 8000.0mil"
Layers=" 20"
Colors="121212121212121266488888888888844111133
[Sec 8]
Name[de]="Bestückungsseite"
Name[en]="Drills and Holes"
Prompt[en]="
Device="GERBER RS274X"
Wheel=".whl"
Rack=""
Scale=1
Output="C:/Users/kkwashnak/Desktop/Kwashnak/Projects/pcb_2layer/Fab/%N.ddrl"
Flags="0 0 0 1 0 1 1"
Emulate="0 0 0"
Offset="0.0mil 0.0mil"
Sheet=1
Tolerance="0 0 0 0 0 0"
Pen="0.0mil 0"
Page="12000.0mil 8000.0mil"
Layers=" 44 45"
```

Code block 3 Custom CAM job for Gerber files

### 4.16 Step 16: Formulate Board Instructions

Before proceeding to combine all files into one package, such as a compressed file format .zip, a generic instruction sheet must be generated. In the Windows Explorer, navigate to

### C:\Users\kkwashnak\Desktop\Kwashnak\Projects\pcb 2layer\Fab

Right-click inside the File Explorer "Fab" folder contents area, hover over 'New', and select 'Text Document' (Fig. 217).



Fig. 217 Creating an instruction file

Name the text file "Instructions" (Fig. 218).

Kwashnak 🕨 Pro	ojects 🕨 pcb_2layer 🕨 Fab			Search Fab 🔎
Organize 👻 🧻 Open 👻	Share with 👻 🦳 Print	New folder		)= • 🔟 🔞
🚖 Favorites	Name	Date modi	Туре	Size
	Instructions.txt	9/23/2020	Text Docu	0 KB
🧮 Desktop	pcb_2layer.bcpr	9/21/2020	BCPR File	23 KB
	pcb_2layer.bslk	9/21/2020	BSLK File	2 KB
	pcb_2layer.bstp	9/21/2020	BSTP File	1 KB
	pcb_2layer.ddrl	9/21/2020	DDRL File	6 KB
	pcb_2layer.dout	9/21/2020	DOUT File	3 KB
	pcb_2layer.drd	9/21/2020	DRD File	1 KB
	pcb_2layer.dri	9/21/2020	DRI File	1 KB
	pcb_2layer.drl	9/21/2020	DRL File	1 KB
	pcb_2layer.gpi	9/21/2020	GPI File	2 KB
	pcb_2layer.tcpr	9/21/2020	TCPR File	38 KB
	pcb_2layer.tsk	9/21/2020	TSK File	9 KB
	pcb_2layer.tstp	9/21/2020	TSTP File	3 KB
Instructions.txt Date m Text Document	odified: 9/23/2020 10:26 AM Size: 0 bytes	Date creat	ed: 9/23/20201	0:26 AM

Fig. 218 Instructions text file

Edit the text file with similar contents as shown in code block 4:

Author Author Email	Ken Kwashnak ***@***_***
Author Phone	· · · · · · · · · · · · · · · · · · ·
Date	September 23, 2020
Board Name	Demo Board (pcb_2layer)
Board Version	1.0 (baseline)
Board Dimension	1.5" × 1.5" × 0.063"
Board Material	FR4
Copper Weight	1 oz
Lead Free	Yes
Soldermask	Both Sides
Soldermask Color	Green
Silkscreen	Both Sides
Silkscreen Color	White
The attached fabr	ication files were processed from EAGLE v5.3.0:
<ul> <li>pcb_2layer.drl</li> </ul>	(Drill List)
<ul> <li>pcb_2layer.drd</li> </ul>	(excellon.cam, Drill Location Information)
<ul> <li>pcb_2layer.dri</li> </ul>	(excellon.cam, Drill Location Information)
<ul> <li>pcb_2layer.tslk</li> </ul>	(Gerber274x.cam, Top Silkscreen)
<ul> <li>pcb_2layer.tstp</li> </ul>	(Gerber274x.cam, Top Stop)
<ul> <li>pcb_2layer.tcpr</li> </ul>	(Gerber274x.cam, Top Copper)
<ul> <li>pcb_2layer.bcpr</li> </ul>	(Gerber274x.cam, Bottom Copper)
<ul> <li>pcb_2layer.bstp</li> </ul>	(Gerber274x.cam, Bottom Stop)
<ul> <li>pcb_2layer.bslk</li> </ul>	(Gerber274x.cam, Bottom Silkscreen)
<ul> <li>pcb_2layer.dout</li> </ul>	(Gerber274x.cam, Board Outline)
<ul> <li>pcb_2layer.ddrl</li> </ul>	(Gerber274x.cam, Drills and Holes)
- pcb_2layer.gpi	(Gerber274x.cam, Photoplotter)
Additional Notes:	
- Plated Through H	Holes, Sized for No 4 Holes, Quantity 4

Code block 4 Instructions file content

## 4.17 Step 17: Package Files

Packaging the files is very simple in the Windows Explorer. Navigate to the following location (Fig. 219):

C:\Users\kkwashnak\Desktop\Kwashnak\Projects\pcb 2layer

Organize 👻 Include in library		Share with 👻 New f	older	8==	- 🔟	(
Favorites	^	Name	Date modified	Туре	Size	
		퉬 Fab	9/21/2020 11:34 PM	File folder		
Desktop		퉬 Jobs	9/21/2020 11:33 PM	File folder		
🥃 Libraries		💰 eagle.epf	9/22/2020 12:26 AM	EPF File		
<u>k</u> kkwashnak		pcb_2layer.b#1	9/16/2020 12:05 AM	B#1 File		
🖳 Computer	E	pcb_2layer.b#2	9/16/2020 12:00 AM	B#2 File		
🏭 Local Disk (C:)		pcb_2layer.b#3	9/15/2020 11:50 PM	B#3 File		
🔮 CD Drive (D:)		pcb_2layer.b#4	9/15/2020 11:07 PM	B#4 File		
坖 VirtualBoxFiles (\\vboxsr		pcb_2layer.b#5	9/15/2020 10:11 PM	B#5 File		
👊 Network		pcb_2layer.b#6	9/15/2020 10:08 PM	B#6 File		
📴 Control Panel		pcb_2layer.b#7	9/15/2020 10:01 PM	B#7 File		
Recycle Bin		pcb_2layer.b#8	9/15/2020 9:58 PM	B#8 File		
🍌 Kwashnak		pcb_2layer.b#9	9/15/2020 9:47 PM	B#9 File		
ul_MCP17025002ETO		pcb_2layer.bcpr	9/21/2020 11:32 PM	BCPR File		
ul_PTS647SK38SMTR2LFS	+ 4					

Fig. 219 Fab folder location

Right-click the <u>Fab</u> folder, mouse over <Send to>, and select <Compressed (zipped) folder> (Figs. 220 and 221).

	Dura	ianta Alanak Olava						
Washnak F	Proj	jects • pcb_2laye	r •				• • search	pcb 🎾
Organize 🔻 🛛 🛜 Open	Incl	lude in library 🔻	Share with 🔻	New folder				0
🔆 Favorites	Â	Name	*	Date modified		Туре	Size	<b>^</b>
		퉬 Fab	-	0/01/0000 11-0	<b>≜</b> PM	File folder		
🧮 Desktop		퉬 Jobs	Open		PM	File folder		
🥽 Libraries		🖌 eagle	Open in new wind	w	AM	EPF File	20 KB	=
🔋 kkwashnak		pcb_2	7-Zip	+	AM	B#1 File	14 KB	
🖳 Computer	Ξ	pcb_2	CRC SHA	•	AM	B#2 File	14 KB	
🏭 Local Disk (C:)		pcb_2	Share with	•	PM	B#3 File	14 KB	
🔮 CD Drive (D:)		pcb_2	Restore previous v	ersions	PM	B#4 File	14 KB	
🖵 VirtualBoxFiles (\\vboxsr		pcb_2	Include in library		PM	B#5 File	14 KB	
🙀 Network		pcb_2	include in library		Ьм	R#6 File	1/L KR	
📴 Control Panel		pcb_2	Send to	•		Compressed (zipped	d) folder	
👿 Recycle Bin		pcb_2	Cut			Desktop (create sho	rtcut)	
🔒 Kwashnak		pcb_2	Conv			Documents		
ul_MCP17025002ETO		pcb_2	copy			Fax recipient		
ul_PTS647SK38SMTR2LFS	÷	pcb 2	Create shortcut			Mail recipient		-
Fab Date mod	fied	d: 9/21/2020	Delete			VirtualBoxFiles (\\vb	ooxsrv) (E:)	
File folder			Rename		-			1
1			Properties					

Fig. 220 Compressing to a .zip file

🗲 🕞 🗢 🚺 C:\Users\kkv	vashnak\Desktop\Kwashnak\Proje	cts\pcb_2layer	<b>-</b> €	Search pc     Search	b
Organize 👻 🛛 🛜 Open	✓ Share with ▼ New for	lder			?
🔆 Favorites	Name	Date modi	Туре	Size	
	퉬 Fab	9/23/2020	File folder		
🧮 Desktop	🌗 Jobs	9/21/2020	File folder		
	💰 eagle.epf	9/22/2020	EPF File	20 KB	
	🔒 Fab.zip	9/23/2020	Compressed (zi	21 KB	
	pcb_2layer.b#1	9/16/2020	B#1 File	14 KB	
	pcb_2layer.b#2	9/16/2020	B#2 File	14 KB	
	pcb_2layer.b#3	9/15/2020	B#3 File	14 KB	
	pcb_2layer.b#4	9/15/2020	B#4 File	14 KB	
	pcb_2layer.b#5	9/15/2020	B#5 File	14 KB	
	pcb_2layer.b#6	9/15/2020	B#6 File	14 KB	
	pcb_2layer.b#7	9/15/2020	B#7 File	14 KB	
	pcb_2layer.b#8	9/15/2020	B#8 File	14 KB	
	pcb_2layer.b#9	9/15/2020	B#9 File	14 KB	
	pcb_2layer.bcpr	9/21/2020	BCPR File	23 KB	
Fab.zip	Date modified: 9/23/	2020 1:06 PM			

Fig. 221 Compressed fab file

Rename the "Fab" compressed zip file to "pcb\_2layer\_fab" (ensure the .zip extension is still attached to the file name) (Fig. 222).

	t • Projects • pcb_ziayer •		•	Search pcb	· •
Organize 🔻 🛛 🛜 Open	<ul> <li>Share with </li> <li>New folder</li> </ul>	er			0
☆ Favorites	Name	Date modi	Туре	Size	
	🌗 Fab	9/23/2020	File folder		
📃 Desktop	📗 Jobs	9/21/2020	File folder		
	💰 eagle.epf	9/22/2020	EPF File	20 KB	
	pcb_2layer_fab.zip	9/23/2020	Compressed (zi	21 KB	
	pcb_2layer.b#1	9/16/2020	B#1 File	14 KB	
	pcb_2layer.b#2	9/16/2020	B#2 File	14 KB	
	pcb_2layer.b#3	9/15/2020	B#3 File	14 KB	
	pcb_2layer.b#4	9/15/2020	B#4 File	14 KB	
	pcb_2layer.b#5	9/15/2020	B#5 File	14 KB	
	pcb_2layer.b#6	9/15/2020	B#6 File	14 KB	
	pcb_2layer.b#7	9/15/2020	B#7 File	14 KB	
	pcb_2layer.b#8	9/15/2020	B#8 File	14 KB	
	pcb_2layer.b#9	9/15/2020	B#9 File	14 KB	
	pcb_2layer.bcpr	9/21/2020	BCPR File	23 KB	

Fig. 222 Renaming compressed file

### 4.18 Step 18: Send to Manufacturer

This step can get involved, primarily with maximizing cost per board and delivery time. Online quotes may go through a few iterations because of user inputs (designer might not know what the manufacturer is specifying) or the algorithms used in PCB manufacturer's auto-quoting software might make a few critical (and perhaps wrong) assumptions about the board. Contacting the manufacturer directly may help with reducing costs, but now there is another person in the loop, which might affect the fabrication/delivery time.

For the manufacture's online auto-quoting tool, the software might not state the price of a  $12 \times 12$  FR4 sheet of material, but will charge for the entire sheet for the board. The tool may not specify how many boards will fit onto one sheet. If one board is created, the manufacturer may charge for the use of an entire sheet. If the quote asks for multiple boards, the manufacturer may fill up the FR4 sheet and the cost per board may drop, but how many boards are able to fit onto one sheet? Therefore, using the auto-quote tool might take some time finding the price curve in quantity versus cost.

In addition, when reviewing the quote, compromises in copper weight, silkscreen, or other factors may weigh in for the cost. For example, if the board has various graphics and text on the top and bottom layers, is it really necessary? Will this board be installed in a professional setting? Who will see it? The overall price of the board is also driven by the type of material used, how many holes, how many different sized holes (tooling changes), layers thickness, via or signal trace size, and so on.

Also, before giving any files to a manufacturer, always read their license agreements to understand how the manufacturer will handle project files and materials.

### 4.18.1 Advanced Circuits

The PCB manufacturer Advanced Circuits<sup>37</sup> is used to generate a quote for the twolayer PCB (Fig. 223).



Fig. 223 Advanced circuits welcome screen

Create an account and login with proper credentials. On the left of Fig. 224, click on "Upload Files to FreeDMF".

	Leading the PCB Industry in Quality & Innovation
My4PCB Quote & Order History	100% USA Manufacturing   3rd Largest PCB Manufacturer in North America Log Out Welcome, Ken
Get a Quote Upload Files to FreeDFM™	Advanced Circuits is Here to Serve During this COVID-19 Crisis. To Honor Our Medical Customers, We Are Offering 10% OFF Orders That Pertain to Supporting the COVID Pandemic.
Order from Existing Quote	Open PCB Orders
Get Assembly Quote	(more) Enter Web Order Number to Track Any Current Order.
Secure File Upload Update My Profile	Evicting DCD Quetes Without Orders
\$33 Each Special	No Existing Quotes
\$66 Each Special	(more) Desit Cas Mitst Ver And Lablier Field Field Fred Casts Ver Desit Newborn Best Newborn Lab
BareBones Special	Recall Quote Recall Quote Recall Part Number
RF-Material Special	
Aluminum-Clad Special	PCB Order History No Completed Orders (more)
	Assembly Quotes No Existing Quotes (more)

Fig. 224 Advanced circuits main screen

Enter user email and upload the "pcb\_2layer\_fab.zip" compressed file (Figs. 225 and 226).

	ADVANCED CIRCUITS Leading the PCB Industry in Quality & Innovation
100% USA Manufacturin	g   3rd Largest PCB Manufacturer in North America
FreeDFM	Advanced Circuits is Here to Serve During this COVID-19 Crisis. To Honor Our Medical Customers, We Are Offering 10% OFF Orders That Pertain to Supporting the COVID Pandemic.
What FreeDFM Checks	Upload PCB Design Files Now - And Get Your Free DFM Report! Your Email
FreeDFM worked great! -Stanley Mann.	Confirm Email
Mann Electronic Solutions	Choose File No file chosen Upload ZipFile
Click here to view Company Brochure,	(for your results) (Zipped, Up to 28-Layer, 274-X). If > 28 -layer design. Please contact your sales person at 1-800-979-4722
benefits.	Why Use FreeDFM?
Naming Conventions Display a table of software package extensions and naming conventions.	<ul> <li>Get Free PCB design file check for manufacturability. We check to make sure we have the files and data necessary to build your job</li> <li>Results mailed to you in minutes</li> <li>You get a graphical report, delivered in minutes, showing any potential issues with detail as to their nature and location</li> <li>Reduces possibility of CAM Hold issues after order placement</li> <li>Enables 49% faster total thru-put from order placement to shipment</li> <li>Also provides a price quote with your choice of turntimes</li> </ul>

Fig. 225 Advanced circuits FreeDFM



Fig. 226 FreeDFM sample results

Once uploaded, Advanced Circuits will need to parse the files to match their convention. Pair selected files to the appropriate designation, as in Fig. 227.

	Leading the PCB Industry in	Quality & Innovation	
100% USA Manufacturing	3rd Largest PCB Manufacturer in North /	America	
	FreeDFM Quote S	pecifications	
File IDentification And (	General Information		
is very important that yo yould be that our automat	u identify your files accurately. The ed systems can not readily identif	Primary reason for delayed or undelivered to the contents of your zip file or that a key	ed results v piece of
abrication or design inforr	nation is missing.	,,,,	
Click Here to see a table	of package extensions and file	naming conventions.	
Design File Information			
Please take a few mom	ents to help us identify your file	is.	
File Name:	File Content:		
Instructions.txt	Drawing/Other	~	
pcb_2layer.bcpr	Bottom Copper	~	
pcb_2layer.bslk	Bottom Silkscreen	~	
pcb_2layer.bstp	Bottom Soldermask	~	
pcb_2layer.ddrl	Drawing/Other	~	
pcb_2layer.dout	Drawing/Other	~	
pcb_2layer.drd	NC Drill	~	
pcb_2layer.dri	Drawing/Other	~	
pcb_2layer.drl	NC Drill	~	
pcb_2layer.gpi	Drawing/Other	~	
pcb_2laver.tcpr	Top Copper	<b>~</b>	
poo	Top Silkscreen	~	
pcb_2layer.tslk			

Fig. 227 Advanced circuits quote specifications, page 1

receive accurate FreeDFM results and that your board order will be fabricated correctly. As we are always striving for continuous improvement we would appreciate your feedback regarding files that were incorrectly identified or not identified (freedfm@4pcb.com).

Please enter your	contact informa	tion below:			
Company Name: First Name: Email 1:	*** Ken ***@***.***		Phone: Last Name:	********* Kwashnak	
Please enter your	quote specs and	l attributes belov	<b>v:</b>		
Part #	10107 •	Revision #	1.0	Layer Count	
X Dimension 1 Array X Dim 2 Tab-Rout?	5	Y Dimension Array Y Dim Scoring?	1.5	Array	
Material Type Finish Thickness Soldermask Sides 1 oz  Silks None	FR4         0.062"       ✓         Both Sides       ✓         creen Sides       Both	Einish Plating ( Soldermask Col	✓     Lead Free Solder ✓     Green ✓     ilkscreen Color \	<u>Gold Fingers</u> <u>Copper Wt. (Outer)</u> White ✓ <u>Coppe</u>	None 🗸
Certifications & Qualifications:	IPC Class 2-A6	00 <b>v</b> (F	ricing Subject To File Re	view)	
Vias:	Blind/Buried Vias Microvias? No	None  None None None	<u>Via-In-Pad?</u> ✓ (Fin	None  Vished hole size <= .006	")
Additional Attributes:	<u>Controlled D</u> <u>Controlled Ir</u> <u>Plated Slots?</u> <u>Plated Edges</u> <u>Counterbores</u> <u>Countersinks</u> <u>Castellated H</u>	ielectric? <u>npedance?</u> (Pricin (Pricing Subject 1) (Pricing Subject 1	ng Subject To File Review To File Review) # Non-Plated # Non-Plated tellated Hole Size	<b>*)</b>	
Quantities:	1 5	10 20	ITAR OY	/es 💿 No	
				11	
		S	ubmit		

Once files are associated, input board information, as in Fig. 228.

Fig. 228 Advanced circuits quote specifications page 2

The board has custom Gerber file names that do not conform to an industry standard. Advanced Circuits has gathered similar file names and their association, as shown in Table 13.

Top copper	Bottom copper	Top soldermask	Bottom soldermask	Top silkscreen	Bottom silkscreen	NC drill	Inner layers	Fab print
.TOP	.BOT	.MSK	.STS	.GTO	.GBO	.TAP	.SIG	.DD1
.CMP	.SOL	.STC	.SMB	.OVL	.BSK	.TXT	.G1	.DRD
.GTL	.GBL	.SMT	.MK2	.SLK	.PLS	.DRL	.P1	art.*
.LY1	.LY2	.GTS	.BMK	.SK1	.SK2	.NCD	.GP1	fab.*
art01.pho	art02.pho	.GTM	.GBS	.SS1	.SS2	.DRD	.GNMD	print.*
		.MK1	.GBM	.TSK	ssb01	.TH	.PWR	.DWG
		.TMK	.BSM	.PLC	ssb0226.pho	thruhole.*	.L1	
		.TSM	.SM2	sst0126.pho			plane.*	
		.SM1	sm02.*				art.*	
		sm01.*	sm0228.pho				.GM1	
		sm0128.pho						

Table 13	Filename	alternatives <sup>3</sup>	8
----------	----------	---------------------------	---

Notes:

.TXT may be a drill file or readme file

.DRD may be a drill file or fabrication drawing

.GM1 may either be a mechanical or inner layer

.GKO is a "keep out" layer note -- do not use

Quote information is directed to the email entered in the earlier steps. Navigating back to the Advanced Circuits account main page, the quote will appear in the "Existing PCB Quotes Without Orders". Click on the hyperlink, under the Quote # (Figs. 229–232).

			Lead	ang the PCB Industry in Qu	VANCED RCUITS	2	
My4PCB Quote & Order History	100% USA Manufa Welcome, Ken	cturing   3	rd Largest PCB N	fanufacturer in North Am	erica	L	.og Out
Get a Quote Upload Files to FreeDFM™ Order from Existing Quote	Advanced Circuit Offering 10% OFF Open PCB Or	s is Here Orders T ders	to Serve Dur That Pertain f	ing this COVID-19 Cr o Supporting the CO	isis. To Honor VID Pandemic	Our Medical Customers, We	e Are
Place a Re-Order	No Current Ope	n Orders					
Secure File Upload Update My Profile	Existing PCB	Quotes	Without O	rders	Track		
\$33 Each Special	<u>Quote #</u>	<u>Part#</u>	Rev#	Description	Quote Da	ate	
\$66 Each Special	5583246	10107	1.0		9/23/202	0 11:54:36 AM	
BareBones Special	(IIIOIte) Don'	t See Wha	at You Are Loo	king For? Enter Your	Quote Number	r or Part Number Here.	
RF-Material Special			Recall Quot	e		Recall Part Number	
Aluminum-Clad Special	PCB Order His	story					
	(more)	rders					
	Assembly Que	otes					
	(more)	nes					

Fig. 229 Quote number



#### Click to View FreeDFM Results

Part #: 10107	Revision: 2.0	Date: 9/23/2020	Quote #: 5583593
Self			Quote valid for 30 days

#### **STANDARD SPEC Price Matrix:**

(Qty = Pieces, Not Arrays)

- Standard Spec bare board pricing assumes lead-free HAL (alternatively, board finish may be upgraded to ENIG or Silver at no additional cost in the rare event lead-free HAL becomes temporarily unavailable), FR4, green mask, white silkscreen, and non-plated slots.
   We now offer both non-plated and plated thru holes on all of our standard spec boards (if no specifications provided, holes will be
- We now offer both non-plated and plated thru holes on all of our standard spec boards (if no specifications provided, noise will be plated).
   INCLUDES UL markings and date codes. 94V-0 is added if requested at time of order entry or if noted in files. Minimum finished hole size .010°. Hole tolerance +/-.005°. Minimum Trace/Space..005°.
   All files receive full CAM review and inspection to IPC class 2-A600.
   If your design includes countersink/counterbore or if you have controlled dielectric/stackup or impedance requirements, refer to

b. If your design includes countersink/counterbore or if you have controlled dielectric/stackup or impedance requirements, refer to Custom Spec matrix.
 6. Electrical Test is optional for 0-6 layer board. Unit price does not include shipping and handling. AC Terms and Conditions apply.
 7. Advanced Circuits has a 90 day limited warranty and we will not accept liability for any cost in addition to the value of the bare boards including but not limited to components, labor, business interruptions and any other consequential damages or losses. Advanced Circuits disclaims any open-ended acceptance of liability for losses beyond our control.

#### To Place Order, Click on UNIT PRICE below:

Qty	Same Day	1-day	2-day	3-day	4-day	5-day	E Test(Lot)	AS A SPEC & PRICING ALTERNATIVE THE STANDARD SPEC MATRIX ASSUMES:
3	<u>\$352.17</u>	<u>\$220.06</u>	<u>\$169.25</u>	<u>\$128.60</u>	<u>\$124.53</u>	<u>\$120.47</u>	\$75.00	
6	<u>\$176.09</u>	<u>\$110.03</u>	<u>\$84.62</u>	<u>\$64.30</u>	<u>\$62.27</u>	<u>\$60.23</u>	\$90.00	See #1 thru #7 above.
12	<u>\$92.29</u>	<u>\$59.26</u>	<u>\$46.56</u>	<u>\$36.40</u>	<u>\$35.38</u>	<u>\$34.37</u>	\$120.00	
24	<u>\$46.15</u>	<u>\$29.63</u>	<u>\$23.28</u>	<u>\$18.20</u>	<u>\$17.69</u>	<u>\$17.18</u>	\$144.20	
Tooling	NRE = \$0	-						

#### CUSTOM SPEC Price Matrix: (Qty = Pieces, Not Arrays)

1. The Custom Spec bare board Matrix reflects pricing for the EXACT SPECS you QUOTED (except as noted in the comments).

- 1. The Custom Spec bare board Matrix reflects pricing for the ECACET SPECS you QUOTED (except as noted in the comments).
   2. Our default multi-layer FR4 material is Isola 185HR or equivalent.
   3. All files receive full CAM review and inspection to the certification / qualification you select below.
   4. INCLUDES UL markings and date codes. 94V-0 is added if requested at time of order entry or if noted in files. Finished hole tolerance
   +/-.003"(if requested). Unit price does not include shipping and handling. AC Terms and Conditions apply.
   5. Advanced Circuits has a 90 day limited warranty and we will not accept liability for any cost in addition to the value of the bare boards including but
   not limited to components, labor, business interruptions and any other consequential damages or losses. Advanced Circuits disclaims any open-ended
   acceptance of liability for losses beyond our control.
   Place Order, Clinker, publics, balorum nded

#### To Place Order, Click on UNIT PRICE below:

Qty	Same Day	1-day	2-day	3-day	4-day	1-week	2-week	3-week	4-week
3	<u>\$309.76</u>	<u>\$270.19</u>	<u>\$181.14</u>	<u>\$129.19</u>	<u>\$116.82</u>	<u>\$111.88</u>	<u>\$106.93</u>	<u>\$97.03</u>	<u>\$87.14</u>
6	<u>\$156.26</u>	<u>\$136.27</u>	<u>\$91.29</u>	<u>\$65.06</u>	<u>\$58.81</u>	<u>\$56.31</u>	<u>\$53.81</u>	<u>\$48.82</u>	<u>\$43.82</u>
12	<u>\$79.51</u>	<u>\$69.31</u>	<u>\$46.37</u>	<u>\$32.99</u>	<u>\$29.81</u>	<u>\$28.53</u>	<u>\$27.26</u>	<u>\$24.71</u>	<u>\$22.16</u>
24	<u>\$41.13</u>	<u>\$35.83</u>	<u>\$23.91</u>	<u>\$16.96</u>	<u>\$15.30</u>	<u>\$14.64</u>	<u>\$13.98</u>	<u>\$12.66</u>	<u>\$11.33</u>
Tooling	NRE = \$163.77	(Tooling	waived when	re-ordered)		* Testing =	\$144.20		

\* ET charges shown above do not include Hi-Pot test charges.

Fig. 230 Quote information

Modify Specs			Viewer (Click Update Quote to change Active Quote to Iteration Spec	rs Selected)
Copper Outers:	1 oz 🗸		Plugin Help	
Copper Inners:	0.5 oz ¥			🗹 s1
Material Thickness	0.062"			🗹 m1
Waterial Thickness.	0.062			11
Create Nev	w Ouote Iteration			
				🗹 lb
Active Quote	033105205583593			✓ m2
Ousta	5592502			🗹 arp
QUOLE	5565595 10107	Î		
MATERIAL TYPE	FR4			Trace/Space Thresholds
MATERIAL THICK	0.062		KK KK	
COPPER OUTERS	1	- 18		
COPPER INNERS	0	- 18		FIRST BACK NEXT
TRACE	0.008			No orrors at this
SM HOLE	0.018	- 18	· • • • • • • • • • • • • • • • • • • •	thread and
LAYER	2			threshold.
DIMENSION_1	1.5			
DIMENSION_2	1.5			Ducklama Automatically
PLATING	LFSolder			Froblems Automatically
TRACEINNER	0.008			Fixed by FreeDFM.com
GOLD	None			To view a list of potential
MOUNTS_TOP	0			manufacturing problems with
MOUNTS_BOTTOM	0			fixed please click here
PITCH	0			lixed, please click liele.
SOLDER_SIDES	2			
SOLDER_TYPE	LPI			1
SILKSCREEN_SIDES	1			
ROUTE_POINTS	4			
TAB_ROUTE_ARRAY	No			
SCORING	No			
REVISION	2.0			
SOLDER_COLOR	Green			
SILKSCREEN COLOR	R White	<b>~</b>		

Fig. 231 Advanced circuits DFM, page 1

pecs	·			teratio		Story (	Quote Var	Tations)		
Origin	nal ———			Action	Ac	tive Quo	te	Itera	tion	
Outer	Copper:	1	Vi	iewing		1	Origin	nal		
Outer	Trace:	0.008				-	-			
Inner	Copper:	0								
Inner	Trace:	0.008								
Min T	Trace/Space:	0.008								
Mater	ial Thickness	: 0.062								
Min H	Iole Size:	0.018								
			ע							
			4							۱.
ricii	<b>ng</b> *(Pricing	is based o	n the l	nominai	Trace/	Space of	0.000)			
ricii Standa	ng *(Pricing ard Pricing —	is based o	on the l	nominal	Trace/	Space of	0.000)			
ricii Standa Qty	ng *(Pricing ard Pricing — Same Day	is based o	ay	nominal 2-Day	Trace/	3-Day	4-Day	5-Day	ET	est(Lot)
ricii Standa Qty 3	ng *(Pricing ard Pricing	7 <b>1-D</b> \$220.0	ay 6	<b>2-Day</b> \$169.25	Trace/	<b>3-Day</b> 28.60	<b>4-Day</b> \$124.53	<b>5-Day</b> \$120.47	ET( \$75.00	est(Lot)
Standa Qty 3 6	ng *(Pricing ard Pricing	7 <b>1-D</b> \$220.0 \$110.0	ay 16 5	<b>2-Day</b> \$169.25 \$84.62	Trace/ \$12 \$64	<b>3-Day</b> 28.60 4.30	<b>4-Day</b> \$124.53 \$62.27	<b>5-Day</b> \$120.47 \$60.23	ET \$75.00 \$90.00	est(Lot) ) )
Standa Qty 3 6 12	ng *(Pricing ard Pricing	7 <b>1-D</b> \$220.0 \$110.0 \$59.26	ay 96 8 93 8	<b>2-Day</b> \$169.25 \$84.62 \$46.56	1race/ \$12 \$64 \$36	<b>3-Day</b> 28.60 4.30 5.40	<b>4-Day</b> \$124.53 \$62.27 \$35.38	<b>5-Day</b> \$120.47 \$60.23 \$34.37	ET0 \$75.00 \$90.00 \$120.0	est(Lot) ) ) )()
Qty           3           6           12           24	ng *(Pricing ard Pricing	7 1-D \$220.0 \$110.0 \$59.26 \$29.63	<b>ay</b> 96 8 93 8 9 8	<b>2-Day</b> \$169.25 \$84.62 \$46.56 \$23.28	\$12 \$12 \$64 \$36 \$18	<b>3-Day</b> 28.60 4.30 5.40 8.20	4-Day \$124.53 \$62.27 \$35.38 \$17.69	<b>5-Day</b> \$120.47 \$60.23 \$34.37 \$17.18	ET \$75.00 \$90.00 \$120.0 \$144.2	est(Lot) ) ) ) ) 00 20
Standa Qty 3 6 12 24 Toolin	ng *(Pricing ard Pricing	7 1-D \$220.0 \$110.0 \$59.26 \$29.63 00	ay 96 \$ 33 \$ 6 \$	<b>2-Day</b> \$169.25 \$84.62 \$46.56 \$23.28	\$12 \$64 \$36 \$18	<b>3-Day</b> 28.60 4.30 5.40 3.20	4-Day \$124.53 \$62.27 \$35.38 \$17.69	<b>5-Day</b> \$120.47 \$60.23 \$34.37 \$17.18	ET \$75.00 \$90.00 \$120.0 \$144.2	est(Lot) ) ) )0 20
Standa Standa Qty 3 6 12 24 Toolir	ng *(Pricing ard Pricing	7 1-D \$220.0 \$110.0 \$59.26 \$29.63 00	ay 96 9 93 9 93 9 93 9	<b>2-Day</b> \$169.25 \$84.62 \$46.56 \$23.28	\$12 \$12 \$64 \$36 \$18	<b>3-Day</b> 28.60 4.30 5.40 3.20	4-Day \$124.53 \$62.27 \$35.38 \$17.69	<b>5-Day</b> \$120.47 \$60.23 \$34.37 \$17.18	ET0 \$75.00 \$90.00 \$120.0 \$144.2	est(Lot) ) ) ) 00 20
Standa Standa Qty 3 6 12 24 Toolin Custor	ng *(Pricing same Day \$352.17 \$176.09 \$92.29 \$46.15 ng NRE = \$0. m Pricing	7 <b>1-D</b> \$220.0 \$110.0 \$59.26 \$29.63 00	ay 96 § 33 §	<b>2-Day</b> \$169.25 \$84.62 \$46.56 \$23.28	\$12 \$64 \$30 \$18	<b>3-Day</b> 28.60 4.30 5.40 3.20	4-Day \$124.53 \$62.27 \$35.38 \$17.69	<b>5-Day</b> \$120.47 \$60.23 \$34.37 \$17.18	ET \$75.00 \$90.00 \$120.0 \$144.2	est(Lot) ) ) ) ) ) ) ) ) ) ) ) ) ) ) ) ) ) )
Standa Qty 3 6 12 24 Toolin Custor Qty	ng *(Pricing same Day \$352.17 \$176.09 \$92.29 \$46.15 ng NRE = \$0. m Pricing Same Day	7 1-D \$220.0 \$110.0 \$59.26 \$29.63 00 1-Day	ay 06 \$ 03 \$ 5 \$ 5 \$ 5 \$ 5 \$ 5 \$	<b>2-Day</b> \$169.25 \$84.62 \$46.56 \$23.28 <b>ay 3-</b>	1race/ 312 \$64 \$18 \$18	<b>3-Day</b> 28.60 4.30 5.40 3.20 <b>4-Day</b>	4-Day \$124.53 \$62.27 \$35.38 \$17.69 1-week	5-Day \$120.47 \$60.23 \$34.37 \$17.18 V Z-week	ETr \$75.00 \$90.00 \$120.0 \$144.2	est(Lot) ) ) ) 0 0 20 4-weel
Standa Qty 3 6 12 24 Toolin Custon Qty 3	ng *(Pricing Same Day \$352.17 \$176.09 \$92.29 \$46.15 mg NRE = \$0. m Pricing	Is based c           7         1-D           \$220.0         \$110.0           \$59.26         \$29.63           00         \$227.019	ay 06 \$ 3 \$ 5 \$ 2-D \$181.	<b>2-Day</b> \$169.25 \$84.62 \$46.56 \$23.28 <b>ay 3-</b> 14 \$12!	1race/ \$12 \$64 \$18 \$18 <b>Day</b> 9.19	3-Day 28.60 4.30 5.40 3.20 4-Day \$116.82	4-Day \$124.53 \$62.27 \$35.38 \$17.69 1-week \$111.88	5-Day \$120.47 \$60.23 \$34.37 \$17.18 <b>2-week</b> \$106.93	ETa \$75.00 \$90.00 \$120.0 \$144.2 <b>3-week</b> \$97.03	est(Lot) ) ) 0 20 4-weel \$87.14
Standa Qty 3 6 12 24 Toolin Custon Qty 3 6	ng *(Pricing same Day \$352.17 \$176.09 \$92.29 \$46.15 ng NRE = \$0. m Pricing Same Day \$309.76 \$156.26	15 based c       7     1-D       \$220.0     \$110.0       \$59.26     \$29.63       00     \$270.19       \$136.27	ay 96 \$ 13 \$ 3 \$ 3 \$ 5 \$ 5 \$ 5 \$ 5 \$ 5 \$ 5 \$ 5 \$ 5	2-Day           \$169.25           \$84.62           \$46.56           \$23.28           ay         3-           14         \$12'           9         \$65.5	1race/ \$12 \$64 \$18 \$18 <b>Day</b> 9.19 .06	3-Day 28.60 4.30 5.40 3.20 4-Day \$116.82 \$58.81	4-Day \$124.53 \$62.27 \$35.38 \$17.69 <b>1-week</b> \$111.88 \$56.31	5-Day \$120.47 \$60.23 \$34.37 \$17.18 \$17.18 \$106.93 \$53.81	ETa \$75.00 \$90.00 \$120.0 \$144.2 \$144.2 \$97.03 \$48.82	est(Lot) ) ) 0 20 4-weel \$87.14 \$43.82
Qty       3       6       12       24       Toolin       Qty       3       6       12       24       Toolin       Qty       3       6       12       12       24       Toolin       Qty       3       6       12	ng *(Pricing same Day \$352.17 \$176.09 \$46.15 ng NRE = \$0. Pricing Same Day \$309.76 \$156.26 \$79.51	15 based c 7 1-D \$220.0 \$110.0 \$59.26 \$29.63 00 1-Day \$270.19 \$136.27 \$69.31	ay 96 \$ 3 \$ 3 \$ 5 \$ 5 \$ 5 \$ 5 \$ 5 \$ 5 \$ 5 \$ 5	2-Day           \$169.25           \$84.62           \$46.56           \$23.28           ay         3-           14         \$12'           9         \$65.7           7         \$32:	1race/ \$12 \$62 \$30 \$18 <b>Day</b> 9.19 .06 .99	<b>3-Day</b> 28.60 4.30 5.40 3.20 <b>4-Day</b> \$116.82 \$58.81 \$29.81		5-Day           \$120.47           \$60.23           \$34.37           \$17.18           \$106.93           \$53.81           \$52.26	ETa \$75.00 \$90.00 \$120.0 \$144.2 \$144.2 \$97.03 \$48.82 \$24.71	est(Lot) ) ) ) ) 20  4-weel \$87.14 \$43.82 \$22.16
Qty           Standa           Qty           3           6           12           24           Toolin           Qty           3           6           12           24           Toolin           Qty           3           6           12           24	ng *(Pricing same Day \$352.17 \$176.09 \$92.29 \$46.15 ng NRE = \$0. m Pricing m Pricing \$309.76 \$156.26 \$79.51 \$41.13	Is based c           x           1-D           \$220.0           \$110.0           \$59.26           \$29.63           00           Image: state	ay 96 \$ 3 \$ 5 \$ 5 \$ 5 \$ 5 \$ 5 \$ 5 \$ 5 \$ 5	<b>2-Day</b> \$169.25 \$84.62 \$46.56 \$23.28 <b>ay 3-</b> 14 \$12! 9 \$65 7 \$32 1 \$16	11race/ 11race/ \$12 \$62 \$30 \$18 <b>Day</b> 9.19 .06 .99 .96	<b>3-Day</b> <b>3-Day</b> <b>28.60</b> <b>4.30</b> <b>5.40</b> <b>3.20</b> <b>4-Day</b> <b>\$116.82</b> <b>\$58.81</b> <b>\$29.81</b> <b>\$15.30</b>	4-Day \$124.53 \$62.27 \$35.38 \$17.69 <b>1-week</b> \$111.88 \$56.31 \$28.53 \$14.64	S-Day           \$120.47           \$60.23           \$34.37           \$17.18           \$53.81           \$53.81           \$52.266           \$13.98	ETi \$75.00 \$90.00 \$120.0 \$144.2 <b>3-week</b> \$97.03 \$48.82 \$24.71 \$12.66	est(Lot) ) ) ) 0 20 4-weel \$87.14 \$43.82 \$22.16 \$11.33

Fig. 232 Advanced circuits DFM, page 2

## 4.18.2 Oshpark

The PCB manufacturer Oshpark<sup>39</sup> is used to generate a quote for the two-layer PCB Click on <BROWSE FOR FILES>, indicated on Fig. 233, and select the project board file .brd (Fig. 234).



Fig. 233 Oshpark welcome screen

	niects k nch 2laver v2 k	- 40 Senar			
Consistent Character Chara					
		::== <b>▼</b>			
🛛 🔶 🛧 Favorites	Name	Date modified	Туре 🖍		
	pcb_2layer.b#9	9/15/2020 10:11 PM	B#9 File		
🛛 📃 Desktop	pcb_2layer.brd	9/23/2020 7:29 PM	BRD File		
	pcb_2layer.pro	9/16/2020 12:02 AM	PRO File		
	pcb_2layer.s#1	9/16/2020 12:05 AM	S#1 File		
	pcb_2layer.s#2	9/16/2020 12:00 AM	S#2 File		
	pcb_2layer.s#3	9/15/2020 11:50 PM	S#3 File		
	pcb_2layer.s#4	9/15/2020 11:07 PM	S#4 File		
	pcb_2layer.s#5	9/15/2020 5:33 PM	S#5 File		
	pcb_2layer.s#6	9/15/2020 5:33 PM	S#6 File <sub>≣</sub>		
	pcb_2layer.s#7	9/15/2020 5:30 PM	S#7 File		
	pcb_2layer.s#8	9/15/2020 5:29 PM	S#8 File		
	pcb_2layer.s#9	9/15/2020 5:27 PM	S#9 File		
	📕 pcb_2layer.sch	9/23/2020 7:29 PM	SCH File 👻		
	<		•		
pcb_2layer.brd Date r BRD File	nodified: 9/23/2020 7:29 PM Size: 13.3 KB	Date created: 9/23/2020 6:36 PM			

Fig. 234 Project board .brd file



Enter project name, details, and contact email information as in Fig 235.

Fig. 235 Oshpark instant quote

The following images (Fig. 236–244) are Oshpark's extracted layer information from the imported project. Review these images to make sure the board pairs with the EAGLE design.



# **Board Top**

This shows the final manufactured board as if you held it in your hand.

Your design should show gold copper, purple mask, white silk, black drills, and the board outline.

Internal cutouts are indicated by a black outline but are not filled in.

If the image here is entirely white, you'll want to find and fix any gaps in the board outline.

There should be no dimension or measurement ruler

Fig. 236 Oshpark board top



# **Board Bottom**

This shows the final manufactured board as if you held it in your hand.

Your design should show gold copper, purple mask, white silk, black drills, and the board outline.

Internal cutouts are indicated by a black outline but are not filled in.

If the image here is entirely white, you'll want to find and fix any gaps in the board outline.

There should be no dimension or measurement ruler

Fig. 237 Oshpark board bottom



Rendered from "Drills.xln"

# Drills

Drills should show up as white circles or dots on a purple background.

Drill files should be NC Drill or Excellon format files. Multiple files will be accepted, and merged into one for fabrication.

Drill hits that pass through copper will be plated. All other drill hits will be non-plated.

Most drill formats are detected and displayed as you'd see on the board. If your drills look incorrect, try exporting with INCH units and either No Zero Suppression or Leading Zero Suppression.

Drill slots and "oval" drills included as part of the drill file are supported. Most design tools do this when using the tool's native slot commands. Supported slots will appear on this preview. Note, it's possible to use this supported callout in an unsupported way. See our Cutouts and Slots page for details regarding unsupported applications of slot drill commands.

Drills sizes below our minimums will be increased to the minimum size. See the design rules or our drills help page. for additional details on drill specs. Additionally, the following are not supported.

- Overlapping drill hits
- Blind or buried vias

Fig. 238 Oshpark drills



# Top Silk Screen

We will ignore the portion of the silkscreen that extends beyond the board outline.

We will automatically remove any silkscreen that crosses drilled holes or exposed copper.

If a logo isn't showing up on this layer, try changing your design tool import settings to create that silk image with 400 DPI or less, or check out our Eagle-specific import-bmp script instructions.





## **Bottom Layer**

This layer should appear 'mirrored' as if you were looking down on it through the board from the top.

We will place copper everywhere we see gold color on this layer.

If you are using Altium Designer or Altium CircuitMaker, carefully examine the board to make sure there are no shorts from the mechanical layers being included on this layer. See here for more.

If you are using Eagle, be aware that airwires are not the same as routed traces. If there are no copper links between pads showing on this layer, please review your .brd file for airwires.

See our design tools pages for more.

Fig. 240 Oshpark bottom layer



# Top Solder Mask

Soldermask layers show us where to remove the purple solder resist. The gold-colored areas will be exposed on the final board, and purple areas will be covered.

If you submitted an empty file, we won't remove any mask so this entire side of the board will be covered in purple soldermask

To expose the entire board, submit this file with a single polygon that covers the entire board. We will remove all mask everywhere and expose all the copper and board substrate.

Fig. 241 Oshpark top solder mask



# Top Layer

We will place copper everywhere we see gold color on this layer.

If you are using Altium Designer or Altium CircuitMaker, carefully examine the board to make sure there are no shorts from the mechanical layers being included on this layer. See here for more.

If you are using Eagle, be aware that airwires are not the same as routed traces. If there are no copper links between pads showing on this layer, please review your .brd file for airwires.

See our design tools pages for more.

Fig. 242 Oshpark top layer



# **Board Outline**

The board outline should be a watertight purple outline showing at least the edge of the board with no gaps.

We will cut non-rectangular board shapes, but you will be billed for the smallest rectangle that will encompass the design.

As an example, a 2in diameter circle is billed the same as a 2in by 2in square.

Non-plated Board Cutouts can be represented on the board outline layer, with some limitations. Slots are unsupported when indicated on the board outline layer, but usually work. To make slots with full support, use Drill Slots on the drill layer.





# Bottom Solder Mask

This layer should appear 'mirrored' as if you were looking down on it through the board from the top.

Soldermask layers show us where to remove the purple solder resist. The gold-colored areas will be exposed on the final board, and purple areas will be covered.

If you submitted an empty file, we won't remove any mask so this entire side of the board will be covered in purple soldermask

To expose the entire board, submit this file with a single polygon that covers the entire board. We will remove all mask everywhere and expose all the copper and board substrate.

Fig. 244 Oshpark bottom solder mask

## 5. Reception and Testing of PCB

### 5.1 Reception of Oshpark Boards

The PCB manufacturer Oshpark fabricated the boards shown in Fig. 245. Upon physical inspection, the ordered PCB mirrors the eCAD design. Using a caliper to inspect the physical dimensions of the board, the board and hole sizes are in

agreement with the manufacturer's tolerances. In addition, the silkscreen graphics look decent, with no blur. The board does have sharp protruding material projecting from the edges. This is most likely left-over spurs from holding/cutting the PCB array; a file or sandpaper is used to clean up the board's edges. Using a multimeter, all traces are tested for proper connections and there is no error found on the board.



Fig. 245 Oshpark fabricated board

## 5.2 Assembling and Testing PCB Power

Utilizing readily available components, a prototype board is assembled, as shown in Fig. 246. The board is powered with an external power supply set to 3.70 V. The substitute linear regulator (STMicroelectronics, PN: L78L33ACZ-AP) outputs 2.825 V to the microcontroller. The blue LED illuminiates signifying power to the system.



Fig. 246 Testing power on prototype PCB

## 5.3 Extended Testing

In order to assess each LED, the board is programmed using Microchip's MPLAB X Integrated Development Environment (IDE) v5.4.0<sup>33</sup> and XC8 compiler v2.30.<sup>34</sup> A serial progrogramming kit (PICKit3) is connected from the laptop to the PCB.

Once the MPLAB X IDE and the XC8 compiler are installed on the virtual machine, open the software (Fig. 247).



Fig. 247 MPLAB X welcome screen

In the top left of the MPLAB window go to <File>, <New Project> (Fig. 248).

😒 New Project		×
Steps	Choose Project	
1. Choose Project 2	Q Filter:	
	Categories:	Projects: Standalone Project Existing MPLAB IDE v8 Project Prebuilt (Hex, Loadble Image) Project User Makefile Project Library Project Import START MPLAB Project Import Atmel Studio Project
	Description:	
	Creates a new standalone application proj project.	ect. It uses an IDE-generated makefile to build your
	< Back	Next > Finish Cancel Help

Fig. 248 Creating a standalone project

Select the drop-down menus in Fig. 249; Family: Mid-Range 8-bit MCUs (PIC10/12/16/MCP), Device: PIC12F1501, and Tool: PICkit3-SN:\*\*\*. The PICkit must be attached to the computer in order to be recognized by the program. Ensure the PICkit's pins are correctly paired with the PCB. Once finished, click the <Next> button.

teps	Select Device	2
Choose Project Select Device	Family	
Select Header Select Plugin Board Select Compiler	Family:	d-kange 8-bit MCUS (PIC 10/12/16/MCP)
Select Project Name and Folder	Device:	PIC12F1501 V
	Tool:	PICkit3-SN: Show All
MPLAB X IDE		

Fig. 249 Project creation

When prompted for a Supported Debug Header, select 'None' from the drop-down menu (Fig. 250). Afterward, click on <Next>.



Fig. 250 Supported debug header

Select the Cross Compiler Toolchain: XC8 (v2.30) [pathway to cross compiler] (Fig. 251). Afterward, click <Next>.

Vew Project		
teps	Select Compiler	
Choose Project     Select Device     Select Header     Select Plugin Board     Select Compiler     Select Project Name and     Folder	Compiler Toolchains in XC8 La <mark>XC8 (v2.30) [D: Microchip\Software\Compilers\vc8\230\bin]</mark>	
MPLAB X IDE		

Fig. 251 Cross compiler toolchain

Enter the project's name and define its location on the computer (Fig. 252).

🛞 New Project			×			
Steps	teps Select Project Name and Folder					
1. Choose Project 2. Select Device 3. Select Header	Project Name:	Demo				
<ol> <li>Select Plugin Board</li> <li>Select Compiler</li> </ol>	Project Location:	C:\Users\Kwashnak\Desktop\Demo Browse				
6. Select Project Name and Folder	Project Folder:	C: \Users\Kwashnak\Desktop\Demo				
	Overwrite existing p	project.				
	Also delete sources					
	Set as main project					
	Use project location	as the project folder				
	Encoding: ISO-4	8859-1				
		< Back Next > Finish Cancel Help				

Fig. 252 Project name and location

In the project tree, on the left, navigate to the 'Source Files' folder, right-click, mouse over to 'New', 'C Source File...'. Name the file "main.c" and select 'c' for the 'Extension' (Fig. 253). This file is our main code to program the microcontroller.



Fig. 253 Creating main.c source file

Once completed, paste the contents of code block 5 into the "main.c", which implements Microchip's microcontroller configuration parameters.<sup>40</sup>

```
/* Author:
                   Ken Kwashnak
   Contact (e):
                  ***@***.***
   Contact (p): (***)***-****
                October 8, 2020
   Date:
   Platform:
                Program intended for PIC12(L)F1501 for testing Demo v1.0 PCB functionality.
   Function:
                Upon power on, individual LEDs illuminate. If switch is pressed, LEDs turn off.
*/
// define crystal frequency
#define _XTAL_FREQ 16000000
#include <xc.h>
// Device Configuration, pg 38-39 of Microchip DS40001615C
// In order for the device to function properly, all registers must be configured.
// Clock settings
#pragma config CLKOUTEN = 1
// Brown-Out Reset
#pragma config BOREN = 00
// Code Protection
#pragma config CP = 1
// MCLR Pin Function
// since LVP is enabled, 1, this bit is ignored
#pragma config MCLRE = 0
// Power-Up Timer
#pragma config PWRTE = 1
// Watchdog timer
#pragma config WDTE = 00
// Oscillator settings
// using internal oscillator INTOSC
#pragma config FOSC = 00
// Low Voltage Programming
// based off of the voltage supplied to microcontroller
#pragma config LVP = 1
// Low-Power Brown-out Reset
#pragma config LPBOR = 1
// Brown-out Reset Voltage
#pragma config BORV = 1
// Stack Over/Underflow Reset
#pragma config STVREN = 0
// Flash Memory Self-Write
#pragma config WRT = 11
void init(){
// I/O, PORTA Settings
// No alternate pin functions
// RA2, RA4, RA5 = Output = LEDs
// RA3 = Input = Push Button
// Direction Control, 1 input, 0 output
  TRISA = 0x0B; // Binary: 0000 1011
// No Latch Register Definitions
// Analog Select Register
// Digital only
 ANSELA = 0;
// Weak Pull-Up
// grounds input push button with switch closure, prevents floating
// input will read a high state, until button pressed
// no debouncing method or circuity, not necessary
// Clearing register for individual pull-up use
  OPTION REGbits.nWPUEN = 0;
// R3 Pull-up
 WPUA = 0x08; // Binary: 0000 1000
}
void main(void){
// Frequency Configuration
  OSCCONbits.IRCF = 0xF; // 1111 = 16MHz
// Internal Clock Configuration
  OSCCONbits.SCS = 3;
// Initializes device, as per function above
  init();
// Infinite Loop, Stay in State
// PORTAbits = reads state
// LATA = write state
```

```
_delay_ms( time in milliseconds ); note, not best if you have interrupt service routines, ISRs
11
// the delay assists with the microcontroller performing the selected instruction
// ensures there's enough time to execute the next statement
// the loop essentially reads state of switch, if value is high, then write ON values to LEDs
// once the switch is pressed, input state goes low, and LEDs turn off
  while(1){
    if(PORTAbits.RA3){
      LATA = 0b000100; // RA2
        _delay_ms(100);
      LATA = 0b010000; // RA4
       _delay_ms(100);
      LATA = 0b100000; // RA5
        _delay_ms(100);
    }else{
      LATA = 0b110100;
    }
  }
}
```

Code block 5 PIC12F1501 code

Afterward, click on the <Clean and Build Project> button to compile the program, as in Fig 254.



Fig. 254 main.c code compilation

With no errors in the code, ensure the PCB is powered with 3.7 V and the PCB is hooked up to the laptop via the PICkit 3. Click on the <Make and Program Device> button, as shown in Fig 255.
S MPLAB X IDE v5.40 - Demo : default		
File Edit View Navigate Source Refactor Production Debug Team To	sels Window, Mar	Q · Search (Ctrl+I)
👚 🎦 🔛 🦷 🦻 🍘 🖉 🖅 😭	'월 - D ( ) · · · · · · · · · · · · · · · · · ·	
Projects x Files Classes Services	Make and Program Device Main Project	
E Demo		
B      B Header Files		
Important files		
Unker Files	2 +actine_AAA_FREQ 1000000	
Source Files	4 findlude carbo	
e la ubranes		
ii iii constants	6 #pragma config LVP = ON	
	7 // #pragma config LVP = OFF	
	5 #pragma config CLKOUTEN = OFF	
	9 #pragma config NDTE = OFF // or control in software - SWDTEN	
	10 #pragma config PWRTE = OFF	
	11 #pragma config CP = OFF	
	12 Pprogram config BOREN = OFF	
	14 fragme config FORCE - OFF	
	15	
	16 #preuma config STVREN = ON	
	17 #pragma config LPBOR = OFF	
	16 #pragma config BORV = LO	
	19 #pragma config WRT = ALL // we don't need self write	
	20 F void init()(	
Navigator Demo - Dashboard ×		
To Demo	22 EI // IO setup	
Call Device	24 // R22 R34 R35 - OUT - IVD	
PIC12F1901	25 / (Ral - IN - button	
🛖 🔤 Checksum: 0x8CA4	26 TRISA = 0x08: // 0000 1011	
- ES CRC32: 04632639167	27 ANSELA = 0; // all pins are digital	
PIC12-16E two DEP (1.2.63)	28 OPTION_REGbits.nNPUEN = 0; // enable weak pull-ups	
E-T Compiler Toolchain	29 WFUA = 0x08; // pull up on RAS	
- 👔 XC8 (v2.30) [D: \M crochip\Software\Compilers\vc8\230/bin]		
Production Image: Optimization: Disabled     Device connect information: DIC 13, 16E tool: DED (1, 2, 67)	Determine and	
Memory	output x	ini ini
Data 64 (0x40) bytes	ProjectLoading Warning × Demo (Build, Load,) × PICNIT3 ×	
6%		^
Data Used: 4 (0x4) Pree: 60 (0x0C)	Device Trased	
8%	Programming	
E-St Debug Tool	The following memory area(s) will be programmed:	
Prost Reference Dehum Tarvi Staturs	program memory: start address = 0x0, end address = 0x3ff	
E - 🐺 Debug Resources	Programming/Verify complete	
Program BP Used: 0 Pree: 0		
Data BP: No Support		
Unlimited BP (S/W): No Support		
		×
		(a) 1-1 INS

Fig. 255 PCB programming

## 6. Conclusion

Overall, the EAGLE design process is easy and straightforward. Creating resources may take some time, but it is necessary in order to ensure proper component functionality. Various manufacturers and designers have complete devices readily available for use—just check the license agreement and core design elements (symbol and package). A schematic is useful for assembling the design and essential for associating board components. Once the schematic is completed, a board file is manipulated to form a well-defined PCB (Fig. 256). The CAM feature allows generation of board information and Gerber files that a PCB manufacturer uses to make the eCAD board into a physical entity. After employing all steps described in this paper, a two-layer microcontroller PCB is fabricated, assembled, programmed, and tested.



Fig. 256 Completed functional microcontroller PCB

### 7. References

- Kester solder. Digikey Electronics; 2020 [accessed 2020 Oct 8]. https://www.digikey.com/en/products/detail/kester-solder/24-9574-7618 /3660017.
- 2. Aoyue Int 936. Aoyue International Ltd.; 2020. www.aoyue3d.com/en/pro/default.asp?id=35.
- Soldering flux. Digikey Electronics; 2020 [accessed 2020 Oct 8]. https://www.digikey.com/en/products/detail/mg-chemicals/8341-10ML /2233037?s=N4IgTCBcDaIBwGYAsBGAtCgDAWwDYgF0BfIA.
- KORAD KD3005D. SRA Soldering Products; 2013–2020 [accessed 2020 Oct 8]. https://sra-solder.com/korad-kd3005d-precision-variable-adjustable-30v-5a-dc-linear-power-supply-digital-regulated-lab-grade.
- 5. Test leads. Digikey Electronics; 2020 [accessed 2020 Oct 8]. https://www.digikey.com/en/products/detail/b-k-precision/TL-5A/272826.
- Fluke 117. Digikey Electronics; 2020 [accessed 2020 Oct 8]. https://www.digikey.com/en/products/detail/fluke-electronics/FLUKE-117/1506333.
- 7. Desolder braid. Digikey Electronics; 2020 [accessed 2020 Oct 8]. https://www.digikey.com/en/products/detail/chemtronics/60-5-5/306973.
- Assorted wire. Digikey Electronics; 2020 [accessed 2020 Oct 8]. https://www.digikey.com/en/products/detail/sparkfun-electronics/PRT-11375/5956252.
- Wire strippers. Digikey Electronics; 2020 [accessed 2020 Oct 8]. https://www.digikey.com/en/products/detail/american-hakko-productsinc/CSP-30-1/8346234.
- Tweezer Set. Digikey Electronics; 2020 [accessed 2020 Oct 8]. https://www.digikey.com/en/products/detail/american-hakko-productsinc/CSP-30-1/8346234.
- 11. Isopropyl alcohol. Digikey Electronics; 2020 [accessed 2020 Oct 8]. https://www.digikey.com/en/products/detail/mg-chemicals/824-1L/9181284.
- 12. Brush. Digikey Electronics; 2020 [accessed 2020 Oct 8]. https://www.digikey.com/en/products/detail/mg-chemicals/855-5/949490.

- PICKit3. Digikey Electronics; 2020 [accessed 2020 Oct 8]. https://www.digikey.com/en/products/detail/microchip-technology /PG164130/2171224.
- 14. Programming cable. Digikey Electronics; 2020 [accessed 2020 Oct 8]. https://www.digikey.com/en/products/detail/digilent-inc/240-035/4090162.
- 15. ESD caliper. Digikey Electronics; 2020. [accessed 2020 Oct 8]. https://www.digikey.com/en/products/detail/wiha/41105/1706539.
- PIC12LF1501-I/P. Digikey Electronics; c1995–2020 [accessed 2020 Sep 23]. https://www.digikey.com/en/products/detail/microchip-technology /PIC12LF1501-I-P/3046619.
- 17. ICS-308-T. Digikey Electronics; c1995–2020 [accessed 2020 Sep 23]. https://www.digikey.com/products/en?keywords=ics-308-T.
- 18. CR2450VP. Digikey Electronics; c1995–2020 [accessed 2020 Sep 23]. https://www.digikey.com/products/en?keywords=cr2450vp.
- 19. BS2450. Digikey Electronics; c1995–2020 [accessed 2020 Sep 23]. https://www.digikey.com/products/en?keywords=bs2450.
- MCP1700-3002E/TO. Digikey Electronics; c1995–2020 [accessed 2020 Sep 23]. https://www.digikey.com/products/en?keywords=mcp1700-3002E%2FTO.
- 21. 5988170107F. Digikey Electronics; c1995–2020 [accessed 2020 Sep 23]. https://www.digikey.com/product-detail/en/dialight/5988170107F/350-.
- PTS 647 SK38 SMTR2 LFS. Digikey Electronics; c1995–2020 [accessed 2020 Sep 23]. https://www.digikey.com/product-detail/en/c-k/PTS-647-SK38 -SMTR2-.
- CRCW0805240RFKEA. Digikey Electronics; c1995–2020 [accessed 2020 Sep 23]. https://www.digikey.com/en/products/detail/vishay-dale /CRCW0805120RJNEA/1175176.
- 24. C0805C104M3RACTU. Digikey Electronics; c1995–2020 [accessed 2020 Sep 23]. https://www.digikey.com/product-detail/en/kemet /C0805C104M3RACTU/399-.
- 25. 0022232051. Digikey Electronics; c1995–2020 [accessed 2020 Sep 23]. https://www.digikey.com/products/en?keywords=22-23-2051.
- 26. 0022012057. Digikey Electronics; c1995–2020 [accessed 2020 Sep 23]. https://www.digikey.com/products/en?keywords=0022012057.

- 27. 22-23-2021. Digikey Electronics; c1995–2020 [accessed 2020 Sep 23]. https://www.digikey.com/products/en?keywords=22232021.
- 28. 0022012027. Digikey Electronics; c1995–2020 [accessed 2020 Sep 23]. https://www.digikey.com/products/en?keywords=0022012027.
- 29. 00086508004. Digikey Electronics; c1995–2020 [accessed 2020 Sep 23]. https://www.digikey.com/product-detail/en/molex/0008650804/WM2756CT-.
- 30. EAGLE. Autodesk EAGLE; 2020 [accessed 2020 July 19]. www.autodesk.com/products/eagle.
- How to get started with a new Autodesk EAGLE license. Autodesk EAGLE;
   2016 [accessed 2019 July 31]. https://www.autodesk.com/products/eagle
   /blog/how-to-get-started-with-a-new-autodesk-eagle-license/..
- 32. Free download eagle. Autodesk EAGLE; 2016 [accessed 2019 July 31]. https://www.autodesk.com/products/eagle/free-download.
- Microchip. MPLAB X integrated development environment (IDE). Microchip Technology Inc.; 2020 [accessed 2020 Oct 8]. https://www.microchip.com/mplab/mplab-x-ide.
- MPLAB XC compilers. Microchip Technologies Inc.; 2020 [accessed 2020 Oct 8]. https://www.microchip.com/en-us/development-tools-tools-andsoftware/mplab-xc-compilers.
- 35. Eagle help, design rules. MIT; 2005 [accessed 2020 Aug 17]. www.mit.edu/xavid/arch/i386\_rhel4/help/127.htm.
- PCB Trace Width Calculator. Advanced Circuits; 2020 [accessed 2020 Sep 4]. www.4pcb.com/trace-width-calculator.html.
- 37. Your account. Advanced circuits; n.d. [accessed 2020 Sep 23]. https://www.my4pcb.com.
- Filename extension summary. Advanced circuits; 2020 [accessed 2020 Sep 23]. https://www.my4pcb.com/net35/FreeDFMQuoteSpecs.aspx.
- 39. Let's get started! Oshpark; 2020 [accessed 2020 Sep 23]. https://oshpark.com.
- 40. PIC12(L)F1501. Microchip Technology Inc.; c2011–2015 [accessed 2020 Oct 8]. https://ww1.microchip.com/downloads/en/DeviceDoc/40001615C.pdf. DS40001615C.

Appendix A. List of Manufacturing References

### A.1 PCB

- 1) Advanced Circuits
- 2) OshPark
- 3) Sierra Circuits
- 4) Sunstone Circuits
- 5) JLC PCB
- 6) Avanti Circuits

## A.2 Enclosure and Panel

- 1) Protocase
- 2) Lansing Enclosures
- 3) Phoenix Mecano
- 4) Front Panel Express

### A.3 Panel Overlay

- 1) Bradley Nameplate
- 2) Pannam Imaging
- 3) Design Mark

## A.4 Component References and Distributors

- 1) Octopart
- 2) Digikey
- 3) Mouser
- 4) Arrow

## A.5 List of eCAD Software Packages

- 1) KiCAD
- 2) Altium
- 3) pSpice
- 4) SolidWorks Electrical
- 5) Falstad Circuit Simulator Applet

Appendix B. Hypertext Markup Language (HTML)

Tables B-1 through B-3 provide descriptions of Hypertext Markup Language (HTML) formatting basic, formatting, and color code tags.<sup>1</sup> Other resources include a color mixer<sup>2</sup> and color picker<sup>3</sup> to assist with the design.

Tag	Description
	Defines the document type
<html></html>	Defines an HTML document
<head></head>	Contains metadata/information for the document
<title></title>	Defines a title for the document
<body></body>	Defines the document's body
<h1> to <h6></h6></h1>	Defines HTML headings
	Defines a paragraph
	Inserts a single line break
<hr/>	Defines a thematic change in the content
	Defines a comment

Table B-1 Basic HTML

Table B-2	HTML	formatting
-----------	------	------------

Tag	Description
<acronym></acronym>	Not supported in HTML5. Use <abbr> instead. Defines an acronym</abbr>
<abbr></abbr>	Defines an abbreviation or an acronym
<address></address>	Defines contact information for the author/owner of a document/article
<b></b>	Defines bold text
<bdi></bdi>	Isolates text that may be formatted in a different direction from other text outside it
<bdo></bdo>	Overrides the current text direction
<big></big>	Not supported in HTML5. Use CSS instead. Defines big text
<blockquote></blockquote>	Defines a section that is quoted from another source
<center></center>	Not supported in HTML5. Use CSS instead. Defines centered text
<cite></cite>	Defines the title of a work
<code></code>	Defines a piece of computer code
<del></del>	Defines text that has been deleted from a document
<dfn></dfn>	Specifies a term that is going to be defined within the content
<em></em>	Defines emphasized text
<font></font>	Not supported in HTML5. Use CSS instead. Defines font, color, and size for
	text
<i>&gt;</i>	Defines a part of text in an alternate voice or mood
<ins></ins>	Defines a text that has been inserted into a document
<kbd></kbd>	Defines keyboard input
<mark></mark>	Defines marked/highlighted text
<meter></meter>	Defines a scalar measurement within a known range (a gauge)
<pre></pre>	Defines preformatted text
<progress></progress>	Represents the progress of a task
<q></q>	Defines a short quotation
<rp></rp>	Defines what to show in browsers that do not support ruby annotations

<sup>&</sup>lt;sup>1</sup> HTML element reference. Refsnes Data; c1999–2020 [accessed 2020 Aug 20]. https://www.w3schools.com/TAGS/default.asp.

<sup>&</sup>lt;sup>2</sup> Color mixer. HTML color mixer. Refsnes data; c1999–2020 [accessed 2020 Aug 20].

https://www.w3schools.com/colors/colors mixer.asp?colorbottom=00FFFF&colortop=FFFFFF.

<sup>&</sup>lt;sup>3</sup> Color picker. Refsnes data; c1999–2020 [accessed 2020 Aug 20].

https://www.w3schools.com/colors/colors picker.asp?colorhex=F0F8FF.

Tag	Description
<rt></rt>	Defines an explanation/pronunciation of characters (for East Asian
	typography)
<ruby></ruby>	Defines a ruby annotation (for East Asian typography)
< <u>s</u> >	Defines text that is no longer correct
<samp></samp>	Defines sample output from a computer program
<small></small>	Defines smaller text
<strike></strike>	Not supported in HTML5. Use <del> or <s> instead. Defines strikethrough</s></del>
	text
<strong></strong>	Defines important text
<sub></sub>	Defines subscripted text
<sup></sup>	Defines superscripted text
<template></template>	Defines a container for content that should be hidden when the page loads
<time></time>	Defines a specific time (or datetime)
<tt></tt>	Not supported in HTML5. Use CSS instead. Defines teletype text
<u></u>	Defines some text that is unarticulated and styled differently from normal text
<var></var>	Defines a variable
<wbr/>	Defines a possible line-break

 Table B-3
 HTML color codes

<b>Color name</b>	HEX	RGB	Color name	HEX	RGB
AliceBlue	F0F8FF	240,248,255	LightSkyBlue	87CEFA	135,206,250
AntiqueWhite	FAEBD7	250,235,215	LightSlateGray	778899	119,136,153
Aqua	<b>00FFFF</b>	0,255,255	LightSteelBlue	B0C4DE	176,196,222
Aquamarine	7FFFD4	127,255,212	LightYellow	FFFFE0	255,255,224
Azure	F0FFFF	240,255,255	Lime	00FF00	0,255,0
Beige	F5F5DC	245,245,220	LimeGreen	32CD32	50,205,50
Bisque	FFE4C4	255,228,196	Linen	FAF0E6	250,240,230
Black	0	0,0,0	Magenta	FF00FF	255,0,255
BlanchedAlmond	FFEBCD	255,235,205	Maroon	800000	128,0,0
Blue	0000FF	0,0,255	MediumAquaMarine	66CDAA	102,205,170
BlueViolet	8A2BE2	138,43,226	MediumBlue	0000CD	0,0,205
Brown	A52A2A	165,42,42	MediumOrchid	BA55D3	186,85,211
BurlyWood	DEB887	222,184,135	MediumPurple	9370DB	147,112,219
CadetBlue	5F9EA0	95,158,160	MediumSeaGreen	3CB371	60,179,113
Chartreuse	7FFF00	127,255,0	MediumSlateBlue	7B68EE	123,104,238
Chocolate	D2691E	210,105,30	MediumSpringGreen	00FA9A	0,250,154
Coral	FF7F50	255,127,80	MediumTurquoise	48D1CC	72,209,204
CornflowerBlue	6495ED	100,149,237	MediumVioletRed	C71585	199,21,133
Cornsilk	FFF8DC	255,248,220	MidnightBlue	191970	25,25,112
Crimson	DC143C	220,20,60	MintCream	F5FFFA	245,255,250
Cyan	<b>00FFFF</b>	0,255,255	MistyRose	FFE4E1	255,228,225
DarkBlue	00008B	0,0,139	Moccasin	FFE4B5	255,228,181
DarkCyan	008B8B	0,139,139	NavajoWhite	FFDEAD	255,222,173
DarkGoldenRod	B8860B	184,134,11	Navy	80	0,0,128
DarkGray	A9A9A9	169,169,169	OldLace	FDF5E6	253,245,230
DarkGreen	6400	0,100,0	Olive	808000	128,128,0
DarkKhaki	BDB76B	189,183,107	OliveDrab	6B8E23	107,142,35

Color name	HEX	RGB	Color name	HEX	RGB
DarkMagenta	8B008B	139,0,139	Orange	FFA500	255,165,0
DarkOliveGreen	556B2F	85,107,47	OrangeRed	FF4500	255,69,0
DarkOrange	FF8C00	255,140,0	Orchid	DA70D6	218,112,214
DarkOrchid	9932CC	153,50,204	PaleGoldenRod	EEE8AA	238,232,170
DarkRed	8B0000	139,0,0	PaleGreen	98FB98	152,251,152
DarkSalmon	E9967A	233,150,122	PaleTurquoise	AFEEEE	175,238,238
DarkSeaGreen	8FBC8F	143,188,143	PaleVioletRed	DB7093	219,112,147
DarkSlateBlue	483D8B	72,61,139	PapayaWhip	FFEFD5	255,239,213
DarkSlateGray	2F4F4F	47,79,79	PeachPuff	FFDAB9	255,218,185
DarkTurquoise	00CED1	0,206,209	Peru	CD853F	205,133,63
DarkViolet	9400D3	148,0,211	Pink	FFC0CB	255,192,203
DeepPink	FF1493	255,20,147	Plum	DDA0DD	221,160,221
DeepSkyBlue	00BFFF	0,191,255	PowderBlue	B0E0E6	176,224,230
DimGray	696969	105,105,105	Purple	800080	128,0,128
DodgerBlue	1E90FF	30,144,255	RebeccaPurple	663399	102,51,153
FireBrick	B22222	178,34,34	Red	FF0000	255,0,0
FloralWhite	FFFAF0	255,250,240	RosyBrown	BC8F8F	188,143,143
ForestGreen	228B22	34,139,34	RoyalBlue	4.17E+04	65,105,225
Fuchsia	FF00FF	255,0,255	SaddleBrown	8B4513	139,69,19
Gainsboro	DCDCDC	220,220,220	Salmon	FA8072	250,128,114
GhostWhite	F8F8FF	248,248,255	SandyBrown	F4A460	244,164,96
Gold	FFD700	255,215,0	SeaGreen	2E8B57	46,139,87
GoldenRod	DAA520	218,165,32	SeaShell	FFF5EE	255,245,238
Gray	808080	128,128,128	Sienna	A0522D	160,82,45
Green	8000	0,128,0	Silver	C0C0C0	192,192,192
GreenYellow	ADFF2F	173,255,47	SkyBlue	87CEEB	135,206,235
HoneyDew	F0FFF0	240,255,240	SlateBlue	6A5ACD	106,90,205
HotPink	FF69B4	255,105,180	SlateGray	708090	112,128,144
IndianRed	CD5C5C	205,92,92	Snow	FFFAFA	255,250,250
Indigo	4B0082	75,0,130	SpringGreen	00FF7F	0,255,127
Ivory	FFFFF0	255,255,240	SteelBlue	4682B4	70,130,180
Khaki	F0E68C	240,230,140	Tan	D2B48C	210,180,140
Lavender	E6E6FA	230,230,250	Teal	8080	0,128,128
LavenderBlush	FFF0F5	255,240,245	Thistle	D8BFD8	216,191,216
LawnGreen	7CFC00	124,252,0	Tomato	FF6347	255,99,71
LemonChiffon	FFFACD	255,250,205	Turquoise	40E0D0	64,224,208
LightBlue	ADD8E6	173,216,230	Violet	EE82EE	238,130,238
LightCoral	F08080	240,128,128	Wheat	F5DEB3	245,222,179
LightCyan	E0FFFF	224,255,255	White	FFFFFF	255,255,255
LightGoldenRodYellow	FAFAD2	250,250,210	WhiteSmoke	F5F5F5	245,245,245
LightGray	D3D3D3	211,211,211	Yellow	FFFF00	255,255,0
LightGreen	90EE90	144,238,144	YellowGreen	9ACD32	154,205,50
LightPink	FFB6C1	255,182,193			
LightSalmon	FFA07A	255,160,122			
LightSeaGreen	20B2AA	32,178,170			

 Table B-3
 HTML color codes (continued)

Appendix C. Sample Library Device Hypertext Markup Language (HTML) Description

Description of MICROTEST-DIP_8		×		
Headline:	MICROTEST-DIP_8			
	Author: Ken Kwashnak			
	Date: August 20, 2020			
	This is a sample description file for a Library Device component.			
<b>1</b> Testing Paramet	ers			
Sample Text				
• A: This is an indented line item. T	his is <b>BOLD</b> text.			
• B: This is an indented line item. T	his is <i>ITALICIZED</i> text.			
• C: This is an indented line item. T	his is <u>UNDERLINED</u> text.			
• This is a double indented	line item. Here is a combination of <b>BOLD and ITALICIZED</b> text.			
• Sample <sub>3</sub> Text*				
2 List of Compone				
	1 Sample Description 1			
	2 Sample Description 2			
html		-		
<body></body>				
<title> MICROTEST-DIP_8 </title>				
<center><b>Author</b>: Ken Kwashna</center>	k			
<b>Contact</b> : ***@****.***				
<b>Date</b> : August 20, 2020 <td>er&gt;</td> <td>E</td>	er>	E		
$\langle n \rangle \langle center \rangle \langle i \rangle$ This is a sample desc	intion file for a Library Device component			
	and the rest of the second product of the strength states.			
<h1> 1 Testing Parameters </h1>				
Sample Text <ul> <li>A: This is an indented line iter</li></ul>	n. This is <b>BOLD</b> text.			
<ul> <li>B: This is an indented line iten</li></ul>	1. This is <i>ITALICIZED </i> text.			
<ul><li><ul><li><li><li><li><li><li><li><li><li><l< td=""><td>line item. Here is a combination of <b><i>BOLD and ITALICIZED</i></b></td><td></td></l<></li></li></li></li></li></li></li></li></li></ul></li></ul>	line item. Here is a combination of <b><i>BOLD and ITALICIZED</i></b>			
<ul><ul><ul><li>Sample<sub>3</sub>1</li><li></li></ul></ul></ul>	ext <sup>1</sup>			
<hr/> <h1>2 List of Components </h1>				
<table border="1" cellnadding="4" cellspace<="" td=""><td>ing="0" bacolor="#ffffff" width="200" align="center"&gt;</td><td></td></table>	ing="0" bacolor="#ffffff" width="200" align="center">			
<thread></thread>				
> <th <="" colspan="2" style="color:white" td=""><td>" bacolor=black&gt; Table Header</td><td></td></th>	<td>" bacolor=black&gt; Table Header</td> <td></td>		" bacolor=black> Table Header	
	adana mané unu neuro d'ané	-		
	OK Cancel Undo Red	)		

Fig. C-1 Hypertext Markup Language (HTML) example

```
<!DOCTYPE html>
<html>
<body>
<title> MICROTEST-DIP_8 </title>
<center><b>Author</b>: Ken Kwashnak
<br>
<b>Contact</b>: ***@***.***
<br>
<b>Date</b>: August 20, 2020 </center>
<center> <i> This is a sample description file for a Library Device component. </i>
</center> 
<h1> 1 Testing Parameters </h1>
Sample Text
A: This is an indented line item. This is <b>BOLD</b> text. 
 B: This is an indented line item. This is <i>ITALICIZED</i> text.
 C: This is an indented line item. This is <u>UNDERLINED</u> text.
This is a double indented line item. Here is a combination of <b><i>BOLD
and ITALICIZED</b></i> text. 
Sample<sub>3</sub> Text<sup>1</sup> 
<hr><h1>2 List of Components </h1> </hr>
<table border=1 cellpadding="4" cellspacing="0"
                                               bgcolor="#ffffff"
                                                                width="200"
align="center">
<thread>
 Table Header
</thread>
 <center>Item</center>
<center>Description</center>
 <center>1</center>
<center>Sample Description 1</center>
 <center>2</center>
<center>Sample Description 2</center>
<!--This is a comment-->
</body>
</html>
```

```
Code block 6 HTML code for sample library device description
```

Appendix D. Advanced Circuits Manufacturing Specifications

This appendix contains Advanced Circuits manufacturing specifications for the two-layer printed circuit board (PCB) listed in Table D-1.<sup>1</sup>

Rule	Value	Unit
Inner layer clearance	0.010	Inch
Copper to edge of PCB, outer layers	0.010	Inch
Copper to edge of PCB, inner layers	$0.015 - 0.020^{a}$	Inch
Scoring, outer layers	0.015	Inch
Scoring, inner layers	0.020	Inch
Pad size/annular ring	0.010 over finished hole size for vias	Inch
	0.014 over finished hole size for	
	components	
Hole size, tolerance	0.003-0.005	Inch
Hole size, minimum	0.004	Inch
PCB tolerance	10	%
Minimum	0.005	Inch
Board thickness		Inch
2-Layer	0.020	
4-Layer	0.020	
6-Layer	0.031	
8-Layer	0.047	
10-Layer	0.062	
Rout (board outline)	$0.005^{b}$ 0.010	Inch
Copper spacing <sup>c</sup>		Inch
1 oz finished copper weight (inner	0.003	
layers)	0.005	
2 oz finished copper weight (inner &	0.009	
outer)	0.010	
3 oz finished copper weight (inner &		
outer)		
4 oz finished copper weight (inner &		
outer)		
Copper trace width/air gap <sup>c</sup>	20	%
	0.002	Inch
Copper thickness, maximum	20	OZ
Slot width	Custom—0.031ª	Inch
Tab rout	0.100	Inch
Silkscreen (legend)	0.005 minimum	Inch
Bevel options	20	Degree
	30ª	
	45	
Acceptability standard	IPC A600 class 2 through IPC 6012	
	class 3A	
Underwriter's laboratory approval	Yes	
CAM software used	Frontline's genesis	

#### Table D-1 Advanced circuits manufacturing specifications

<sup>a</sup> preferred <sup>b</sup> special request

<sup>°</sup> premium charge for trace width/spacing less than 0.007 inch. Will process .004 in for 1 oz. CU. finished (outer layers) and 0.5 oz. CU finished (inner layers)

<sup>1</sup> Advanced circuits specs. Advanced circuits; 2020 [Accessed 2020 Oct 8]. https://www.4pcb.com/pcb-prototype-2-4-layer-boards-specials.html.

Appendix E. Oshpark Manufacturing Specifications

This appendix contains Oshpark manufacturing specifications for the two-layer printed circuit board (PCB).<sup>1</sup>

#### E.1 Two-Layer 2-oz 0.8-mm Service

The PCB is a special stackup intended for a variety of high-power and low-weight circuit designs. This service features a 0.032-inch (0.8-mm)-thick PCB, and includes 2-oz (2.8-mil, 70-um) copper.

#### E.2 Pricing

The pricing is \$5 per square inch, which includes three copies of your design. For example, a 2-square-inch board would cost \$10 and you would get three copies of your board. You can order as many copies as you want, as long as they are in multiples of three.

#### E.3 Turn Times

Orders are sent to fabrication every week and have a turnaround time from the fabrication of 1 week.

#### E.4 Fabrication Specs

#### E.4.1 Common Specs

Spec	Value		
Manufactured in the United States	Yes		
Lead Free compatible	Yes		
RoHS complaint	Yes		
High temp	Yes, 175 Tg or higher (see Material Specs)		
PCB finish	ENIG (Gold), compliant with IPC-4552		
Soldermask type	SMOBC (Soldermask over bare copper), both sides		
Silkscreen type	High res DLP, both sides		

<sup>&</sup>lt;sup>1</sup> Services. Oshpark; 2017. [accessed 2020 Oct 8]. https://docs.oshpark.com/services/two-layer-hhdc/.

# E.4.2 Stackup

Thickness	Layer	Tolerance
1 mil (0.0254 mm)	Silkscreen	+/-0.2 mil (0.00508 mm)
1 mil (0.0254 mm)	Solder resist	+/-0.2 mil (0.00508 mm)
2.8 mil (0.0711 mm)	2-oz copper	
30 mil (0.762 mm)	Core	+/-3 mil (0.0762 mm)
2.8 mil (0.0711 mm)	2-oz copper	
1 mil (0.0254 mm)	Solder resist	+/-0.2 mil (0.00508 mm)
1 mil (0.0254 mm)	Silkscreen	+/-0.2 mil (0.00508 mm)

# E.4.3 Material Specs

Spec	Value	
Substrate	175 Tg FR4	
Board thickness	32 mil (0.8 mm) nominal	
Dielectric	4.5 at 10 Mhz	
Soldermask color	Purple	
Minimum soldermask web	4 mil (0.1016 mm)	
Maximum soldermask alignment	3 mil (0.0762 mm)	
Silkscreen minimum line width	5 mil (0.127 mm) (recommended minimum) 3 mil (0.0762 mm) (short lines, text, graphics)	
Maximum board size	16 inch (406.4 mm) by 22 inch (558.8 mm)	
Minimum board size	0.25 inch (6.35 mm) by 0.25 inch (6.35 mm)	

# E.4.4 Copper Specifications

Spec	Value
Copper layers	2
Copper weight	2 oz
Trace spacing	6 mil (0.1524 mm)
Trace width	6 mil (0.1524 mm)
Annular ring	5 mil (0.127 mm)
Board edge keepout	15 mil (0.381) from nominal board edge
Via plating thickness	1 mil (0.0254 mm)

Spec	Value
Minimum annular ring	5 mil (0.127 mm)
Minimum drill size	10 mil (0.254 mm)
Minimum slot size	20 mil (0.508 mm) (drill slot only)
Drill size tolerance	Max: +/- 2.5 mil (0.0635 mm) Typical: +/- 1.0 mil (0.0254)
Drill positional tolerance	Max: 2 mil (0.0508 mm) Typical: <1 mil (0.0254 mm)
Via tenting	Yes (filled hole and flat surface not guaranteed)
Buried via	No
Blind via	No
Overlapping drills	Allowed, but not guaranteed. May result in missing or slotted holes. 5 mil (0.127 mm) clearance is recommended between holes.
Castellations	Allowed, but not guaranteed
Maximum drill size	None

# E.4.5 Drill Specifications

## **Additional Reading**

### Schematics

Jimblom. How to read a schematic. Sparkfun Electronics; n.d. [accessed 2020 Nov 23]. https://learn.sparkfun.com/tutorials/how-to-read-a-schematic/all.

### **Board Materials**

Peterson Z. PCB core versus prepreg material: what designers need to know. Altium; 2020 Jan 17 [accessed 2020 Aug 27]. https://resources.altium.com/p/pcb-core-vs-prepreg-material-what-designersneed-to-know.

## EAGLE Software Manual

CadSoft. EAGLE manual version 5. Pembroke Pines (FL): CadSoft Computer; 2010 [accessed 2020 Aug 31]. 7th Edition. Also available at www.hades.mech.northwestern.edu/images/b/b4/Eagle\_Manual.pdf.

# List of Symbols, Abbreviations, and Acronyms

3-D	three-dimensional
CAD	computer-aided design
CD	compact disc
COTS	commercial off the shelf
DIP	dual inline package
DRC	Design Review Check
EAGLE	easily applicable graphical layout editor
eCAD	electrical computer-aided design
eCAM	electrical computer-aided manufacturing
ESD	electro-static discharge
EULA	End User License Agreement
GND	ground
HTML	Hypertext Markup Language
IC	integrated circuit
IDE	Integrated Development Environment
LED	light-emitting diode
OS	operating system
РСВ	printed circuit board
PWM	pulse width modulation
SMD	surface mount device
ULP	user language program

1	DEFENSE TECHNICAL
(PDF)	INFORMATION CTR
	DTIC OCA

1	DEVCOM ARI
1	

- (PDF) FCDD RLD DCI TECH LIB
- 1 DEVCOM ARL
- (PDF) FCDD RLW PA
  - K KWASHNAK