ECE401
Part II: Getting Started with Fusion 360
PCB Design

FUSION 360 Schematic Design

SD401-Worksheet Week 7

WK7 Assignment: FUSION 360 Schematic Worksheet

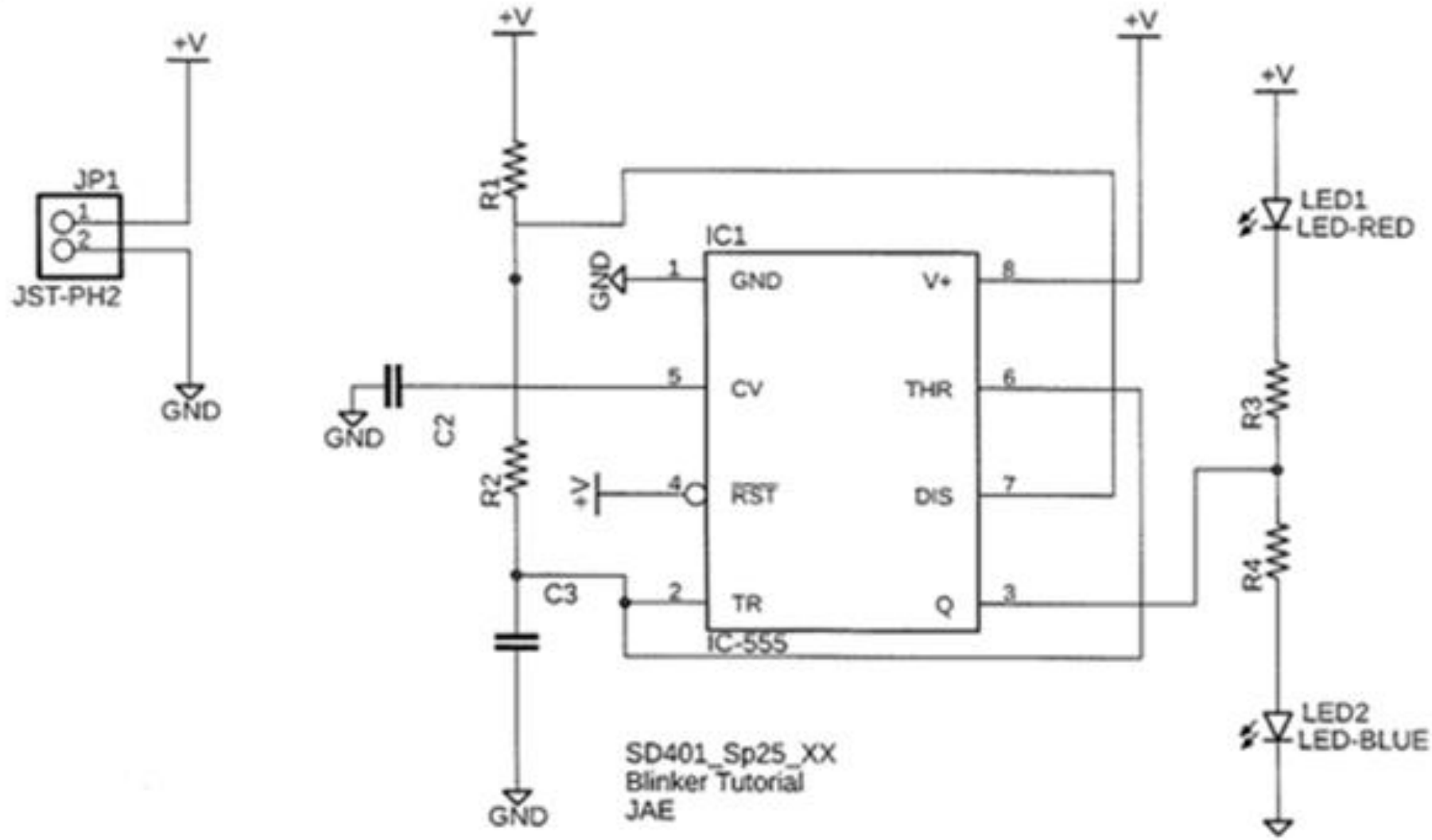
- Create an ECAD Schematic Drawing of a LED Blinker Circuit from the Schematic provided
- Save a file of the Parts list created from Fusion 360 in two separate versions, Text File, & HTML
- 1)Save the Parts List with the options of
  - *a)* List Type: Parts, Output Format: Text File, Save File as Group# Parts TEXT
    ex.SD401_Sp25_XX_Parts_Text
  - b) List Type: Values, Output Type: HTML, Save File as Group# Values_ HTML

Worksheet Objectives:

- ✓ Worksheet for PCB fabrication- Step 1
- ✓ Through Hole Components required- NO SMD'S
- ✓ Add Components to an ECAD Schematic Design
- ✓ Add Text, with different Font size
- ✓ Add Nets (air wires)
- ✓ Identify Layers
- ✓ Add Connectors from library
- ✓ Add PWR and GND
- ✓ Save and Print the ECAD Schematic document as an image and send via email
- ✓ Create and Print the Bill of Materials, in two different styles, showing parts vs values

Email the .pdf image of the Schematic, and the two variations of the Parts List, in a .zip folder

Send to jeffrey.erickson@ndsu.edu


+V
JP1
1
2
JST-PH2
GND
+V
R1
IC1
GND
1
GND
V+
8
5
CV
THR
6
GND
C2
R2
+V
4
RST
DIS
7
C3
2
TR
Q
3
IC-555
GND
SD401_Sp25_XX
Blinker Tutorial
JAE
+V
+V
LED1
LED-RED
R3
R4
LED2
LED-BLUE

Partlist exported from C:/Users/Tyler/AppData/Local/Temp/Neutron/ElectronFileOutput/6832/sch-0010a567-5775-4050-8bbf-21250c60f2e5/Untitled.sch at 2/9/2025 9:28 PM

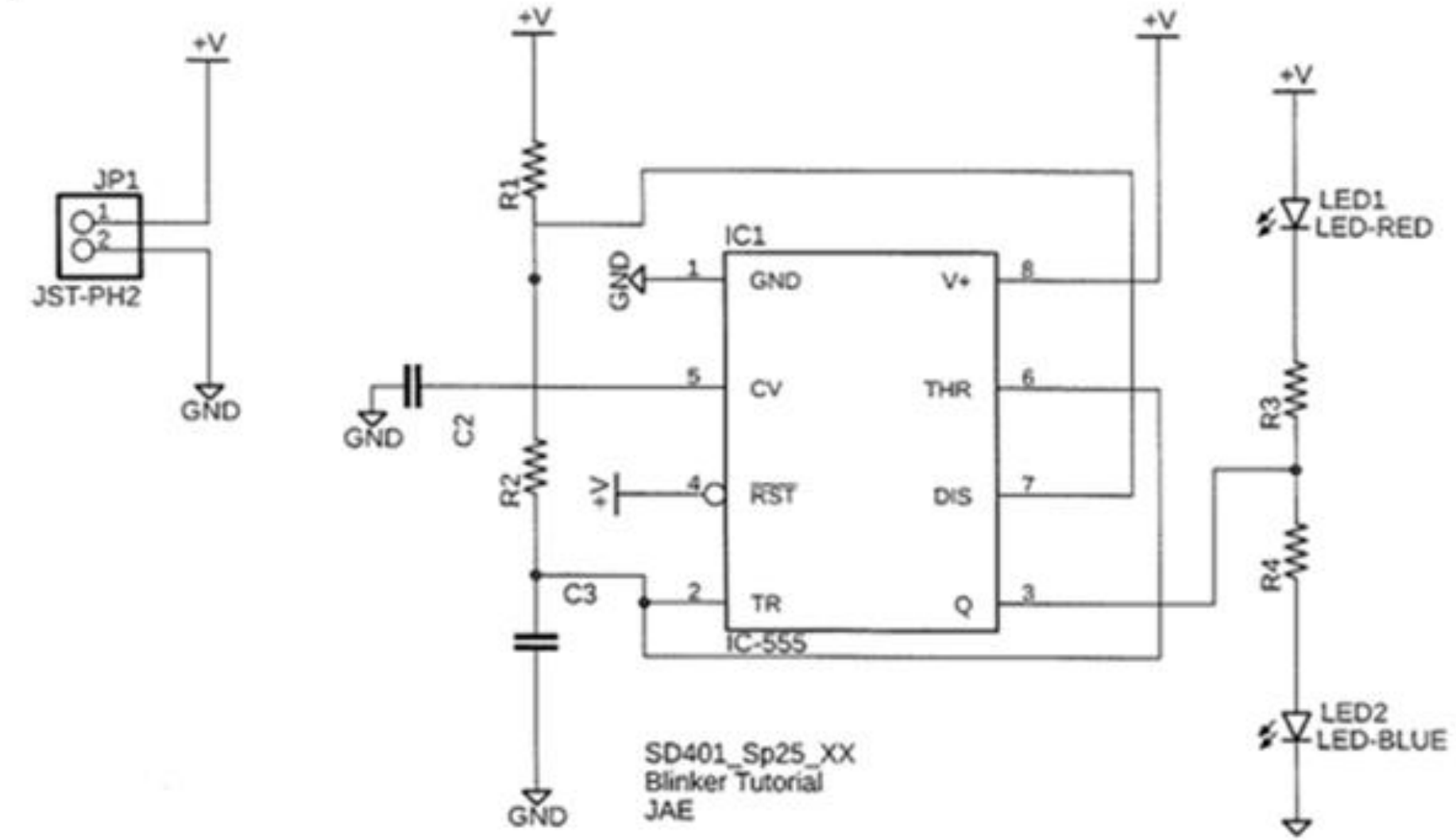
Part	Value	Device	Footprint Name	Detailed Description	CATEGORY	COLOR	DESCRIPTION	MANUFACTURER	MPN	OPERATING_TEMP	PART_STATUS	RATING	ROHS_COMPLIANT	SERIES	SUB-CATEGORY	THERMALLOSS	TOLERANCE	TYPE	VALUE	VOLTAGE_RATING
C1		C_CHIP-0402(1005-METRIC)	CAPC1005X60	Capacitor - Generic	Capacitor															
C2		C_CHIP-0402(1005-METRIC)	CAPC1005X60	Capacitor - Generic	Capacitor															
IC1	IC-555	LM555ND0008A	D0008A																	IC-555
JP1	JST-PH2	JSTPH2	JSTPH2																	JST-PH2
LED1	LED-BLUE	LED_CHIP_BLUE-2012	LEDC2012X110N_B	LED - Generic	Opto-Electronic	Blue														LED
LED-BLUE																				
LED2	LED-RED	LED_CHIP_RED-1608	LEDC1608X39N_R	LED - Generic	Opto-Electronic	Red														LED
LED-RED																				
R1		R-US_CHIP-0402(1005-METRIC)	RESC1005X40	Resistor Fixed - ANSI	Resistor													Fixed		
R2		R-US_CHIP-0402(1005-METRIC)	RESC1005X40	Resistor Fixed - ANSI	Resistor													Fixed		
R3		R-US_CHIP-0402(1005-METRIC)	RESC1005X40	Resistor Fixed - ANSI	Resistor													Fixed		
R4		R-US_CHIP-0402(1005-METRIC)	RESC1005X40	Resistor Fixed - ANSI	Resistor													Fixed		

FUSION360 Parts list downloaded and saved as a TEXT Document

Partlist exported from C:/Users/Tyler/AppData/Local/Temp/Neutron/ElectronFileOutput/6832/sch-0010a567-5775-4050-8bbf-21250c60f2e5/Untitled.sch at 2/9/2025 9:28 PM

Qty	Value	Device	Footprint Name	Parts	Detailed Description	CATEGORY	COLOR
	DESCRIPTION		MANUFACTURER	MPN	OPERATING_TEMP	PART_STATUS	RATING
	ROHS_COMPLIANT		SERIES	SUB-CATEGORY	THERMALLOSS	TOLERANCE	TYPE
	VALUE	VOLTAGE_RATING					
2		C_CHIP-0402(1005-METRIC)		CAPC1005X60	C1, C2	Capacitor - Generic	Capacitor
4		R-US_CHIP-0402(1005-METRIC)		RESC1005X40	R1, R2, R3, R4	Resistor Fixed - ANSI	
	Resistor Fixed						
1	IC-555	LM555ND0008A	D0008A	IC1			
							IC-555
1	JST-PH2	JSTPH2	JSTPH2	JP1			
						JST-PH2	
1	LED-BLUE	LED_CHIP_BLUE-2012		LEDC2012X110N_B	LED1	LED - Generic	Opto-Electronic
	Blue						LED
			LED-BLUE				
1	LED-RED	LED_CHIP_RED-1608		LEDC1608X39N_R	LED2	LED - Generic	Opto-Electronic
	Red						LED

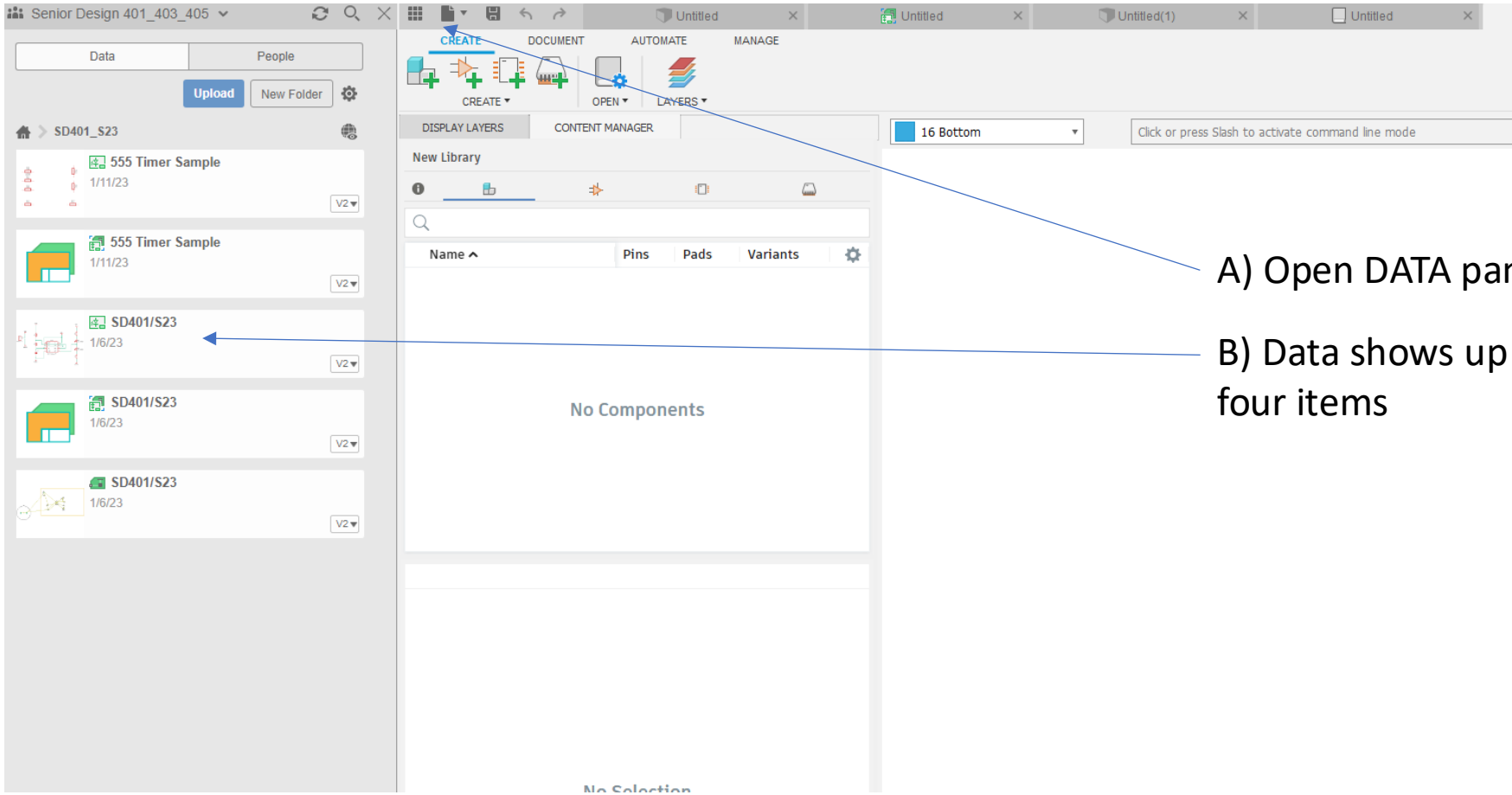
FUSION360 Parts list saved and sent as a HTML File



Step 3 is to turn this schematic into a PCB Design and save as a Gerber file.

Note: In ECE 401, you have some constraints that will be lifted in ECE 403/405

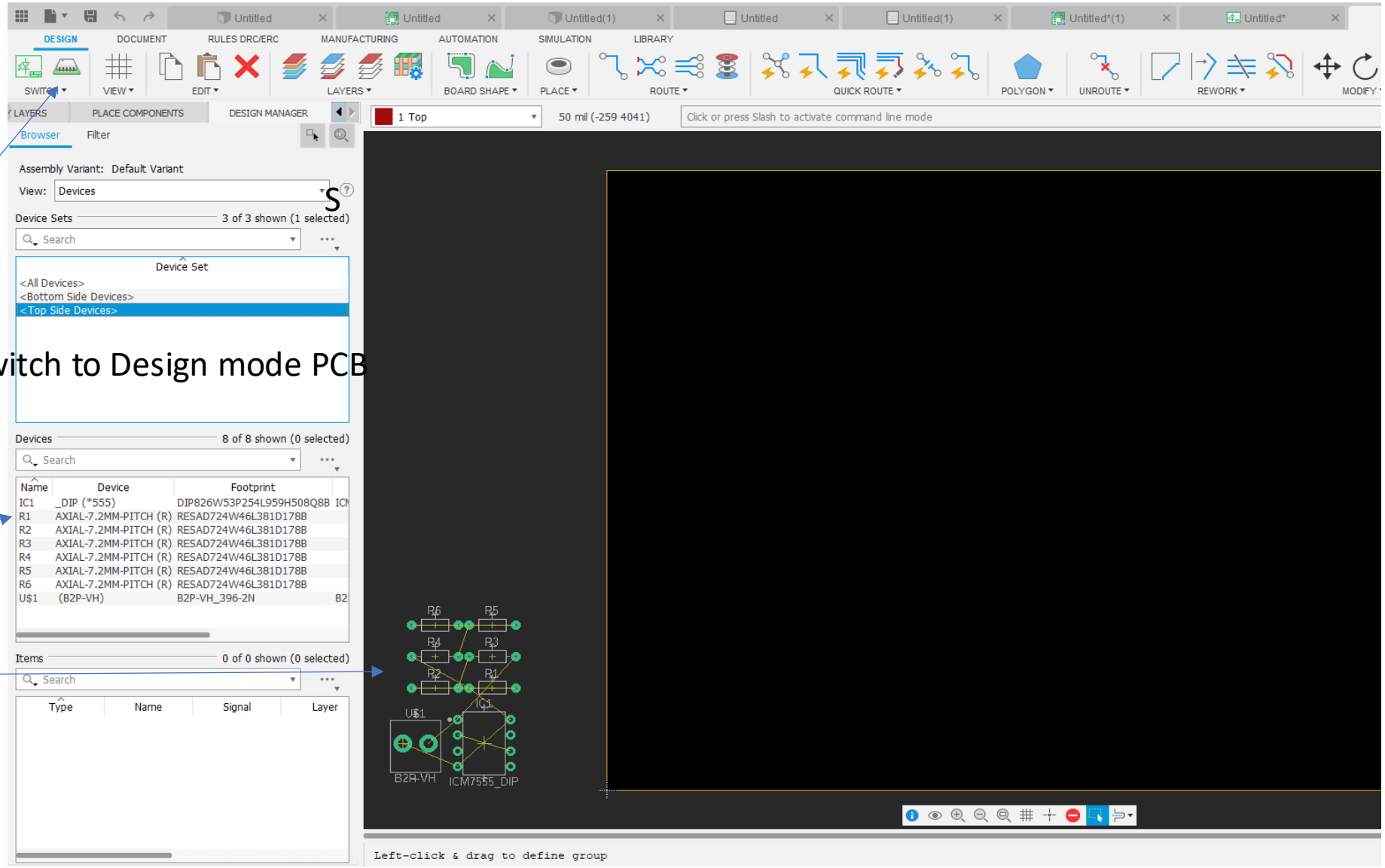
	ECE 401	ECE 403/405
PCB Size	2" x 2" 2”x3”	up to 60 square inches
Mounting Holes	200 mils	200 - 250 mils
Ground Plane	yes	yes
Power Plane	yes	Depends upon design
Trace Width: Power	40 mils	8 mils to 600 mils
Other Traces	20 mils	8 mils to 600 mils
Test Points	yes Through Hole	yes Surface Mount or Through Hole
Components	Through Hole	any (0805, TSOP, DIP recommended)
Silk Screen (top)	yes include date & group number	yes include date & group number
Silk Screen (bottom)	no	yes if components placed on both sides of board
Font Size	50 mil or larger height/10 for thickness	50 mil or larger height/10 for thickness
Digikey Trace Width Calculator	optional	Longest trace with highest current
LEDs	5mm Through Hole 10mA current Power, Signals	Any size, any number 0805 recommended Power, Signals
Power	9V battery 7805 to step down to 5VDC	any
Fuse	1 Ohm resistor Add reverse polarity protection	optional
Microcontroller (if used)	Raspberry Pi-Pico	Any



A) Open DATA panel

B) Data shows up showing four items

[PCB Layout Tutorial Walkthrough – YouTube](#) 0:04/4:38



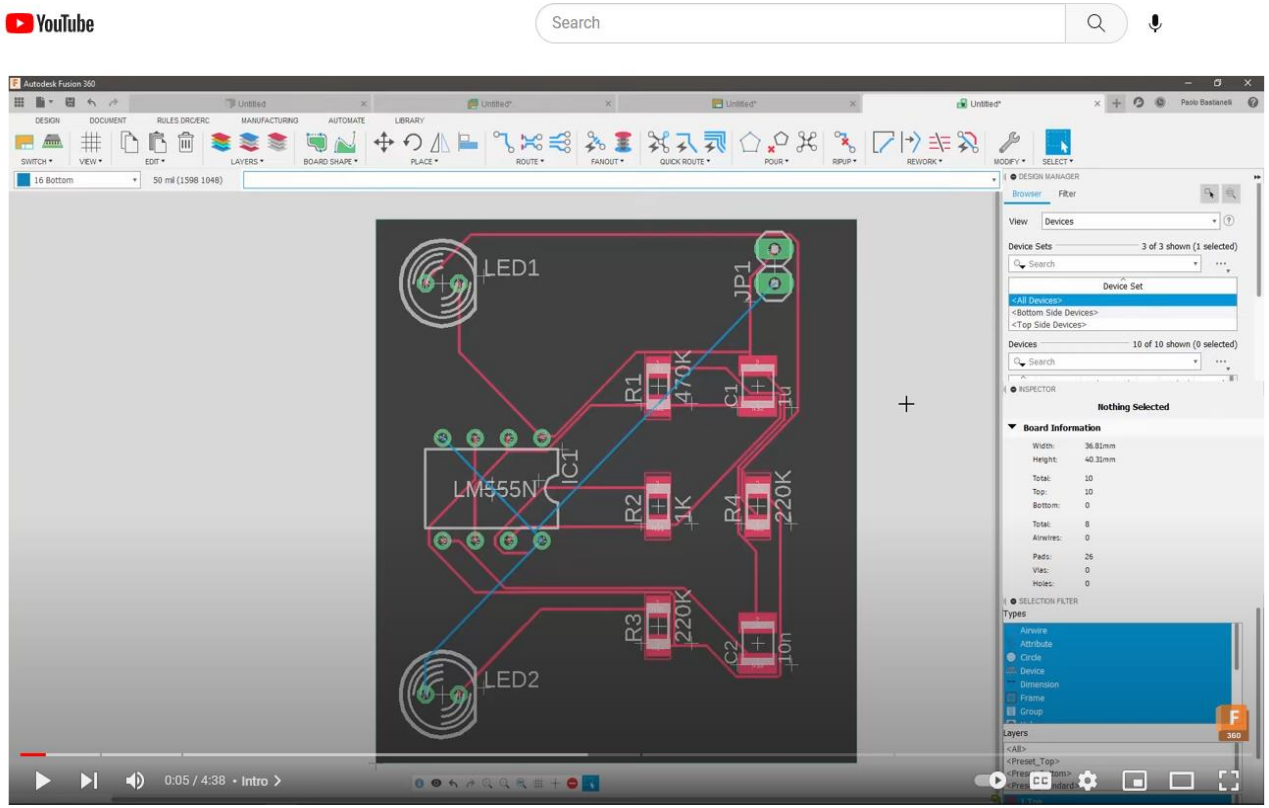
Switch to Design mode PCB

List of components in Design

2D Sim of components in design

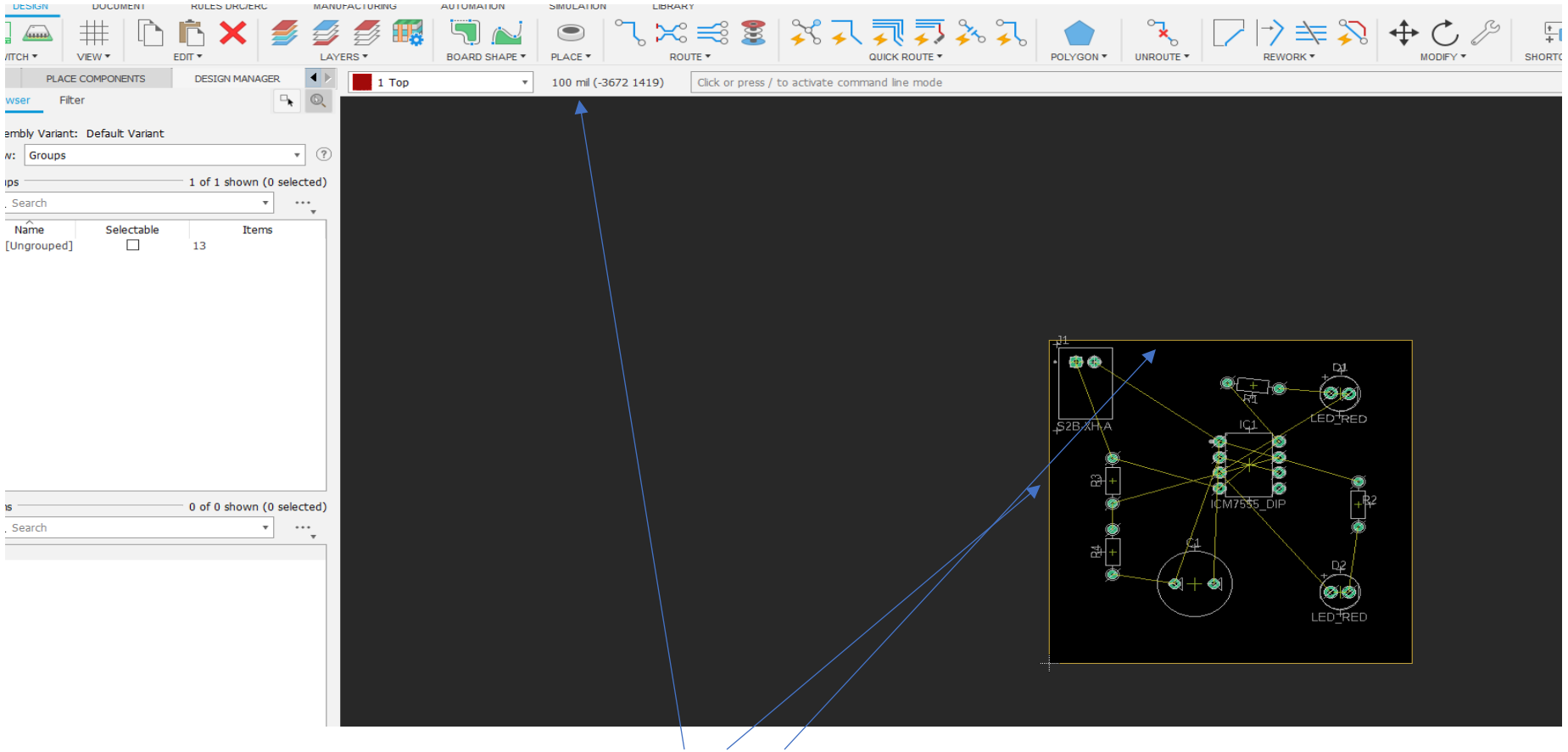
Part 2 Creating Circuit Board Layout

https://www.youtube.com/watch?v=VZZBEocoYDA https://youtu.be/VZZBEocoYDA



PCB Layout Tutorial Walkthrough – YouTube 0:04/4:38

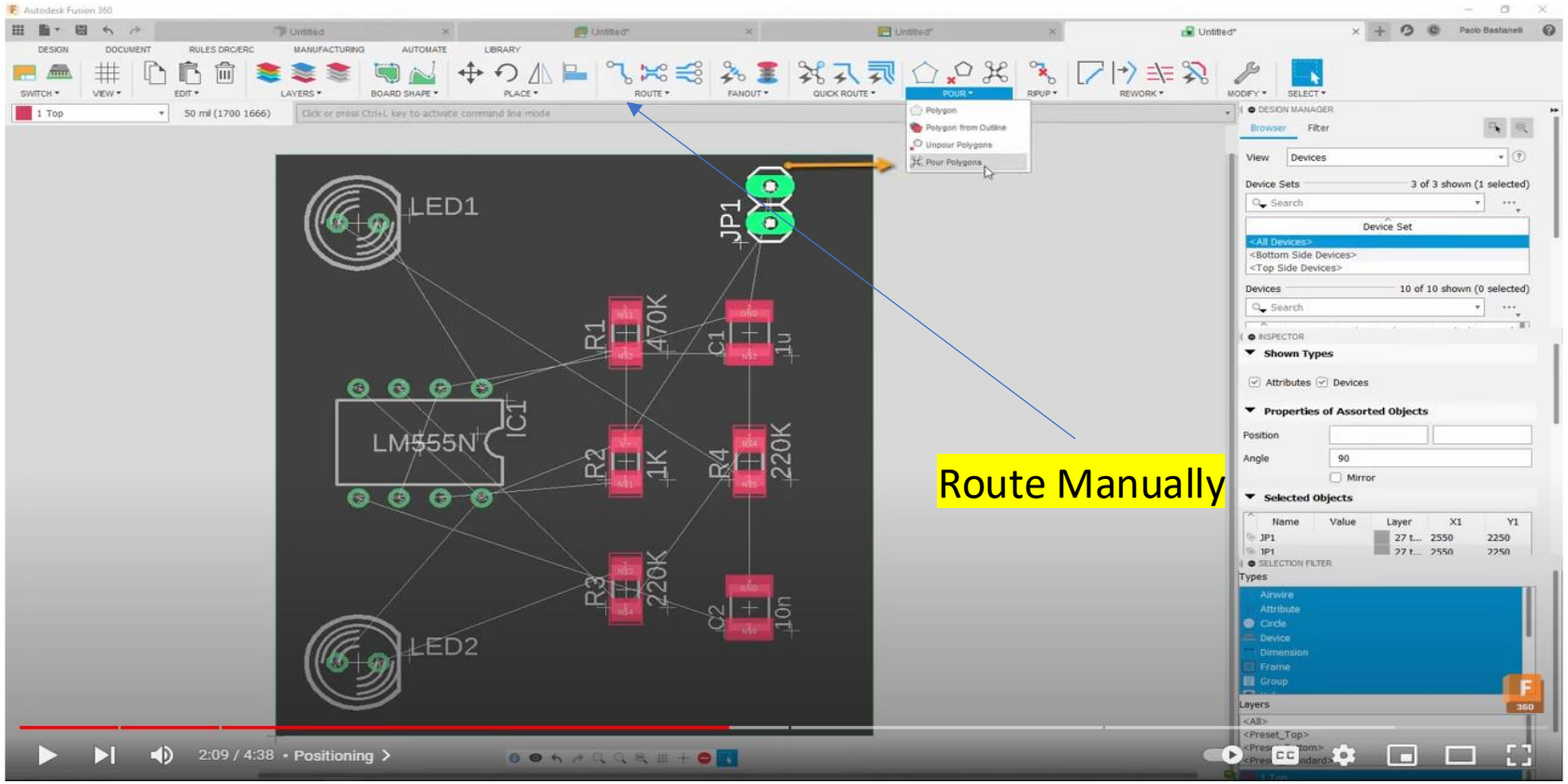
2D sim Components have been dragged into the Black square this is your PCB out line



To change dimension of your board take note of the origin 0,0, pcb design is pos x,y
Grab the top and sides of design to change the dimension of the PCB

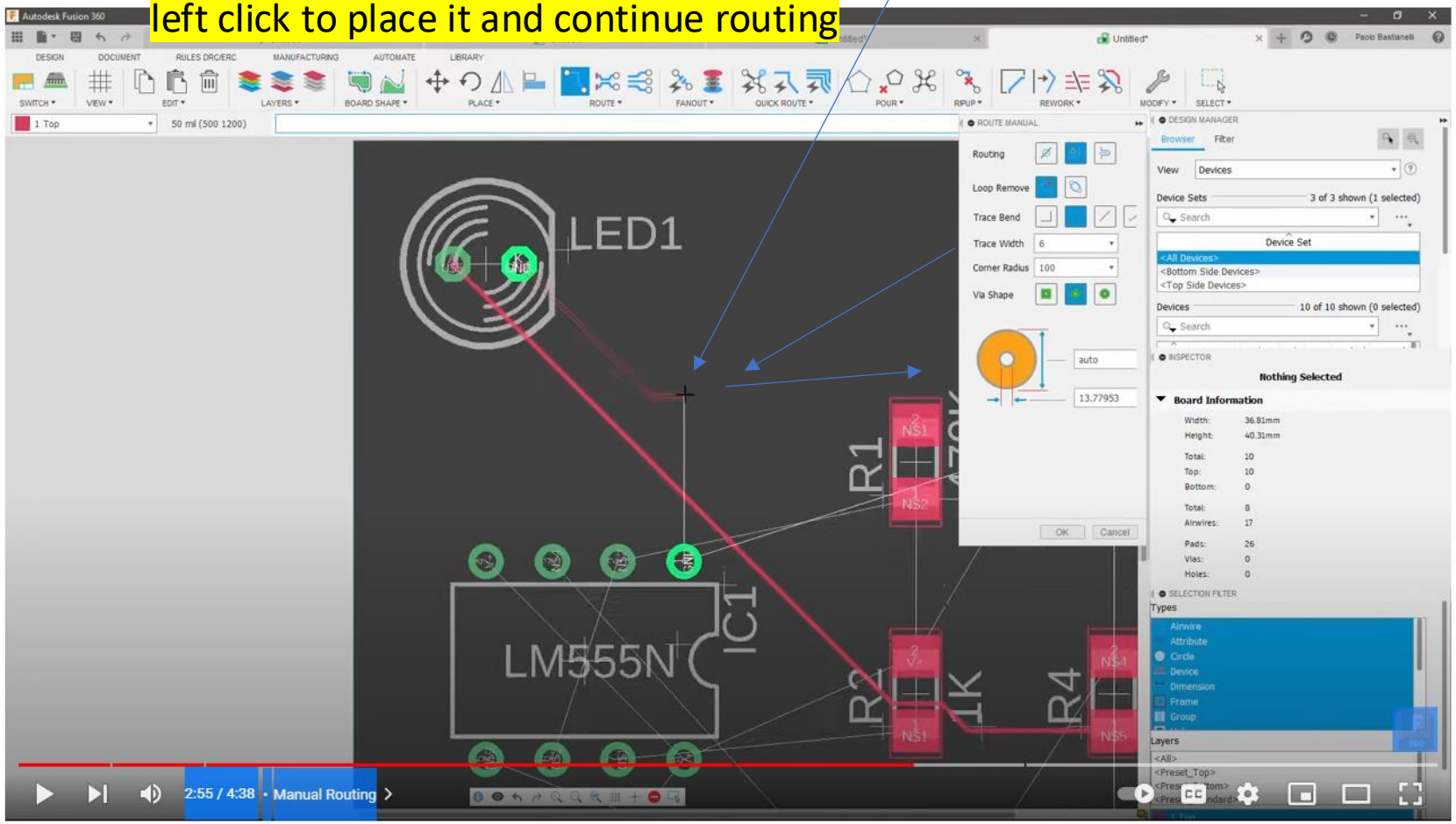
PCB Layout Tutorial Walkthrough – YouTube 0:42/4:38

To route traces manually and then automatically



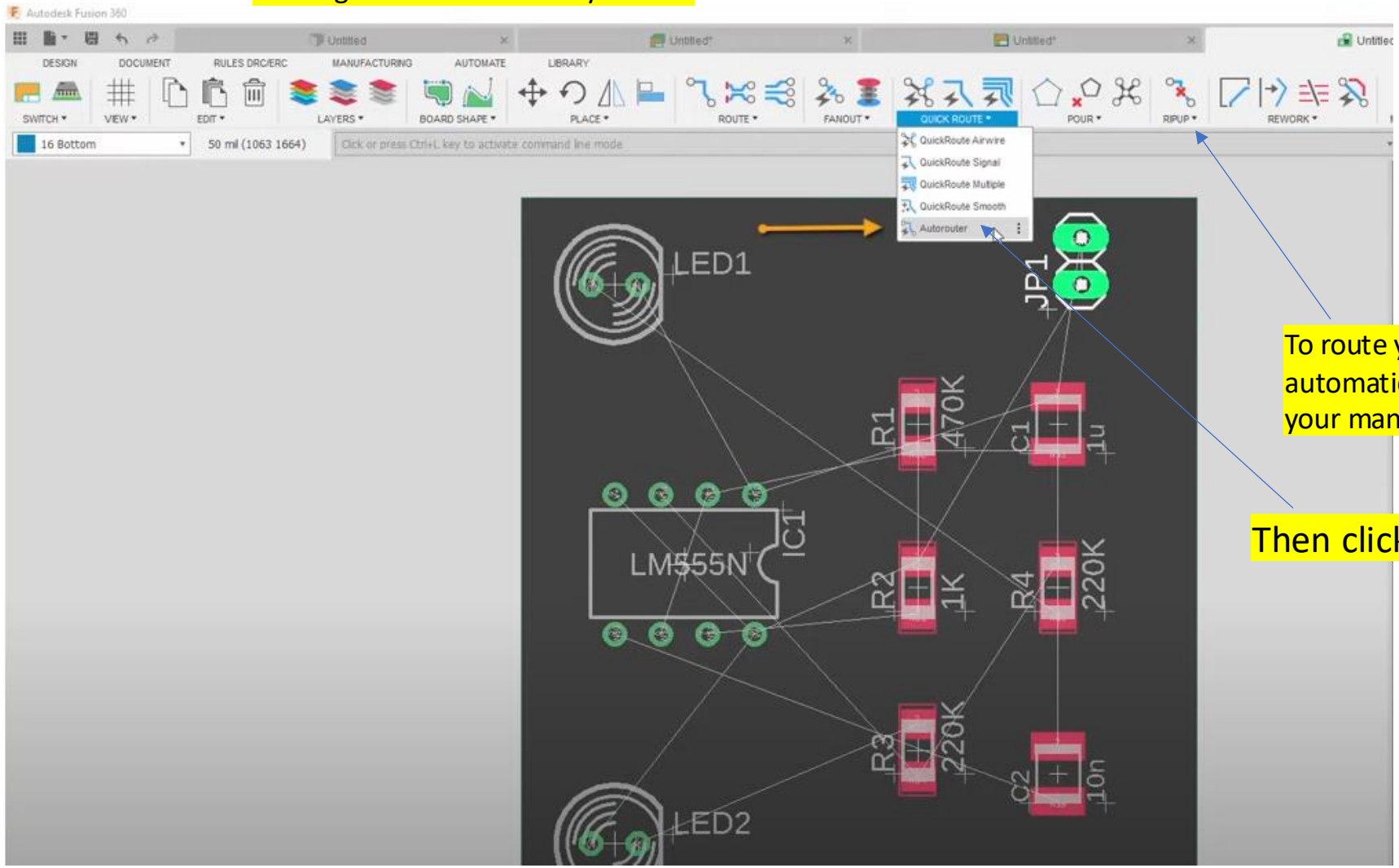
[PCB Layout Tutorial Walkthrough – YouTube](#) 2:32 mark

Traces Top to bottom of board require a via , start a trace, then hit space bar, left click to place it and continue routing



https://www.youtube.com/watch?v=VZZBEocoYDA 2:55/4:38

Routing the traces manually vs auto

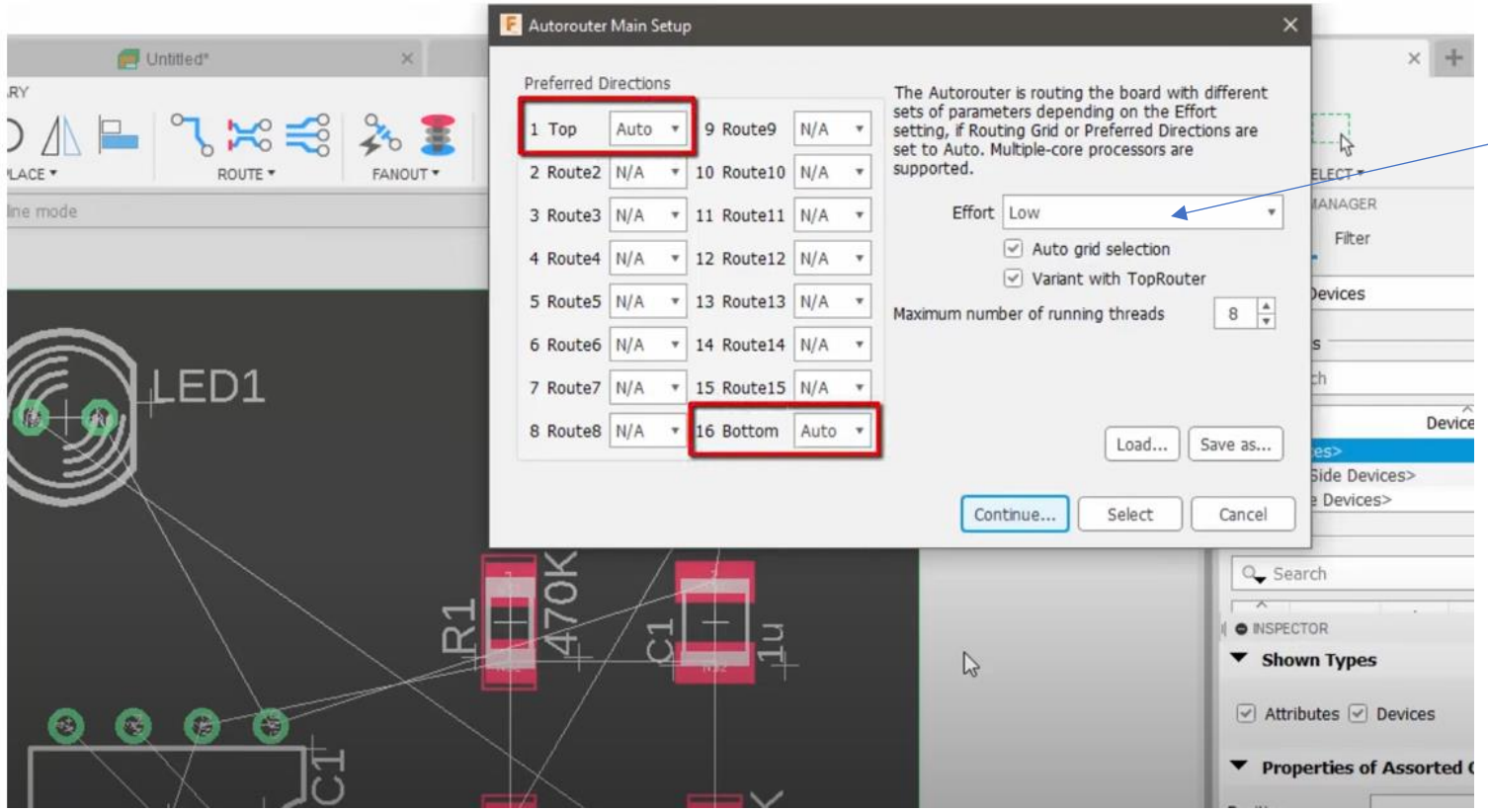


To route your traces automatically you may rip up your manual ones first.

Then click auto router

https://www.youtube.com/watch?v=VZZBEocoYDA 3:39/4:38

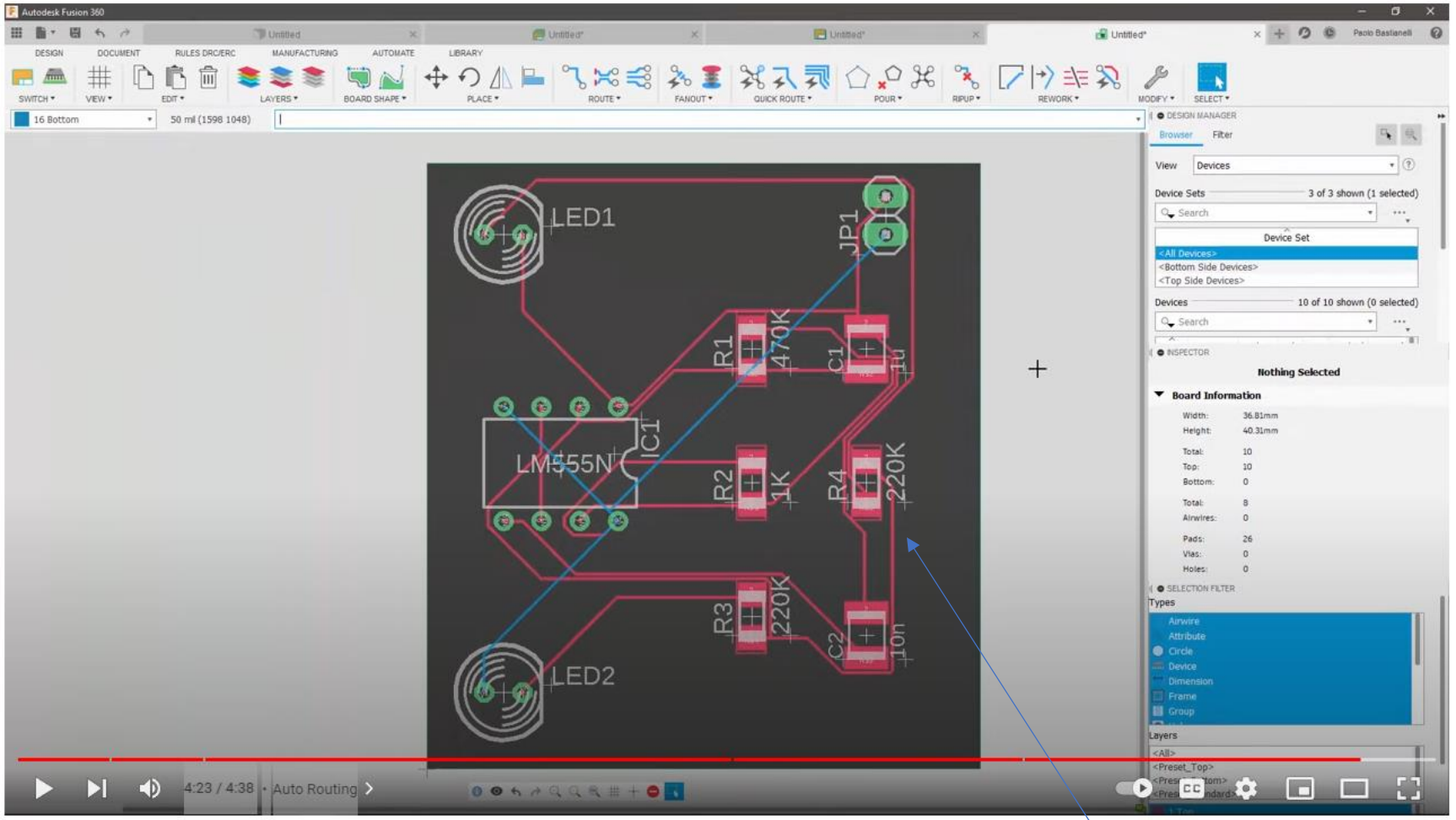
Two layer boards only top and bottom will be selected to auto route



Change the effort to high to get a better selection of solutions

https://www.youtube.com/watch?v=VZZBEocoYDA

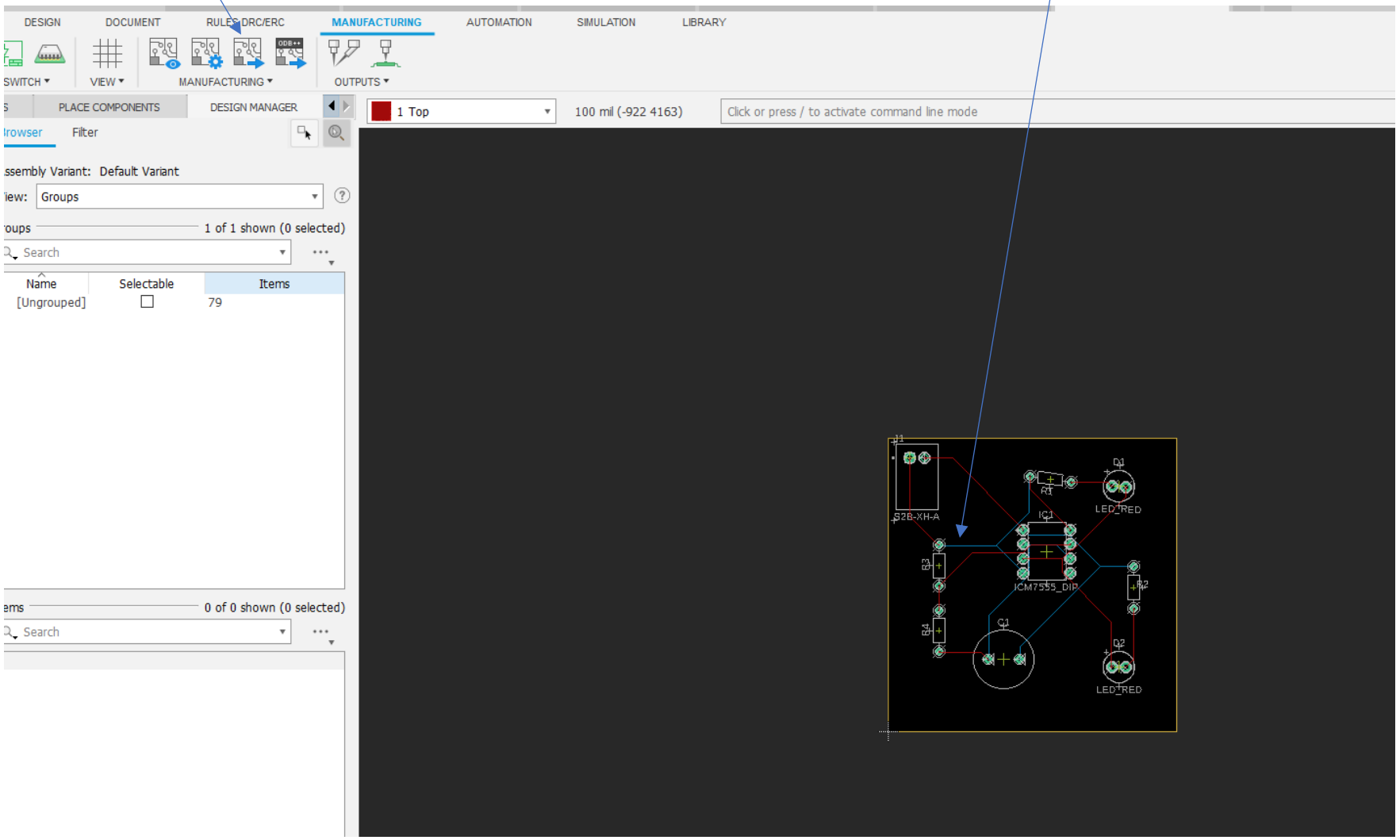
After choosing the solution with the least amount vias, the PCB is complete



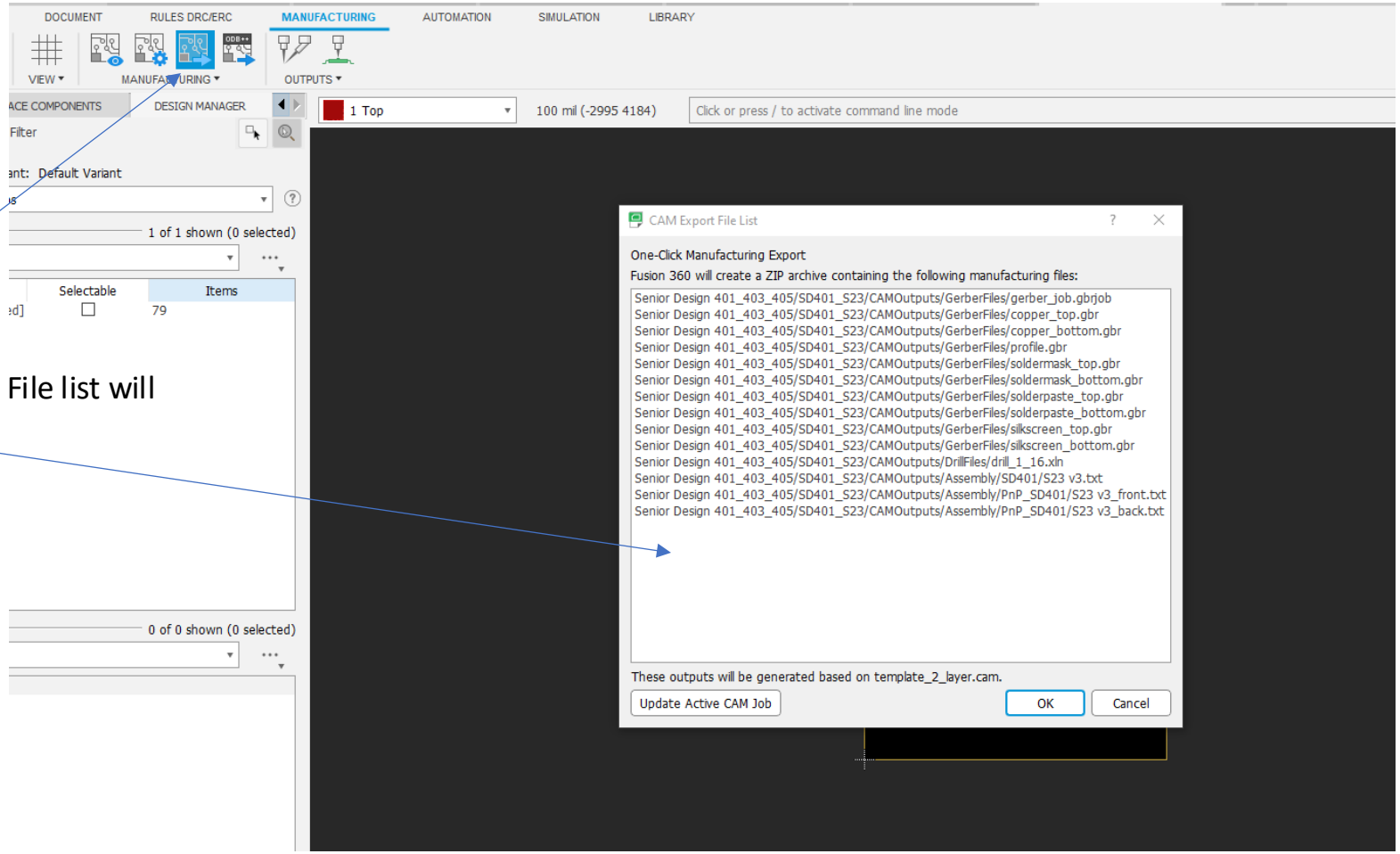
SMD's are not allowed in ECE401 Designs, , SMDS's that show as RED need to be replaced with through Hole components first.- Highlight- right click- choose Package-Variant– Unroute the Design – change the parts and reroute.

PCB Manufacturing- Creating Gerber Files

Notice the Parts are all Through Hole Components, SMD Components can be changed by highlighting, right click and choose correct Variant

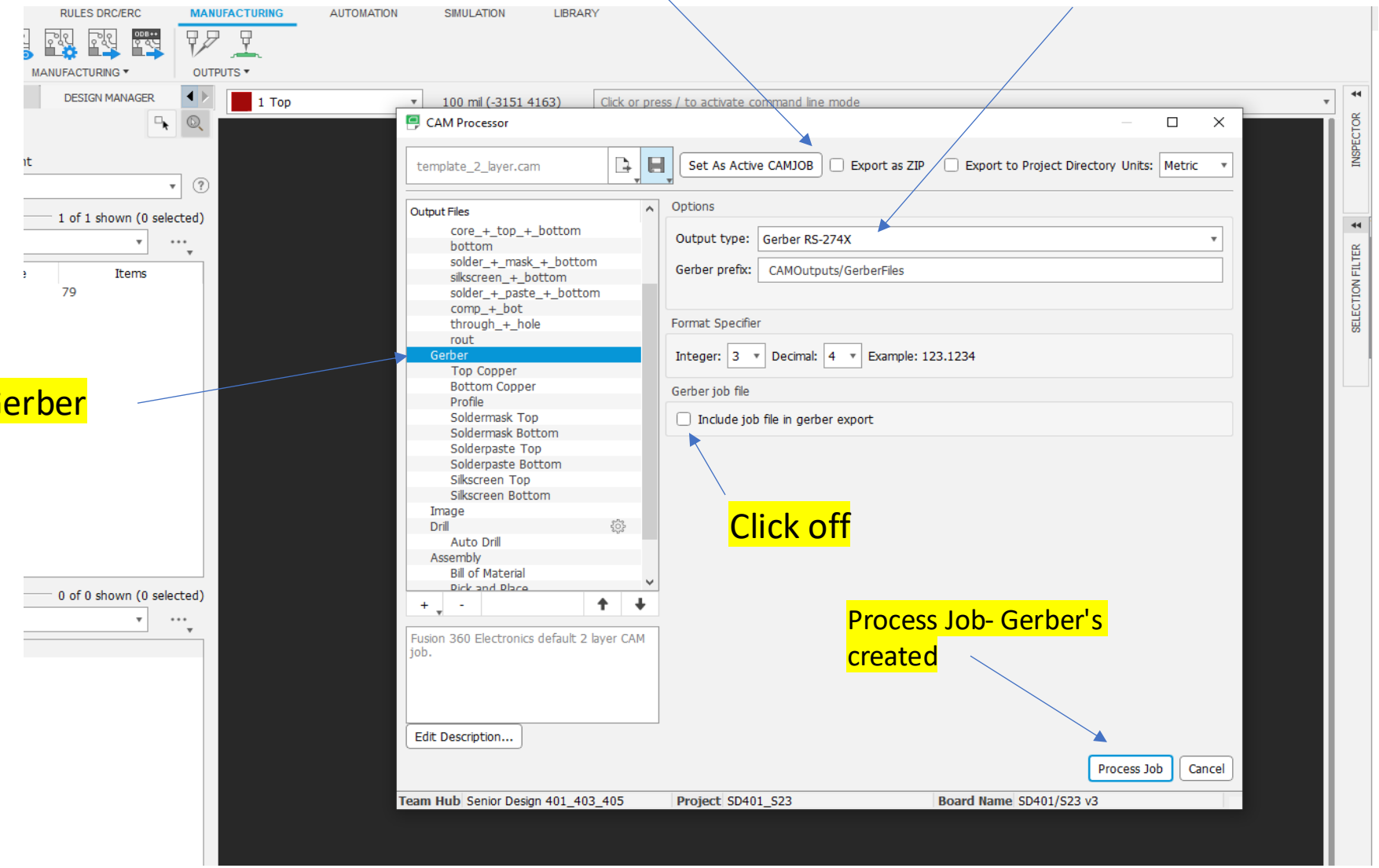


Once clicked the CAM Export File list will be created, click OK



Check the Export to zip box

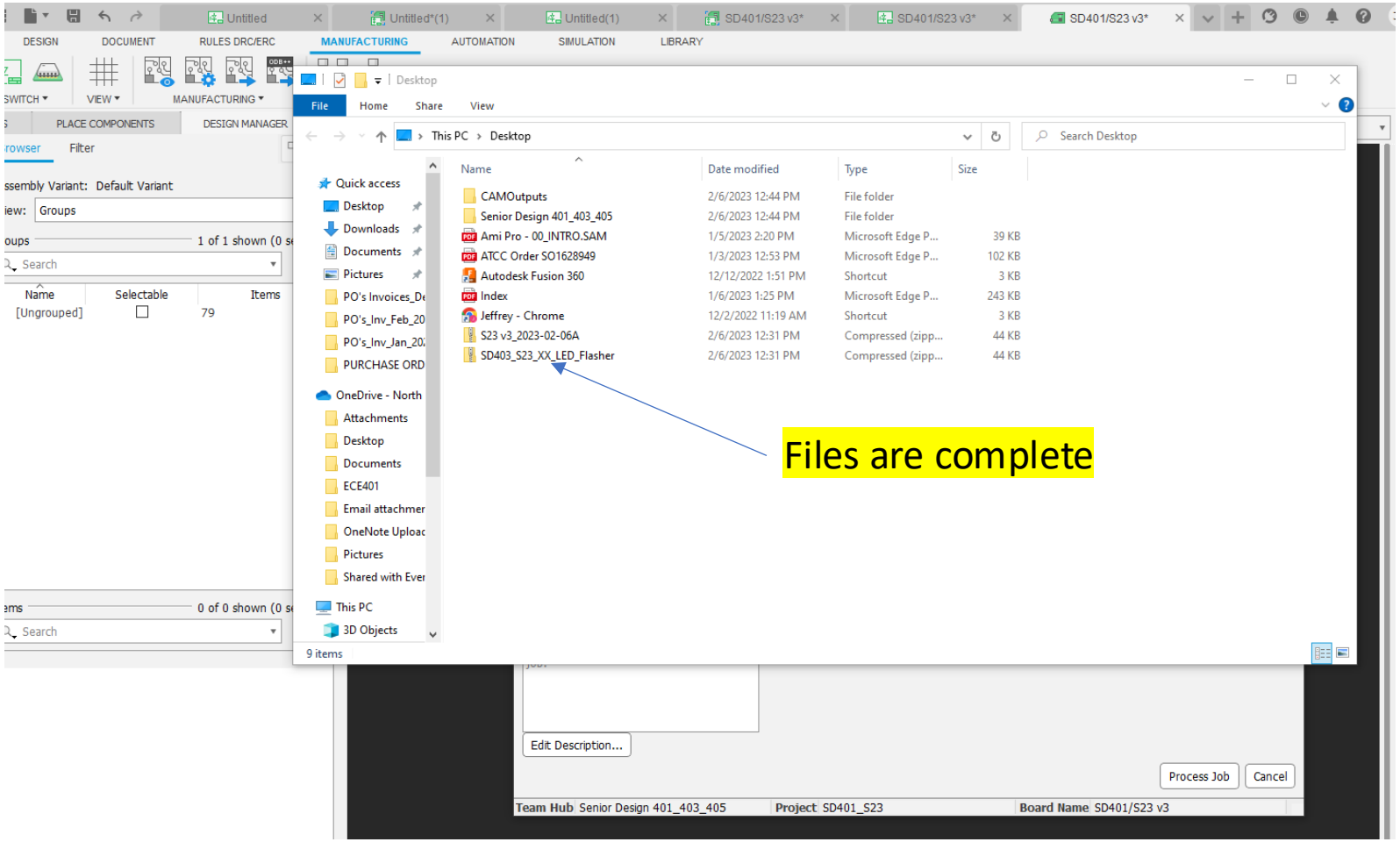
Creating Gerber RS_274X files



Click to Gerber

Click off

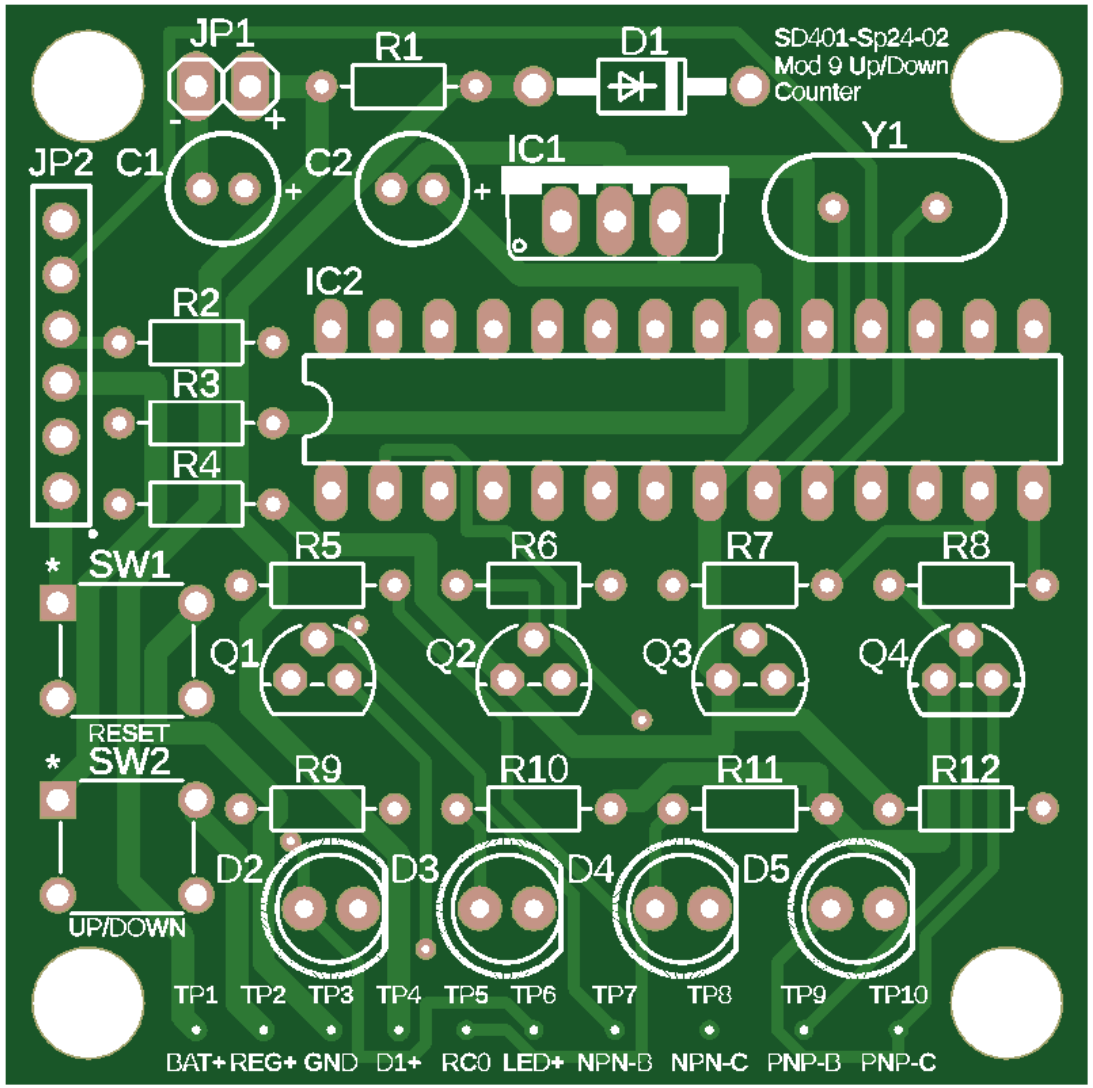
Process Job- Gerber's created

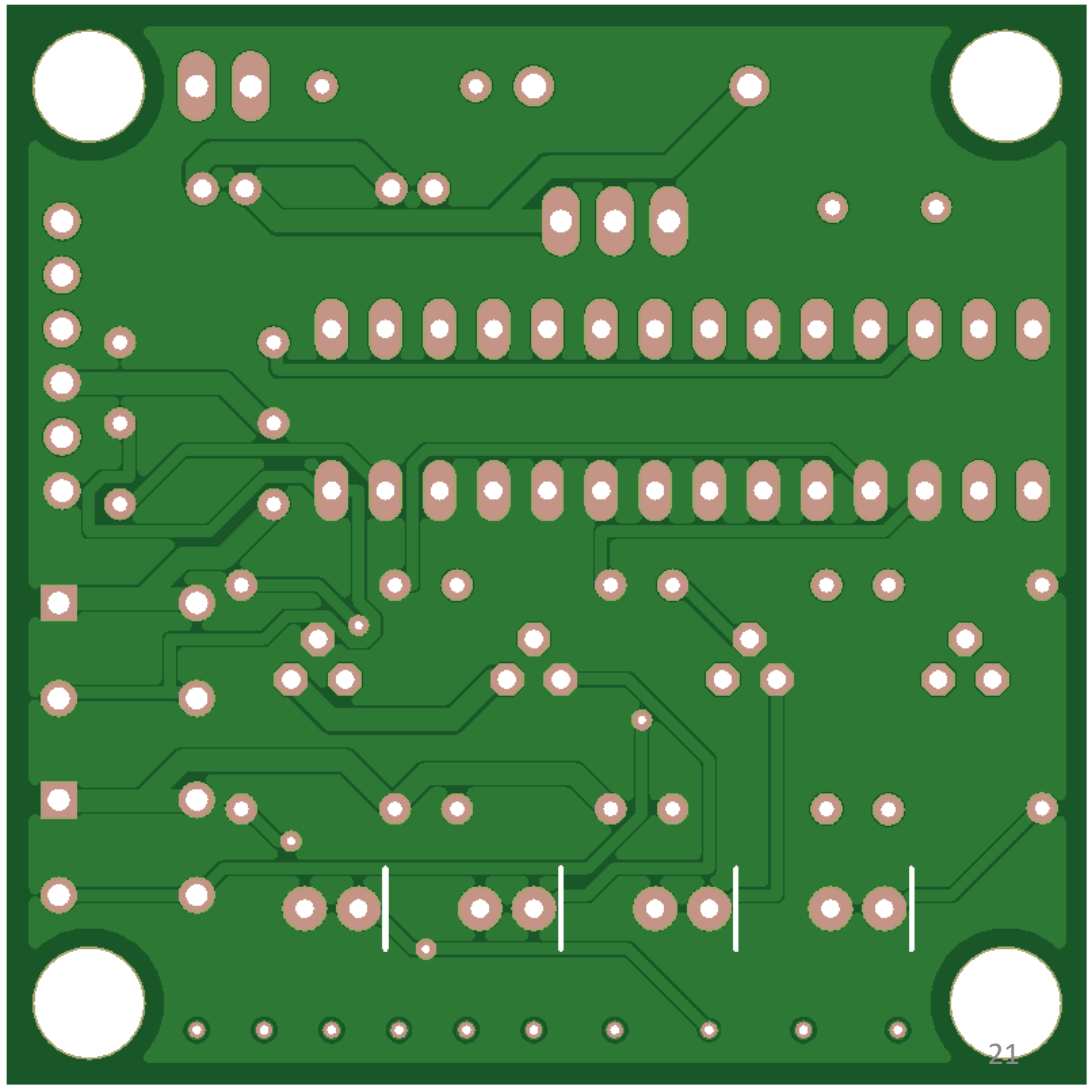


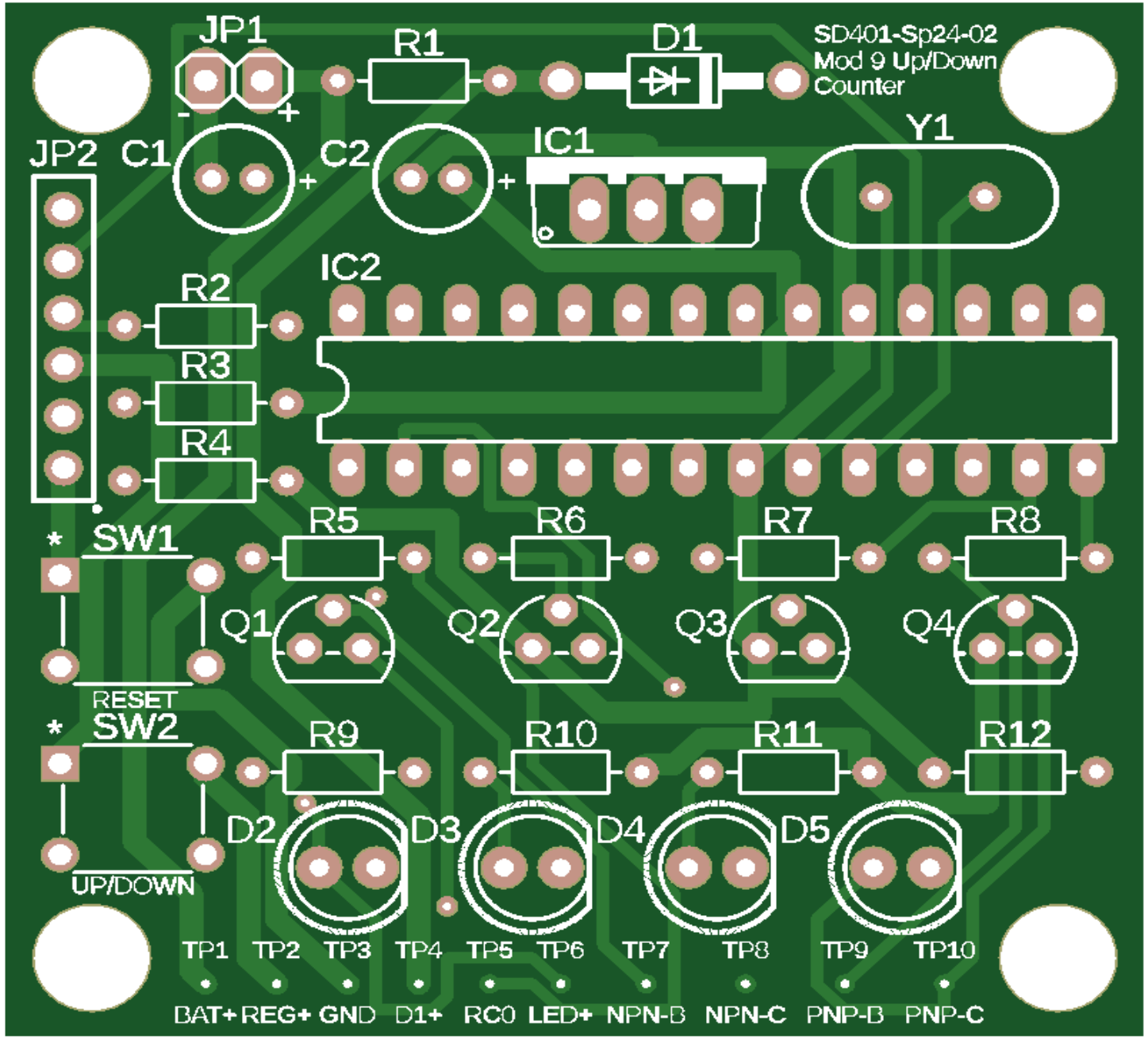
CAM Files are now Gerber files
Change File name to SD403_S23_XX_Flasher and
Email to Jeffrey.Erickson@ndsu.edu for verification and ordering

Part II: Getting Started with Fusion 360

PCB Design







This image shows the top view of a PCB design

- Take note of the 4 (white) holes in the corners- these are mounting holes to secure the PCB into an enclosure. (use 175mils diameter)
- JP1 is a 1x2 header, which shows the polarity of the battery leads when soldered to.
- Look at the uniformity of the Text, showing the Part designator- Part designator is more important than showing the Value of the part although both can be included
- Looking at the width of the traces- some are 40mils, the rest are 20 mils. 40mils for power, and 20mils for Data lines.
- Test points or TP's are labeled TP Bat+. TP GND, TP VREG etc.

EXTRAS: FUSION 360 has a Design documentation for every level of expertise

- Get Started in Fusion 360
- What's new
- Collaborate with Fusion Team
- Extensions
- Tokens
- Assemblies
- Design: Sketch
- Design: Solid
- Design: Surface
- Design: Mesh
- Design: Form
- Design: Sheet Metal
- Electronics
  - Electronics overview
  - Projects and workflow
  - Component libraries
  - Schematic design
  - Board layout preparation
  - Computer-aided manufacturing (CAM) support
  - Tutorials
    - Tutorial: Manage electronic component libraries
    - Tutorial: Create a schematic design

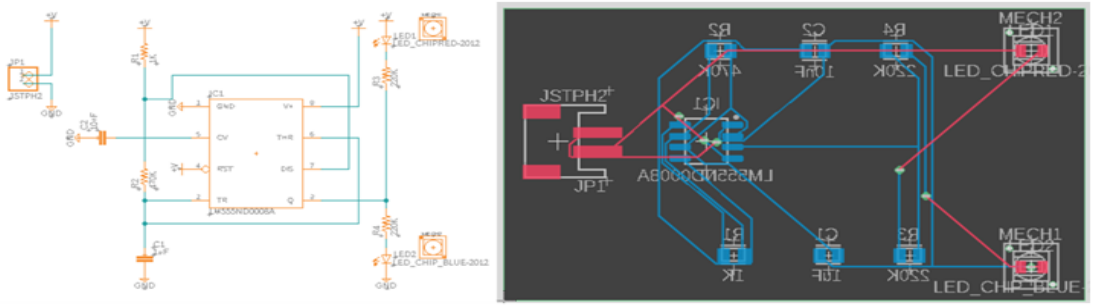
Electronics / Tutorials / Tutorial: Create a PCB layout

Tutorial: Create a PCB layout

The printed circuit board (PCB) layout process is both an art and a science. If you give a schematic to 100 different enginee PCB layouts back, all with unique twists.

In this tutorial, you start with a schematic design, and create a PCB using the following steps:

- Defining the PCB shape.
- Placing components.
- Routing the connections.



Schematic converted to a PCB for the double LED flasher circuit

Prerequisites

- Ensure you have completed the tutorial Create a schematic design.
- Ensure the design you created in the **Create a schematic design** tutorial is open and you are in the **Schematic** workspac

https://help.autodesk.com/view/fusion360/ENU/?guid=ECD-TUT-PCB-TOP-LEVEL

https://cart.jlcpcb.com/quote/gerberviewThree/?qs=fffb0b8182344870b0bbd70c37469468_1_0_1_0_0.html

- Once Completed send your Gerber files to a on line service such as JLCPCB.com, here you will see your final image: highlight and send the above file using your url
- Click Top View Tab
- Click Bottom View Tab
- 2D image Tab
- 3D image Tab
- Layers Tab
- Layers Tab will show you multiple layers required to make a pcb gerber file
- Click DFM check - to analyze your board for rules violations- click DFM Check

FUSION 360 PCB Design

SD401 Worksheet Week8

WK7 Assignment: From the worksheet **FUSION 360 Schematics Part1**.

- Create an ECAD Schematic Drawing of a LED Blinker Circuit
- Email the .pdf image, and two variations of the Parts List.

WK8 Assignment: LED Blinker PCB with Gerber Files Worksheet Part 2

- Using the schematic of the LED Blinker, create a PCB design using the components shown in the previous slide

  Requirements: Through Hole Components are to be exclusively used for component selection

- Use only Through Hole Components
- Resistor Size:
- IC package: DIP, PDIP
- Power input is 9VDC - 9V Battery
- DC input have two options:
  1st option: use a JST 1x2 connector with 100mil/2.5mm pitch-JST Jack
  2nd option uses the following DC jack as it is used in many SD Projects:
  Suggest using this connector from this Library: OPL-Connector Library, Variant-'DC-005', description is Jack DC-005 (used in many SD Projects)
- DC input is 9Vdc – need a 5V regulator.
  Voltage regulator & size LM7805 TO220 package (Variant)
- 1ohm ¼ watt resistor as a fuse and reverse current diode for circuit protection (use a 1N4001,1N4002, or a 1N4003 PN junction diode
- Label the DC input polarity
- Board Size: 2000mil x 2000mil (2"x2")
- Text showing Group Designator, Project name: LED Blinker, Vs.# on front & back of PCB
- Text in an orderly manner
- Mounting holes: 4 corners
- Mounting Hole diameter: 175 mils
- May use any components from any Parts Library- Tutorial Fusion360 is just an option
- Parts may be placed on TOP Layer only
- Power traces 40 mils
- All other traces 20 mils
- GND plane required

Send the Gerber file which includes to drill files to any Gerber Viewer program, such as OSHPARK.com, https://www.gerber-viewer.com, or jlcpcb.com
Email the renamed .zip file folder to Jeffrey.erickson@ndsu.edu

[Fusion 360 Help | Computer-aided manufacturing (CAM) support | Autodesk]

Creating Gerber files