# Oregon **TECH**

An printed circuit board designed to hold all of the components necessary to create an Arduino Uno using Mentor PADS software. This project included designing a usable schematic, and ensuring that the final PCB was error free and ready for submission to a fabrication house.

# Arduino Uno Printed Circuit Board

An Oregon Institute of Technology engineering project

Michael Hernandez

Arduino Uno Printed Circuit Board Project Oregon Institute of Technology EE 485: Printed Circuit Board Design Instructors: Jason Sullins and Allan Douglas Student: Michael Hernandez Summer 2019

# Table of Contents

Lab 1	6
Purpose	6
Part 1: Familiarization with the Arduino Uno	6
ATMEGA328 Microcontroller	6
Power Supply	7
F1231XS USB-to-Serial Converter	9
Part 2: Creating your PartQuest Accounts and Links for Download	
DigiKey – PartsQuest	
ODA Starter Library	
Part 3: Adding all Active Components and Connectors	
A special note about the redesign of the MIC5205	
Conclusion	
Lab 2	
Purpose	
Part 1: Completing the Schematic	
Part 2: Packaging the Schematics	
Part 3: Generate a Bill of Materials	
Lab 3	
Purpose	
Forward & Back Annotation	
Part 2. Define Electrical Levens and the DCD Outline	
Part 2: Define Electrical Layers and the PCB Outline	
Conclusion	
Lab 4	
Objectives	31
Redesigned Board	31
Part 1: Modify the Layer Stack Up and create planes	21 21
Part 1: Noully the Layer Stack-Op and create planes.	
Part 3: Signal Routing	
A note about routing for this project	
Part 4: Run PADs DRCs and Check Plane Laver Thermals.	
Conclusion	
Lab 5	

Introduction	42
Part 1: Renumber Reference Designators, Verify Silkscreen, and Back Annotate Schematics	42
Part 2: Generate and Review Gerber files.	43
Part 3: Generate NC Drill CAM document	45
Part 4: Add Board Outlines and Generate Gerber Files	46
Part 5: ODB++	
Conclusion	49
Appendix A	

# Table of Figures

Figure 1 - Schematic for the ATMEGA328 Chip	7
Figure 2 - Schematic for the power supply	9
Figure 3 - FT231XS USB-to-serial converter	. 10
Figure 4 - Account profile screen with Mentor Graphics	. 11
Figure 5 - Account profile screen with Digi-Key	. 11
Figure 6 - ATMEGA328 successfully downloaded	. 12
Figure 7 - ATMEGA328 placed onto the schematic	. 13
Figure 8 - Redesign of the ATMEGA328 component	. 14
Figure 9 - Redesign of the FT231XS component	. 15
Figure 10 - Redesign of the LM1117 component	. 15
Figure 11 - Redesign of the MIC5205 component	. 16
Figure 12 - Initial schematic once all of the modified components have been added	. 17
Figure 13 - Completed Schematic	. 19
Figure 14 - Screen to select which parameters to varify	. 20
Figure 15 - No errors in this schematic	. 20
Figure 16 - Project Integration Screen	. 24
Figure 17 - Dispersed Components	. 25
Figure 18 - Library Services	. 26
Figure 19 - Board Thickness	. 26
Figure 20 - Layer 1	. 27
Figure 21 - Layer 2	. 27
Figure 22 - Layer 3	. 27
Figure 23 - Layer 4	. 27
Figure 24 - 2X2 board layout with component decals	. 28
Figure 25 - Component pads have been placed to untangle the rats nets as much as possible	. 29
Figure 26 - Redesigned Layout	. 32
Figure 27 - Creating a copper plane	. 33
Figure 28 - Top view of the completed board	. 34
Figure 29 - Bottom view of the completed board	. 35
Figure 30 - Example demonstrating the effects of error checking being disabled. Vias could b	)e
laid on top of pads or traces, and traces could overlap	. 36
Figure 31 - A clearance report revealed a few errors, most of which were fixed except for error	ors
resulting from the IC pads being to close together which would need to be changed in one of t	the
document settings	. 37
Figure 32 - Connectivity report revealing no errors	. 37
Figure 33 - Fabrication report revealing no errors	. 37
Figure 34 - Results of the plane report	. 38
Figure 35 - Via connecting the net to the +3.3V plane	. 39
Figure 36 - Via connecting the +5.0V node to the power plane	. 40
Figure 37 - Via connecting the net to the DGND plane	. 40
Figure 38 - The eco toolbar is greyed out	. 42
Figure 39 - Name and number on top layer	. 42

Figure 40 - Back annotation is disabled	43
Figure 41 - Top Layer	44
Figure 42 - Bottom Layer	44
Figure 43 - DGND Layer	44
Figure 44 - Power Plane Layer	44
Figure 45 - Pastemask Top Layer	45
Figure 46 - Silkscreen Top Layer	45
Figure 47 - NC Drill Drawing	46
Figure 48 - Board Outlines	47
Figure 49 - Zipped gerber files; click on the image to open the files.	47
Figure 50 - ODB++ Error	48
Figure 51 - ODB++ 11 Viewer	49

## Lab 1

## Purpose

This lab exercise is designed to gain familiarity with the schematic and layout of the Sparkfun Redboard variant of the Arduino Uno as well as to learn the basics of how to create a new project using the various pieces of software that will be used throughout the duration of this project. Specifically, the software that will be covered in this lab are as follows:

- Learning how to create a new project in DxDesigner
- Learning how to import components with PartQuest
- Learning how to modify imported schematic symbols

### Part 1: Familiarization with the Arduino Uno

The schematic to be used for the duration of this project is to be downloaded from the Sparkfun website. Please see Appendix A for a copy of the schematic. Part 1 will analyze all three portions of the schematic. The ATMEGA328 microcontroller, the power supply, and the FT231XS USB to serial converter.

#### ATMEGA328 Microcontroller

The ATmega328 is a single-chip microcontroller created by Atmel in the megaAVR family. It has a modified Harvard architecture 8-bit RISC processor core. The pin used in the Sparkfun Redboard variant uses a 32-pin chip. The datasheet for this chip can be found at: <u>http://ww1.microchip.com/downloads/en/DeviceDoc/ATmega48A-PA-88A-PA-168A-PA-328-P-DS-DS40002061A.pdf</u>.

The crystal oscillator is necessary for clock operations in the microcontroller; it feeds into the clock multiplexer along with four other components. The low power crystal oscillator is not capable of driving other clock inputs, however the full swing crystal oscillator is. The crystal oscillator is connected to the XTAL 1 and XTAL 2 pins on the ATMEGA328 microcontroller. The full swing oscillator has rail-to-rail swing at the XTAL 2 output pin. Either a quartz crystal or a ceramic resonator may be used for the crystal oscillator.

Test points are for checking against a ground reference to see if the test point (usually on a power node) has been shorted out.

The In-Circuit Serial Programming (ICSP) connector allow for in-circuit serial programming. In other words, the data is moved into a microcontroller via serial communication; the microcontroller then executes these instructions. In the ATMEGA328 board there are six pins – four pins that are used for data transfer, a 5V VCC input, and a ground pin. In this microcontroller, the MOSI is the master output/slave input pin, the MISO pin is the master

input/slave output pin, , and the SCK pin is the master clock output/slave clock output pin. These three pins are associated with registers DDB 3/4/5 respectively.

An LED is connected to the microcontroller so that the microcontroller can be visually inspected for shorts. If the LED is on then the Arduino has power and is functioning properly. The resistor connected in series with the LED is to reduce the input voltage to the LED, thereby protecting the LED from burning out.



Figure 1 - Schematic for the ATMEGA328 Chip

#### Power Supply

The LM1117 IC3 is a current limiting device which limits the current on the microcontroller and all pins to 150mA regardless of how many pins are in use. The datasheet for the LM1117 can be found at <u>http://www.ti.com/lit/ds/symlink/lm1117.pdf</u>.

D1 is an MBRA140 Schottky power rectifier whose function is to act as a low voltage, high frequency switch. The datasheet for the MBRA140 can be found at <a href="https://www.onsemi.com/pub/Collateral/MBRA140T3-D.PDF">https://www.onsemi.com/pub/Collateral/MBRA140T3-D.PDF</a>.

LED2 is connected to 5 volts as a means of visually checking to determine if the device is powered on.

U2 is an MIC5205 150 mA Low-Noise LDO Regulator. Its primary purpose is to maintain a 150mA output current in order to power the FT231 USB-to-Serial Converter. Two capacitors, C6 and C7 are required on the output pin of U2 to prevent current oscillation. The datasheet can be found at <u>http://ww1.microchip.com/downloads/en/DeviceDoc/20005785A.pdf</u>.

Q1 is a Schottky diode connected in parallel with a MOSFET transistor. An example of this is the MAX17521 60V, 1A, Dual-Output, High-Efficiency, Synchronous Step-Down DC-DC Converter. The MAX17521 uses peak-current-mode control. Each output can be operated in the pulse-width modulation (PWM) or pulse-frequency modulation (PFM) control schemes. The datasheet for MAX17521 can be found at https://datasheets.maximintegrated.com/en/ds/MAX17521.pdf.

## Power Supply



Figure 2 - Schematic for the power supply

#### FT231XS USB-to-Serial Converter

U1 is the FT231XS package developed by Future Technology Devices International. It is a USB to serial UART interface. It is a convenient chip because the entire USB protocol handled on the chip so no USB specific firmware programming required. The datasheet for the FT231X is available at <a href="https://www.ftdichip.com/Support/Documents/DataSheets/ICs/DS\_FT231X.pdf">https://www.ftdichip.com/Support/Documents/DataSheets/ICs/DS\_FT231X.pdf</a>.

F1 is a resettable fuse with a maximum voltage of 6V, and a maximum current of 500mA. The function of the fuse is to protect the USB port. One such fuse is the 0805 SMT fuse from Bourns. The datasheet is available at <u>https://www.sunrom.com/get/285900</u>.

Q2 is the 2N7002PW 60 V, 310 mA N-channel Trench MOSFET. The purpose of Q2 is to act as a high-speed switch between the FT231XS and the ATMEGA328 reset pin. The datasheet is available at

https://static6.arrow.com/aropdfconversion/81cb195590ad52e7f88c5301252a6a7779066d32/178 2814141267262n7002pw.pdf.

Resistors 8 and 9 are to lower the voltage between the transmit and reset pins on the FT231XS. The resistors have values of  $81K\Omega$  and  $91K\Omega$  respectively, causing a substantial voltage drop prior to entering the chip.

The reset portion of the ATMEGA328 contains a switch which pulls the voltage low from the 5-volt VCC. An inverting gate negates the input signal prior to entering the reset pin. The C8 capacitor converts the level-triggered signal DTR into an edge-triggered signal and has the effect of level shifting it to within the operating voltage range of the MCU. Data Terminal Ready (DTR) is a signal line used for hardware flow-control in serial protocols. The DTR signal coming from the FT231XS leaves the chip as an inverted signal and goes through the Q2 switch prior to entering the ATMEGA328. The 3.3V and 5V differential voltages on either side of the 2N7002PW act as voltage triggers; the addition of a voltage signal from the FT231XS chip to the 3.3V side of the 2N7002PW pushes the current forward into the ATMEGA328.



## FT231XS (USB-to-Serial Converter)

Figure 3 - FT231XS USB-to-serial converter

## Part 2: Creating your PartQuest Accounts and Links for Download

This section of the lab is designed to introduce the student to the PartQuest utility from DigiKey, as well as to create a starter library that will be used in subsequent labs.

#### DigiKey - PartsQuest

For this portion of the lab accounts with Mentor Graphics and DigiKey are to be created. After accounts are successfully, the Atmel ATMEGA328P-AUR component from PartsQuest is to be downloaded.

$\leftarrow \  \   \rightarrow \  \   \mathbf{G} \  \   \mathbf{\nabla}$	https://www.mentor.com/profile/							7	☆ 💁 🛛	፣ 📬 📀	B 43	🗖 🚱 🛪   🔕 🗄
			RODUCTS & SOLUTIONS	TRAINING	SERVICES	COMPANY		SUPPORT				Michael Hernandez
											٩	
	Profile											
	MY PROFILE Dashboard Edit Profile	Michae Profile Infor	Hernande:	z			Edit	Newslet	tters			
	Notifications Training Newsletter	Full Name	Michael Hernandez	Username	michael.	nernandez@	pit.edu	You are to any n	not curren newsletters.	ly subscrib	ed	
	Subscriptions Purchase History	Company	Oregon Institute of Technology	Job Function	Student				Add New	sietters		
		Phone	-	Address Postal Code	97070			On-Den You have	nand Train	<b>ng</b> mand train	ing	
		Country	UNITED STATES					subscrip	tions right	now. -Demand		
		Notification	s			١	iew All		Cour	ses		

Figure 4 - Account profile screen with Mentor Graphics

- orginary corporation [05] [ https://www	All Products *	
PRODUCTS	MANUFACTURERS RESOURCES TOOLS	Hello Michael Hernandez 🎽 0 ITEM(S) 🗸
MyDigi-Key		
My Profile	Profile Information	
Welcome Michael Hernandez Customer Number Pending Edit Profile   Add Profile	Login Information Usemame: michaelhern	Display Name: Michael Hermandez
My Ordering and Shopping Carts Eavorites	Password: Change Password	Edit Email: michael.hemandez@oit.edu Edit
My Tools	Shipping Address and Contact Information	
My Billing		
Attach Existing Open Account		
My Digi-Key Team		
Digi-Key Sales 1-800-344-4539 OR 218-681-6674 Email		
		■ Live Chat
Stay Connected!		f ¥ 🖪 Ø in
Enter your email	Subscribe	Deventeed on the     Off of the
Information United	itates	Copyright © 1995-2019, Digi-Key Electronics.
Blog - TheCircuit Toll Fre	Igikey.com Africa : 1-800-344-4539 Asia	All Rights Reserved. 701 Brooks Avenue South, Third Divers Faile, All 65/201118A

Figure 5 - Account profile screen with Digi-Key

PARTQUEST	Atmel ATMEGA	A328P-AUR	ALL CATEGORIES -	Search			Community Welcome Michae	Feedback	t Hel ndez
	RoHS Compliant	nt 🔳 Lead Free 🔳 I	n Stock						
and the second	Showing 1-3 of 3 pa	art(s)		Select Parts: Select Al	I Select None Bate	ch Actions: <b>Favorite</b>	Add to Project	+ Down	nload
			Manufacturer Part Number	🌣 Manufacturer 🚽	Description	Digi-Key Part 🌣 Number	Unit Price (USD)	×	
ly Parts -	1	Lead Free RoHS Comp	Liant ATMEGA328P-AUR	Microchip Technology	IC MCU 8BIT 32KB FLASH 32TQFP	ATMEGA328P-AURTR-ND	1.782	٨	м
y Projects +	÷ •	Lead Free RoHS Comp	Uant ATMEGA328P-AUR	Microchip Technology	IC MCU 8BIT 32KB FLASH 32TQFP	ATMEGA328P-AURCT-ND	2.140	人	м
ilters • o filter your search results ou must select a <u>sub-</u> t <b>tegory</b> .	÷ •	Lead Free RoHS Comp	Liant ATMEGA328P-AUR	Microchip Technology	IC MCU 8BIT 32KB FLASH 32TQFP	ATMEGA328P-AURDKR- ND	2.140	A	м
				Page 1	1 of 1				
bout   Help   Terms of Use   Pr	ivacy Policy   Downloa	ads				© Mentor Graphi	1-800-547-3000   Conta cs, All rights reserved. Version:	Men ct Mentor Gr 2.12873127.	raphics 81533

Figure 6 - ATMEGA328 successfully downloaded

#### ODA Starter Library

For this portion of the lab, the ODA Starter Library for PADS Standard Plus is to be downloaded for later use.

Once the library has been downloaded, and components successfully imported into PADS, the ATMEGA328 IC is to be placed on the schematic as seen below.



Figure 7 - ATMEGA328 placed onto the schematic

## Part 3: Adding all Active Components and Connectors

Once part 2 is completed, part three may be started. The following components are to be downloaded from Partquest and imported into PADS: IC3, U1, U2, Q1, Q2, micro USB, J2, JP10, JP12, IOL, and AD. These components should reflect the components found on the schematic in Appendix A.

Once all components are successfully imported into PADS they need to be modified to match the layout of the Redboard schematic.

In order to correctly reflect the schematic that is being used as a guide for this project, several components needed to be redesigned using the component editor tool in PADS. The end result can be seen in the following figures.

<sup>29</sup> PC6_(RESET/PCINT14)	PC0_(ADC0/PCINT8) PC1_(ADC1/PCINT9) PC2_(ADC2/PCINT10) 25
18 4 VCC_1 VCC_2	PC3_(ADC3/PCINT11) 26 PC4_(ADC4/SDA/PCINT12) 27 PC5_(ADC5/SCL/PCINT13) 28 ADC6 19 ADC6 22
20AREF	PD0_(RXD/PCINT16) PD1_(TXD/PCINT17) PD2_(INT0/PCINT18) PD3_(PCINT19)0C238/INT1)
7 PB6_(PCINT6/XTAL1/TOSC1) 8 PB7_(PCINT7/XTAL2/TOSC2)	PD4_(PCINT20/XCK/T0) 2 PD5_(PCINT21/OC0B/T1) 9 PD6_(PCINT22/OC0A/AIN0) 10 PD7_(PCINT23/AIN1) 11
21 GND_3 3 GND_1 GND_2	PB0_(PCINT0/CLKO/ICP1) PB1_(PCINT1/OC1A) PB2_(PCINT2/SS/OC1B) PB3_(PCINT3/OC2A/MOSI) PB4_(PCINT4/MISO) PB5_(SCK/PCINT5) 12 13 14 15 16 17

Figure 8 - Redesign of the ATMEGA328 component



Figure 9 - Redesign of the FT231XS component

3			
	ADJ	'GND	

Figure 10 - Redesign of the LM1117 component



Figure 11 - Redesign of the MIC5205 component



Figure 12 - Initial schematic once all of the modified components have been added

#### A special note about the redesign of the MIC5205

When the MIC5205 was downloaded from Partquest and imported, an error message appeared in PADS stating that the component already existed. Therefore, it was not possible to import the MIC5205 into PADS. Several names were given as alternative components to be used. Many of them had minor variations that would change the functionality of the component, for example an inverting gate coming out of pin 4. Also, all of the pins needed to be relabeled from A, B, etc. to be In, En, and Out. The component that was chosen to be replaced was the SN74AHC1G08DCKR.

## Conclusion

This lab introduced the concepts of using Partquest to find parts not natively in PADS. The use of PADS software for schematic design, and how to use PADS to modify existing schematic components. An existing schematic for an Arduino Uno was analyzed in detail, particularly the active components that will be used for creating the device. This project will attempt to faithfully recreate the Sparkfun Arduino Uno schematic for purposes of PCB design.

## Lab 2

#### Purpose

The purpose of this lab is to complete the schematic in PADS in order to get it to a point where it is possible to begin routing. A bill of materials will also be generated.

#### **Part 1: Completing the Schematic**

The remainder of the schematic started in lab 1 is to be completed to match the schematic in appendix A. Table 1 outlines the components to be used in the schematic.

SparkFun Redboard Components	Alternatives
ATMEGA328P	Import from PartQuest, then modify the symbol
J2 Connector	Use ODA 15-91-0060
JP10 Connector	Use ODA 15-91-0080
AD Connector	Use ODA 15-91-0060
IOL Connector	Use ODA 15-91-0080
JP2 Connector	Use ODA 15-91-6102
Power Connector	Use ODA 55932-0210
LM1117 Regulator	Use ODA LM1117MP-ADJ/NOPB
U2 Regulator	Use ODA LM2937IMP-3.3/NOPB
Q1 P-Channel FET	Use ODA ZXM61P03F
Q2 N-Channel FET	Use ODA 2N7002
FT231X	Use SOP63R5P600_20_D865N decal, create custom symbol
USB Connector	Use ODA 54819-0572
Reset Button	Use ODA EVQ-P2602W
16MHz Crystal	Use ODA ATS160SM-T and two CAP-0402-NPO-22pF-50V-5TOL
500mA Fuse	Use ODA 0460_500
47pF Capacitor	Use ODA CAP-0603-NPO-47pF-50V-5TOL
0.1uF Capacitor	Use ODA CAP-0603-X7R-100nF-16V-10TOL
10uF Capacitor	Import from PartQuest, DIGIKEY_478_8193_2_ND
1uF Capacitor	Use ODA CAP-0603-X5R-1uF-6V3-10TOL
Green LED	Use ODA HLMP-6500-F0011
Red LED	Use ODA HLMP-6000-E0011
Yellow LED	Use ODA HLMP-6400-F0011
Blue LED	Use a green LED instead

Table 1 - Components to be used in the schematic

The schematic that was created closely represents the SparkFun schematic. A few minor changes were included. A custom logo was inserted into the schematic. The USB-to-Serial converter portion was broken up into two different portions, the USB and the FT321XS portions. Each net was labeled with a tag manually drawn.

The resulting schematic can be seen in figure 13.



Figure 13 - Completed Schematic

## **Part 2: Packaging the Schematics**

The packager application is to be ran to ensure that there are no errors or warnings.

🗓 Verify			×
Settings Interconnectivity (12) Migration Connectivity Electrical Hierarchy Integrity Power&Ground Device Specific HDL Checks Links	Context Board: Board1 Path: \ Schematic: Schematic1 Sheet: 1 Scope Board Schematic Schematic Scheet All hierarchy undemeath Level Property: STD UHDL Verilog Report Show hierarchical paths Configuration Files c:/mentorgraphics'padsvx.2.5\sdd_home\standard\VerifyDefaults.ini C:/PADS Projects\DxProjects\Project3\Verify.ini		
	Defaults   > OK 0	Cancel	Help

Figure 14 - Screen to select which parameters to varify

Output	<b>→</b> ‡ ×
Running Drc : Started vdrc -proj "C:\PADS Projects\DxProjects\Project3\Project3.prj" -set "C:/PADS Projects\DxProjects\Project3\Verify.ini" -def "c:/mentorgraphics\padsvx.2.5\sdd_home\standard\VerifyDefaults.ini" -dxd_std -board "Board1"	
0 Error(s), 0 Warning(s), 0 Note(s), Finished vdrc	
Cutput & Errors & Packager & DRC & Properties	

Figure 15 - No errors in this schematic

## Part 3: Generate a Bill of Materials

A bill of materials needs to be generated. The output file will be in the form of an excel spreadsheet, although it can be saved as a .txt file as well. The bill of materials is generated

#	QTY	Part Number		Ref
	-			Designator
1	9	CAP-0603-X7R-100NF-16V-10TOL	.1u	C1-C9
2	1	CAP-0603-X5R-1UF-6V3-10TOL	1u	C10
3	3	TAJT106K010	10u	C11,C13,C14
4	1	TAJT106K010	47u	C12
5	2	CAP-0603-NPO-47PF-50V-5TOL	47p	C15,C16
6	2	HLMP-6500-F0011	GREEN	CR1,CR2
7	1	HLMP-6400-F0011	YELLOW	CR3
8	1	HLMP-6000-E0011	RED	CR4
9	1	MBR0530		D1
10	1	0460_500		F1
11	1	DIGIKEY_ATMEGA328P_AURTR_ND		IC1
12	1	DIGIKEY_768_1129_2_ND		IC2
13	2	15-91-0060		J1,J2
14	2	15-91-0080		J3,J4
15	2	15-91-0080	IOL	J3,J4
16	1	15-91-6102		J5
17	1	54819-0572		JZ
18	1	ZXM61P03F		Q1
19	1	2N7002		Q2
20	3	RES-0402-10K-1TOL	10K	R1-R3
21	5	RES-0402-1K-1TOL	1K	R4-R8
22	1	RES-0402-10K7-1TOL		R9
23	1	RES-0402-240K-1TOL	240	R10
24	1	RES-0402-715R-1TOL	715	R11
25	1	RES-0603-4K7-1TOL	4.7K	R12
26	2	RES-0402-27R-5TOL	27	R13,R14
27	1	EVQ-P2602W		S1
28	6	TP104-01-00		TP1-TP6
29	1	LM2937IMP-3_3/NOPB		U1
30	1	LM1117MP-ADJ/NOPB		U2
31	1	ATS160SM-T	16MHz	Y1

under tools  $\rightarrow$  part lister. The parameters that are to be listed as well as the format of the output file can be configured in the window that appears.

Table 2 - Bill of Materials

## Conclusion

The schematic is created by selecting components, usually imported from a source such as partquest, and connecting them in an easy to read format. It is important to select the

components that will be used to create the PCB board as those will be listed in the resulting bill of materials. Once all components of the schematic are connected appropriately and laid out in an easy to read manner, it is then necessary to run a check to ensure that no errors were made in designing the schematic.

## Lab 3

## Purpose

This lab has three objectives. The schematic components and design netlist need to be integrated onto the board. The electrical layers need to be defined as well as the outline of the PCB. Finally, all of the components need to be placed onto the board for routing.

## Part 1: PADS Layout and Integration

Under setup  $\rightarrow$  project integration, the screen shown in Figure 16 - Project Integration Screen will appear. Clicking on the yellow button should solve any warnings that appear. When all of the buttons turn green, the components will appear on the layout. Once the components have been placed on the layout, they are to be spread out so that the rats nest can be seen. This can be easily done using the disperse components feature located under tools  $\rightarrow$  disperse components. The resulting dispersed layout can be seen in Figure 17 - Dispersed Components.

#### Forward & Back Annotation

Forward annotation is the process of updating a layout to reflect changes made in the schematic. It is to be used whenever a change is made to the schematic and the change needs to be reflected in the layout.

Back annotation is the process of updating the schematic to reflect changes made to the layout. It is to be used whenever a change is made to the layout and the change needs to be reflected in the schematic.

F Project Integration	×					
Project file and Board						
C:\PADS Projects\DxProjects\Project3\Project3.prj	Browse					
Board Board1	Library					
Schematic connectivity & constraint status						
Request desired action by pressing amber or red buttons						
Forward Annotation Required, connectivity changed	<u>View report</u>					
No pending schematic constraints changes to load int	to PCB					
No pending PCB constraint changes to load into PCB						
Back Annotation disabled	<u>View report</u>					
Back Annotation options						
Disable commands that create connectivity changes						
✓ Preserve PCB design rules on first Forward Annotation						
Library extraction options						
Only extract missing library data						
O Update local libraries with newer Central Library data						
O Rebuild local library data; preserve locally built data						
O Delete local data; then rebuild library data						
Close	<u>H</u> elp					

Figure 16 - Project Integration Screen



Figure 17 - Dispersed Components.

#### Errors preventing forward annotation

If a component is missing the decal needed for the layout diagram, an error will appear which states "ERROR: Can not make new PCB item. iCDB counterpart identifier: [two nine digit numbers will appear here]". Following this, each missing pin will be identified. In order to solve this, the decal must be added to the component within the parts manager library. First open the parts manager library. Then add the .edx file for the component with the missing decal. Third, add the component when the component is added, the screen in Figure 18 - Library Serviceswill appear. Select yes and click through all of the prompts that appear. At this point the decal can now be added to the library without any issues for the component.

CIDICITY SCIV	ces		×
Part 'DIC the sour Do you	IKEY_768_1129_2_ND' is ce but contains elements the wish to continue?	older in the ta nat are newer i	rget library than in n the target library.
Press th	e log file button for details.		
Overwrite all	affected older items withou	t additional pro	mpting

Figure 18 - Library Services

## Part 2: Define Electrical Layers and the PCB Outline

This board is to have four layers. Layer 1 is to be for placing components, layer 2 is to be the ground plane, layer 3 is for the +3.3V and +5 volt nets, and layer 4 is for routing. All unused layers are to be disabled.

Next the thickness is to be adjusted so that there is 1 Oz of copper on each layer. The thickness of the two middle layers is to be adjusted so that the board is 62 mils thick.

Name	Туре	Thickness	Dielectric	ОК
	Coating	0	3.3	Cancel
Тор	Component	1.00	1	ouncer
	Substrate	10	4.3	Help
DGND	Plane	1.00	4.3	
	Substrate	23.3	4.3	<u>E</u> dit
POWER_PLANE	Plane	1.00	4.3	Board
	Substrate	23.3	4.3	Thickness:
Bottom	Routing	1.00	1	62 mil
	Coating	0	3.3	

Figure 19 - Board Thickness

🔢 Layers Setup	×	🔢 Layers Setup	$\times$
Lev Type Dir Name		Lev. Type Dir Name	
1 CM H Top	01		01/
2 PL V Inner Layer 2	OK	2 PL V Inner Layer 2	UK
3 RX H Inner Layer 3 4 RT V Bottom	Cancel	3 RX H Inner Layer 3 4 RT V Bottom	ancel
	<u>H</u> elp	н	lelp
<u>N</u> ame: Top		Name: Inner Layer 2	
Electrical Layer Type		Electrical Layer Type	
Component     Routing	Associations	O Component Routing	
Plane Type Routing Direction	_	Plane Type Routing Direction	1
No Plane     O Horizontal     45		No Plane Horizontal 45	
CAM Plane Vertical 0-45		CAM Plane     Vertical     -45	
O Split/Mixed O Any			_
O shuttinger		Assig	n Nets
Electrical Layers		Electrical Layers	
Single-sided <u>b</u> oard support		Single-sided board support	
Count: 4 of 20 Modi <u>fy</u> <u>R</u> eassign	Thickness	Count: 4 of 20 Modify Reassign Thickne	ess
Nonelectrical Layers		Nonelectrical Layers	
Count: 0 of 26 Enable/Disable	Max Lavers	County 0 of 26 Enable/Disable	avera
	Max <u>c</u> ayers		ayers
Figure 20 - Layer 1		Figure 21 - Layer 2	
🔐 Layers Setup	×	🔀 Layers Setup	×
Layers Setup Lev. Type Dir. Name	×	Layers Setup Lev. Type Dir. Name	×
Layers Setup Lev. Type Dir. Name	К	Layers Setup Lev. Type Dir. Name	Х
Layers Setup Lev. Type Dir. Name 1 CM H Top 2 PL V Inner Layer 2	К	Layers Setup Lev. Type Dir. Name 1 CM H Top 2 PL V Inner Layer 2 3 RX H Inper Layer 3	Х
Layers Setup Lev. Type Dir. Name 1 CM H Top 2 PL V Inner Layer 2 3 RX H Inner Layer 3 4 RT V Bottom	Cancel	Lev. Type Dir. Name          1       CM       H       Top       C         2       PL       V       Inner Layer 2       C         3       RX       H       Inner Layer 3       C         4       RT       V       Bottom       C	X DK ncel
Layers Setup Lev. Type Dir. Name 1 CM H Top 2 PL V Inner Layer 2 3 RX H Inner Layer 3 4 RT V Bottom	OK Cancel	Layers Setup Lev. Type Dir. Name 1 CM H Top 2 PL V Inner Layer 2 3 RX H Inner Layer 3 4 RT V Bottom	X DK ncel
Layers Setup Lev. Type Dir. Name 1 CM H Top 2 PL V Inner Layer 2 3 RX H Inner Layer 3 4 RT V Bottom	Х ОК Cancel <u>H</u> elp	Layers Setup Lev. Type Dir. Name 1 CM H Top 2 PL V Inner Layer 2 3 RX H Inner Layer 3 4 RT V Bottom H	X DK ncel
Layers Setup Lev. Type Dir. Name 1 CM H Top 2 PL V Inner Layer 2 3 RX H Inner Layer 3 4 RT V Bottom	Х Ок Cancel <u>H</u> elp	Layers Setup Lev. Type Dir. Name 1 CM H Top 2 PL V Inner Layer 2 3 RX H Inner Layer 3 4 RT V Bottom H	X DK ncel
Layers Setup Lev. Type Dir. Name 1 CM H Top 2 PL V Inner Layer 2 3 RX H Inner Layer 3 4 RT V Bottom	Х ОК Cancel <u>H</u> elp	Layers Setup Lev. Type Dir. Name 1 CM H Top 2 PL V Inner Layer 2 3 RX H Inner Layer 3 4 RT V Bottom	X DK ncel
Layers Setup Lev. Type Dir. Name          1       CM       H       Top         2       PL       V       Inner Layer 2         3       RX       H       Inner Layer 3         4       RT       V       Bottom	Х ОК Cancel <u>H</u> elp	Layers Setup Lev. Type Dir. Name          1       CM       H       Top         2       PL       V       Inner Layer 2         3       RX       H       Inner Layer 3         4       RT       V       Bottom         Name:       Bottom	X DK ncel
Layers Setup Lev. Type Dir. Name          Lev. Type Dir. Name         1       CM         1       CM         2       PL         3       RX         1       Inner Layer 2         3       RX         4       RT         V       Bottom	Х ОК Cancel <u>H</u> elp	Layers Setup Lev. Type Dir. Name          Lev. Type Dir. Name         1 CM H Top         2 PL V Inner Layer 2         3 RX H Inner Layer 3         4 RT V Bottom         Name:         Bottom         Electrical Layer Type	× ncel
Layers Setup Lev. Type Dir. Name 1 CM H Top 2 PL V Inner Layer 2 3 RX H Inner Layer 3 4 RT V Bottom Name: Inner Layer 3 Electrical Layer Type Component Routing	Х ОК Cancel <u>H</u> elp	Layers Setup Lev. Type Dir. Name          Lev. Type Dir. Name         1 CM H Top         2 PL V Inner Layer 2         3 RX H Inner Layer 3         4 RT V Bottom         Mame:         Bottom         Electrical Layer Type         Component         © Rougting	× ncel
Layers Setup Lev. Type Dir. Name          1       CM       H       Top         2       PL       V       Inner Layer 2         3       RX       H       Inner Layer 3         4       RT       V       Bottom         Name:       Inner Layer 3         Electrical Layer Type       © Component       © Routing         Plane Type	× ОК Cancel <u>H</u> elp	Layers Setup Lev. Type Dir. Name          Lev. Type Dir. Name         1 CM H Top         2 PL V Inner Layer 2         3 RX H Inner Layer 3         4 RT V Bottom         Cal         Human         Electrical Layer Type         Component         © Component         © Routing         Plane Type	× ncel
Layers Setup Lev. Type Dir. Name          1       CM       H       Top         2       PL       V       Inner Layer 2         3       RX       H       Inner Layer 3         4       RT       V       Bottom         Name: Inner Layer 3         4       RT       V       Bottom         Electrical Layer Type         Component       Routing Direction         Plane Type       Routing Direction       045	× ОК Cancel <u>H</u> elp	Layers Setup Lev. Type Dir. Name          Lev. Type Dir. Name         1 CM H Top         2 PL V Inner Layer 2         3 RX H Inner Layer 3         4 RT V Bottom         Mame:         Bottom         Electrical Layer Type         Component         © Component         Plane Type         No Plane         O Horizontal         4 Nor Plane	× DK ncel
Layers Setup Lev. Type Dir. Name          1       CM       H       Top         2       PL       V       Inner Layer 2         3       RX       H       Inner Layer 3         4       RT       V       Bottom         Name: Inner Layer 3         Electrical Layer Type         Component       Routing         Plane Type       Routing Direction         O No Plane       Horizontal       45         O KM Blazo       45	× ОК Cancel <u>H</u> elp	Layers Setup Lev. Type Dir. Name          1       CM       H       Top         2       PL       V       Inner Layer 2         3       RX       H       Inner Layer 3         4       RT       V       Bottom         Mame:       Bottom       Call         Human       Electrical Layer Type       Component       Routing Direction         Plane       Horizontal       45         ©       CAM Plane       Vertical       -45	× ncel
Layers Setup Lev. Type Dir. Name          1       CM       H       Top         2       PL       V       Inner Layer 2         3       RX       H       Inner Layer 3         4       RT       V       Bottom         Name:       Inner Layer 3         Electrical Layer Type       © Component       © Routing         Plane Type       © Horizontal       0.45         © CAM Plane       © Horizontal       0.45         © Satit/Mined       © Arrow       Arrow	× ОК Cancel Help	Layers Setup Lev. Type Dir. Name          1       CM       H       Top         2       PL       V       Inner Layer 2         3       RX       H       Inner Layer 3         4       RT       V       Bottom         Vame:       Bottom       Ca         Bottom       Electrical Layer Type       Electrical Layer Type         O Component       Routing Direction         Image:       No Plane       Horizontal         Solit/Mixed       Anv	× ncel elp
Layers Setup Lev. Type Dir. Name          1       CM       H       Top         2       PL       V       Inner Layer 2         3       RX       H       Inner Layer 3         4       RT       V       Bottom         Name: Inner Layer 3         Electrical Layer Type         Component       Routing         Plane Type       Routing Direction         No Plane       Horizontal       45         CAM Plane       Any	× ОК Cancel Help	Layers Setup Lev. Type Dir. Name          1       CM       H       Top         2       PL       V       Inner Layer 2         3       RX       H       Inner Layer 3         4       RT       V       Bottom         Name:       Bottom       Call         Hame:       Component       Routing         Plane Type       Ocomponent       Horizontal         No Plane       Oyertical       45         O Split/Mixed       Any	× ncel elp
Layers Setup Lev. Type Dir. Name          1       CM       H       Top         2       PL       V       Inner Layer 2         3       RX       H       Inner Layer 3         4       RT       V       Bottom         Name:         Inner Layer 3         Electrical Layer Type         Component       Routing Direction         Plane Type       Routing Direction         No Plane       Horizontal       45         CAM Plane       Any         Electrical Layers       Any	Х ОК Cancel Help	Layers Setup Lev. Type Dir. Name          1       CM       H       Top         2       PL       V       Inner Layer 2         3       RX       H       Inner Layer 3         4       RT       V       Bottom         Name:       Bottom       Ca         Electrical Layer Type       Component       Routing Direction         Plane Type       Routing Direction       Horizontal       45         O CAM Plane       Yertical       245         O Split/Mixed       Any       Any	× ncel elp
Layers Setup Lev. Type Dir. Name          1       CM       H       Top         2       PL       V       Inner Layer 2         3       RX       H       Inner Layer 3         4       RT       V       Bottom         Name:       Inner Layer 3         Electrical Layer Type       © component       © Routing         Plane Type       Routing Direction         No Plane       © Horizontal       45         © CAM Plane       © Vertical       -45         © Split/Mixed       Any       Electrical Layers         Single-sided board support       Single-sided board support	Х ОК Cancel Help	Layers Setup Lev. Type Dir. Name          1       CM       H       Top         2       PL       V       Inner Layer 2       3         3       RX       H       Inner Layer 3       Ca         4       RT       V       Bottom       Ca         Name:       Bottom       Electrical Layer Type       Electrical Layer Type       Routing Direction         Image:       No Plane       O CaM Plane       Horizontal       45         O CAM Plane       O Split/Mixed       Any         Electrical Layers       Single-sided board support	× ncel elp
Layers Setup Lev. Type Dir. Name          1       CM       H       Top         2       PL       V       Inner Layer 2         3       RX       H       Inner Layer 3         4       RT       V       Bottom         Name:       Inner Layer 3         Electrical Layer Type       © component       © Routing Direction         Plane Type       Routing Direction       @ Horizontal       45         © CAM Plane       © Herical       _245         © Split/Mixed       Any       Electrical Layers         Single-sided board support       Single-sided board support	Х ОК Сапсе! Help	Layers Setup Lev. Type Dir. Name          1       CM       H       Top         2       PL       V       Inner Layer 2       Ca         3       RX       H       Inner Layer 3       Ca         4       RT       V       Bottom       Ca         Name:       Bottom       Electrical Layer Type       Component       Routing Direction         Image:       No Plane       O CaM Plane       Horizontal       45         O CAM Plane       O Split/Mixed       Any         Electrical Layers       Single-sided board support	× ncel elp
Layers Setup Lev. Type Dir. Name          1       CM       H       Top         2       PL       V       Inner Layer 2         3       RX       H       Inner Layer 3         4       RT       V       Bottom         Name:       Inner Layer 3         Electrical Layer Type       © component       © Routing Direction         Plane Type       Routing Direction       45         O CAM Plane       © Horizontal       45         O CAM Plane       O Any       Electrical Layers         Single-sided board support       Count: 4 of 20       Modify	× ОК Сапсе! Неlp Assign Nets	Layers Setup Lev. Type Dir. Name          1       CM       H       Top         2       PL       V       Inner Layer 2       Ca         3       RX       H       Inner Layer 3       Ca         4       RT       V       Bottom       Ca         Name:       Bottom       Electrical Layer Type       Component       Routing Direction         Plane       Ogomponent       Routing Direction       45       CAM Plane         O Split/Mixed       Any       Any       Electrical Layers         Single-sided board support       Count: 4 of 20       Modify       Reassign       Thicknee	× ncel elp
Layers Setup Lev. Type Dir. Name          1       CM       H       Top         2       PL       V       Inner Layer 2         3       RX       H       Inner Layer 3         4       RT       V       Bottom         Name:       Inner Layer 3         Electrical Layer Type       © Component       © Routing Direction         Plane Type       Routing Direction       45         O CAM Plane       © Horizontal       45         O Split/Mixed       Any       Electrical Layers         Single-sided board support       Count: 4 of 20       Modify	X OK Cancel Help Assign Nets	Layers Setup Lev. Type Dir. Name          1       CM       H       Top         2       PL       V       Inner Layer 2       Ca         3       RX       H       Inner Layer 3       Ca         4       RT       V       Bottom       Ca         Name:       Bottom       Electrical Layer Type       H         © Component       © Routing Direction       H         Plane       O gamponent       Horizontal       45         © CAM Plane       O Yertical       245         © Split/Mixed       Any       Electrical Layers         Single-sided board support       Count: 4 of 20       Modify       Reassign	× ncel elp
Layers Setup Lev. Type Dir. Name          1       CM       H       Top         2       PL       V       Inner Layer 2         3       RX       H       Inner Layer 3         4       RT       V       Bottom         Name:       Inner Layer 3         Electrical Layer Type       © component       © Routing Direction         Plane Type       Routing Direction       45         O CAM Plane       © Horizontal       45         O CAM Plane       O Any       Electrical Layers         Single-sided board support       Count: 4 of 20       Modify         Nonelectrical Layers       Nonelectrical Layers	X OK Cancel Help Assign Nets Thickness	Layers Setup Lev. Type Dir. Name          1       CM       H       Top         2       PL       V       Inner Layer 2       Ca         3       RX       H       Inner Layer 3       Ca         4       RT       V       Bottom       Ca         Name:       Bottom       Electrical Layer Type       Gomponent       Routing Direction         Plane       Ogmponent       Routing Direction       45       Gottom         O CAM Plane       Oyertical       245       Gattom       145         Single-sided board support       Count: 4 of 20       Modify       Reassign       Thicknee	× ncel elp
Layers Setup Lev. Type Dir. Name          1       CM       H       Top         2       PL       V       Inner Layer 2         3       RX       H       Inner Layer 3         4       RT       V       Bottom         Name:       Inner Layer 3         Electrical Layer Type       Component       Routing         Plane Type       Routing Direction       45         O CAM Plane       Horizontal       45         O CAM Plane       Any       Electrical Layers         Single-sided board support       Count: 4 of 20       Modify         Nonelectrical Layers       Count: 0 of 26       Enable/Disable	X OK Cancel Help Assign Nets Thickness	Layers Setup Lev. Type Dir. Name          1       CM       H       Top         2       PL       V       Inner Layer 2       Ca         3       RX       H       Inner Layer 3       Ca         4       RT       V       Bottom       Ca         Name:       Bottom       Electrical Layer Type       Component       Routing Direction         Plane Type       Routing Direction       H       H         © No Plane       Horizontal       45         © CAM Plane       Vertical       2-45         Single-sided board support       Count: 4 of 20       Modify       Reassign         Nonelectrical Layers       Count: 0 of 26       Enable/Disable       Max La	× OK ncel elp ess
Layers Setup         Lev. Type Dir. Name         1 CM H Top         2 PL V Inner Layer 2         3 RX H Inner Layer 3         4 RT V Bottom         Name:         [Inner Layer 3]         Electrical Layer Type         Component         No Plane         O CAM Plane         O Split/Mixed         Any         Electrical Layers         Single-sided board support         Count: 4 of 20         Modify         Reassign         Nonelectrical Layers         Count: 0 of 26	OK         Cancel         Help         Assign Nets         Thickness         Max Layers	Layers Setup Lev. Type Dir. Name          1       CM       H       Top         2       PL       V       Inner Layer 2       Ca         3       RX       H       Inner Layer 3       Ca         4       RT       V       Bottom       Ca         Name:       Bottom       Electrical Layer Type       Component       Routing Direction         Plane       Ogmponent       Routing Direction       45       CA         O CAM Plane       Oyertical       245       Single-sided board support         Count:       4 of 20       Modify       Reassign       Thicknee	× DK ncel elp ess ayers

Once the planes have been defined, the outline needs to be set. A 2x2 outline is to be created for this board to reflect the Redboard Arduino Uno.



Figure 24 - 2X2 board layout with component decals

## Part 3: Placement of all Electrical Components

For part 3, all capacitors and resistors are to be flipped to the bottom side. To do this, the components are to be selected. Once they are selected, they can be flipped to the other side by right clicking and clicking "Flip Side", or by hitting Ctrl + F.

Once the components are switched to their appropriate sides, they will need to be positioned on the board. First all of the active components are to be placed on the board. The connectors should be placed on the edges of the board. Then the active components need to be placed somewhere in the middle of the board. Finally, the passive components such as resistors and capacitors need to be placed. During placement, the components need to be placed in such a way as to try to make routing in the next step easier.

On the board, components on the top layer appear in blue, while components on the bottom layer appear in red.



Figure 25 - Component pads have been placed to untangle the rats nets as much as possible.

### Conclusion

Placing the layout requires that forward annotation work correctly. If any decals are missing from parts imported from partquest then this issue will need to be solved prior to laying out components. Once forward annotation is finished, all components will be imported into PADS Projects. The next step is to define the layers of the board and set the board to the desired thickness. Once this is completed, the physical outline of the board needs to be laid out. Finally

the components need to be placed within the outlined area in a manner that minimizes tangling of the rats nest for later routing.

## Lab 4

## Objectives

The purpose of this lab is mainly to route all of the components on the PCB. Additionally, this lab is designed to give practice changing views to verify plane connections, and to run design rule checks to ensure that the design is complete.

#### Redesigned Board

The layout of the components needed to be redone in order to properly route all of the components later on.



Figure 26 - Redesigned Layout

## Part 1: Modify the Layer Stack-Up and create planes.

Layer 3 is to be changed to be a split/mixed plane; this is accomplished under the layer definition screen. Separate plane areas are to be created for both the +5.0V and the +3.3V sections.

Once this has been done, a copper plane is to be poured on the DGND layer, as well as the power plane for the +3.3V and +5.0V areas. To do this, select the board outline, then select "select

shape", right click the board outline again, and select "create copper plane" a screen should appear as shown in Figure 27. The net assignment is to be set to the desired node.

ট Add Drafting		_			
Type; Copper Plane			/		
Line settings					
<u>W</u> idth: 10	Style:		/		
Shape settings					
Scale factor; 1	Arc appro <u>x</u> im	ation error: 0.5			
Rotation: 0.000	Solid cop	per			
Copper to trace	10		Flood & Hatch Options		
Layer:					
POWER_PLANES			~		
Net assignment					
To assign a net, select a ne	t in the list and	l click Apply			
None		~			
or Assign Net by Click	and select of	pject in the design.	<b>)</b>		
Show plane nets only	Show plane nets only				
Copper bridge. Select th	e nets to bridg	e			
Keepout Restrictions					
Placement		T <u>r</u> ace and cop	per		
<u>C</u> omponent height		Copper plane			
Component <u>d</u> rill		Via and jumpe	er		
		<u>T</u> est point			
Select <u>A</u> ll		Accordions			
	ОК	App <u>ly</u> Can	cel <u>H</u> elp		

Figure 27 - Creating a copper plane

## Part 2: Power and Ground Routing

Vias are to be used to connect the DGND, +3.3V, and +5.0V planes to components that necessitate them. Additionally, the pour manager is to be used to flood the planes with copper.

Due to issues with the program's ability to connect with Mentor's servers, this step was not able to be completed.

## **Part 3: Signal Routing**

All of the required traces are to be connected according to the rat's nest. Vias are to be used to connect the top and bottom sides of the board. Right angles are to be avoided whenever possible.



Figure 28 - Top view of the completed board



Figure 29 - Bottom view of the completed board.

#### A note about routing for this project

Technical difficulties were encountered starting when the routing was approximately halfway completed. The issue appears to be that PADS is unable to connect to Mentor's servers. PADS layout could still be used; however, PADS designer was unable to open the schematic for this board. Once this bug was encountered error checking as well as dynamic routing were disabled, and traces and vias could be laid on top of each other or over the top of pads (see Figure 30, below). Due to this error, the traces were laid using "eyeball" approximations. Because certain features were disabled, when components needed to be moved, several right angles in the traces were unavoidable.



Figure 30 - Example demonstrating the effects of error checking being disabled. Vias could be laid on top of pads or traces, and traces could overlap.

## Part 4: Run PADs DRCs and Check Plane Layer Thermals.

The design of the PCB is to be verified using the verify design tool in PADS. A clearance check, connectivity check, plane check, and fabrication check are to be ran to ensure completion of the board. During this step a number of routes and vias needed to be moved in order to have the appropriate clearences.



Figure 31 - A clearance report revealed a few errors, most of which were fixed except for errors resulting from the IC pads being to close together which would need to be changed in one of the document settings.



Figure 32 - Connectivity report revealing no errors



Figure 33 - Fabrication report revealing no errors

```
therm.err - Notepad
                                                                                     X
                                                                                -----
File Edit Format View Help
THERMAL RELIEF ERRORS REPORT -- Board1.pcb -- Fri Aug 09 12:42:08 2019
      Drilled pads with
                                           Nondrilled pads with
less than 50% thermal extensions less than 50% thermal extensions
Report of Thermal Spokes Generator.
On POWER_PLANES:
        (18705, 17990) \# = 0
On DGND:
        (18720, 18645) \# = 0
        (19080, 18530) \# = 0
        (18745, 18110) \# = 0
        (18985, 17295) \# = 0
        (19295, 18085) \# = 0
        (18760, 17285) \# = 0
        (18230, 18665) \# = 0
Total Drilled pads: 8 Total Nondrilled pads:
                                                                0
```

Figure 34 - Results of the plane report



*Figure 35 - Via connecting the net to the +3.3V plane* 



*Figure 36 - Via connecting the +5.0V node to the power plane* 



Figure 37 - Via connecting the net to the DGND plane

## Conclusion

The layout of the board was completed in lab 3, but needed to be changed in order to make routing feasible using only two layers while staying within the 2"X2" parameters for the board. Vias needed to be added to the board in order to connect the correct nodes to their respective power and ground planes. About halfway through the project the program encountered a problem where it was unable to connect to Mentor's servers, and therefore the schematic could no longer be referenced for the duration of routing. During this time certain features such as dynamic routing and smoothing were disabled, increasing the difficulty of the completing the project.

Lab 5

## Introduction

The purpose of this lab is to put the finishing touches on the PCB and to get it ready to be sent over to a fabrication plant. This lab will generate gerber files which may be used to have a board fabricated.

## Part 1: Renumber Reference Designators, Verify Silkscreen, and Back Annotate Schematics.

The cell size in the eco tool bar is to be renumbered to create a 1" x 1" cell size. This step cannot be completed however due to the eco toolbar being greyed out.



Figure 38 - The eco toolbar is greyed out

The designer's name, student ID number, course number, and term/year are to be added onto the silk screen layer on the primary side of the board.



Figure 39 - Name and number on top layer

Due to errors connecting to Mentor's servers, back annotation is not possible for this project.

JP Project Integration	×			
Project file and Board				
	Browse			
Board	Library			
Schematic connectivity & constraint status				
Design not migrated to PADS Designer flow.				
No connectivity changes to be forward annotated	View report			
No pending schematic constraints changes to load in	to PCB			
No pending PCB constraint changes to load into PCB				
No pending PCB changes to be back annotated	View report			
Manual schematic updates required. In PADS Design Tools menu > ECO > Import for Output window instru-	er, click ructions.			
Back Annotation options				
Disable commands that create connectivity changes				
Preserve PCB design rules on first Forward Annotation				
Library extraction options				
Only extract missing library data				
O Update local libraries with newer Central Library data				
C Rebuild local library data; preserve locally built data				
O Delete local data; then rebuild library data				
Close	<u>H</u> elp			

Figure 40 - Back annotation is disabled

## Part 2: Generate and Review Gerber files.

Gerber files can be generated under file > CAM. A screenshot for each layer is to be taken.



Figure 43 - DGND Layer

Figure 44 - Power Plane Layer



## Part 3: Generate NC Drill CAM document

The NC Drill file is to be generated, along with a drill drawing.

The drawing can be seen in Figure 47. A few of the drill holes are on the edge of the board.



Figure 47 - NC Drill Drawing

## Part 4: Add Board Outlines and Generate Gerber Files

Board outlines are to be added by using the menu as seen in the figure below. Once the gerber files are created, they are to be zipped into a single file for submission to a board fabrication house, such as Sunstone circuits (<u>https://www.sunstone.com/</u>).

Layer Selections A <u>v</u> ailable: DGND POWER_PLANES Bottom Layer_5 Layer_6 Layer_7	<ul> <li>▲dd &gt;&gt;</li> <li>&lt;&lt;<u>R</u>emove</li> </ul>	Selected:	OK Cancel <u>P</u> review
Other	Items on Primary		<u>H</u> elp
🗹 Board Outline	✓ Pads	Ref. Des.	
	✓ Traces	Part Type	
Plated Slots	✓ 2D Lines	✓ Text	
Non-plated Slots	Vias	Attributes	
Component outlines	Copper	Outlines	
Top Mounted	Keepouts	Test Points	
Bottom Mounted	Pins with Associated	Copper ion	
	Pads	🗸 Open Copper	
Color by Net		Filled Copper	
Selected Color			

Figure 48 - Board Outlines



# Project Arduino.zip

Figure 49 - Zipped gerber files; click on the image to open the files.

## Part 5: ODB++

ODB++ files are to be exported for the PCB design.

An error was encountered which prevented the export of an ODB++ type document (see below), however an ASCII type file was able to be exported.

```
    Board1_odbppExportrep - Notepad - □ ×
    File Edit Format View Help
    ODB++ export report file.
    Export to file: C:\PADS Projects\DxProjectT\Board1.tgz
    Report created on: 08/14/2019 22:33:32
    Error: Design translation failed!
```

Figure 50 - ODB++ Error

Had the ODB++ file been exported, it would have been viewed using another product from Mentor called ODB++ Viewer 11, which can be seen in Figure 51. Had the file been able to be exported it would have been able to be referenced with the viewer by the directory that the file was saved to.



Figure 51 - ODB++ 11 Viewer

## Conclusion

Text and images can be added to the silkscreen in order to make the board easier to work with later on after it has been fabricated. Gerber files where created so that the information for the board can be submitted to the fabrication house. There are a number of files that need to be prepared, including the top and bottom layers, the soldermask, the silkscreen, the pastemask, the ground plane and power planes. When the files are prepared, they must then be zipped together in order to be submitted. ODB++ is the latest standard which contains all of the information for the board in a single file.

## Appendix A



Arduino Uno schematic

Downloaded from https://www.sparkfun.com/products/12757