


**SD401**

**Getting Started with  
Fusion360**

## Getting Started: Step by Step creating a PCB with Fusion 360

Sign in 

Email

**NEXT**

[NEW TO AUTODESK? CREATE ACCOUNT](#)

1) Create an Account. Autodesk Fusion 360 is free for one year for any student with a valid NDSU email.

2) [https://www.autodesk.com/products/fusion-360/education?AID=10282382&PID=100357191&SID=tuid%3A2982E00743266CAE26CCF26B47266A54&mktvar002=afc\\_us\\_deepink&cjevent=909e6a5fa63211ed8367bd20a1c0e0b&affname=100357191\\_10282382](https://www.autodesk.com/products/fusion-360/education?AID=10282382&PID=100357191&SID=tuid%3A2982E00743266CAE26CCF26B47266A54&mktvar002=afc_us_deepink&cjevent=909e6a5fa63211ed8367bd20a1c0e0b&affname=100357191_10282382)



# AUTODESK FUSION360

Fusion360 Schematic Design Tutorial

<https://www.youtube.com/watch?v=lqwHkB9lsUo>

Fusion360 PCB Design Tutorial

<https://www.youtube.com/watch?v=vZzBEocYDA>

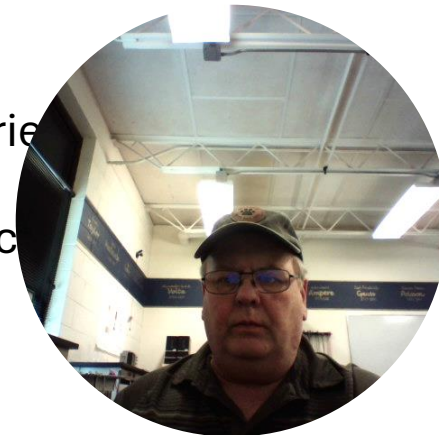
or

[https://www.youtube.com/watch?v=\\_jgUZeBiusw&list=PLmA\\_xUT-8UL80Xm8Gxz98YNum3l9Glnr](https://www.youtube.com/watch?v=_jgUZeBiusw&list=PLmA_xUT-8UL80Xm8Gxz98YNum3l9Glnr) A to Z by George Garcia

Fusion 360 is more advanced than Upverter, with the extra and more advanced libraries has a higher level of complexity.

Without watching the getting started Tutorials it will be difficult to create a Schematic then the PCB design.

There are many more videos on YouTube



## One account/ one active session at a time

### ⚠ Active Sessions Exceeded

There are more active sessions running than are allowed for this user account.  
To continue, select one of the following options:

- Suspend Fusion 360 on the computer selected below and continue on this computer.
- Shut down and sign out of Fusion 360 on the computer selected below. Unsaved changes will be saved to a recovery file.

SYSTEM NAME	OPEN SINCE	LAST ACTIVITY
ece101nb212881	2/5/2023 7:26 PM	2/6/2023 2:20 AM

- Sign in to Fusion 360 with a different account. Note: you will be signed out of running Autodesk products.
- Exit Fusion 360 now and cancel this session.

[Why am I seeing this?](#)  
[Purchase additional subscriptions here.](#)

Check again

Continue



## What does it take to make a PCB? Four steps

**1) Schematic-** After Breadboarding is finished and with the benefit of Circuit Lab schematic

Create a schematic diagram

**2) Parts list-** Libraries- Create your own library (excel works) for complex schematics or it will develop a library for you as your Schematic is created by the parts you choose from within the Fusion 360 parts libraries. Because there are so many type of package designs

Creating a known Parts file is imperative

**3) PCB Design-** Convert your Schematic to PCB Design

**4) Create Gerber files** for manufacturing- Drill and Cam files- as in a zip folder- lab with your SD project #



# PCB Layout

## ECE 401 - Homework #6

Due: Week #12

### 1) Create a detailed schematic using Fusion360

- You must use Fusion360. Kicad and other programs are not allowed
- Reference Designators must be shown on all parts
- All reference designators in a uniform and readable position.
- Text should not overlap the reference designators.
- Show values of all components.
- Schematic must have text showing Project Name and Project # (ex. SD401-Sp24-xx)
- Generate the bill of materials from Upverter
- Test points should be available for measurement {9V, 5V, ground, Input, Output, and Collector(s)}

The schematic must be saved and printed using Fusion360 both as a

- Schematic PDF
- High Res Schematic PNG

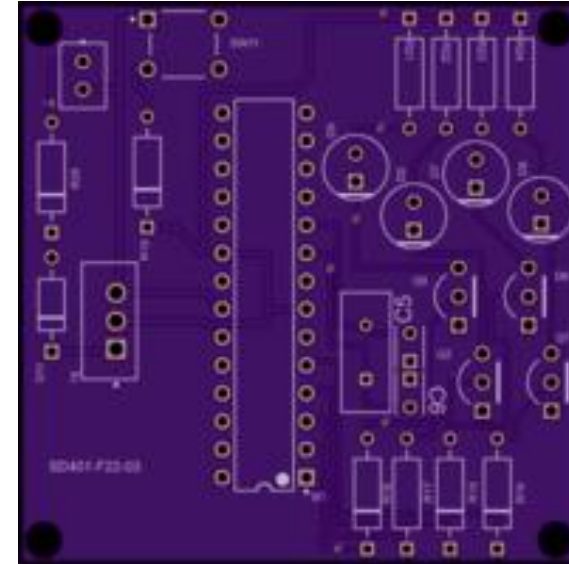
### 2) Once your schematic is approved, create Gerber files

- Using Fusion360

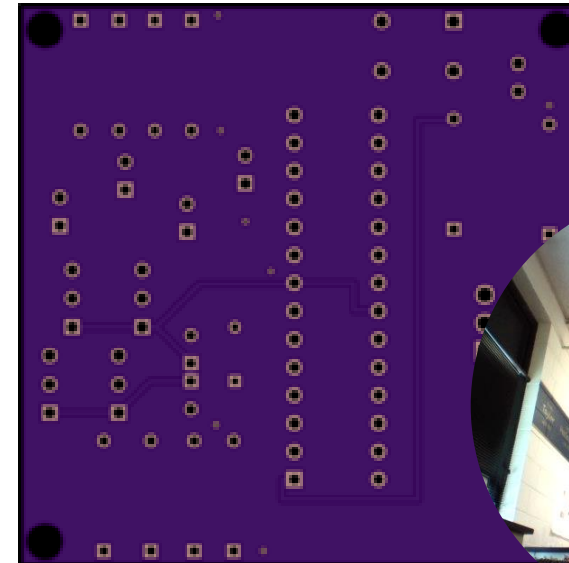
Your PCB must be

- 2.000" x 2.000"
- Mounting holes 200 mils in each corner
- Power & Ground Traces: 40mils
- Ground plane on the bottom side of PCB
- Other Traces: 20mils
- Silk-Screen designators in correct order
- Board must show the project name & team designation (SD401-Sp24-XX)
- (Font15 recommended for font size)

### Top Image of a PCB

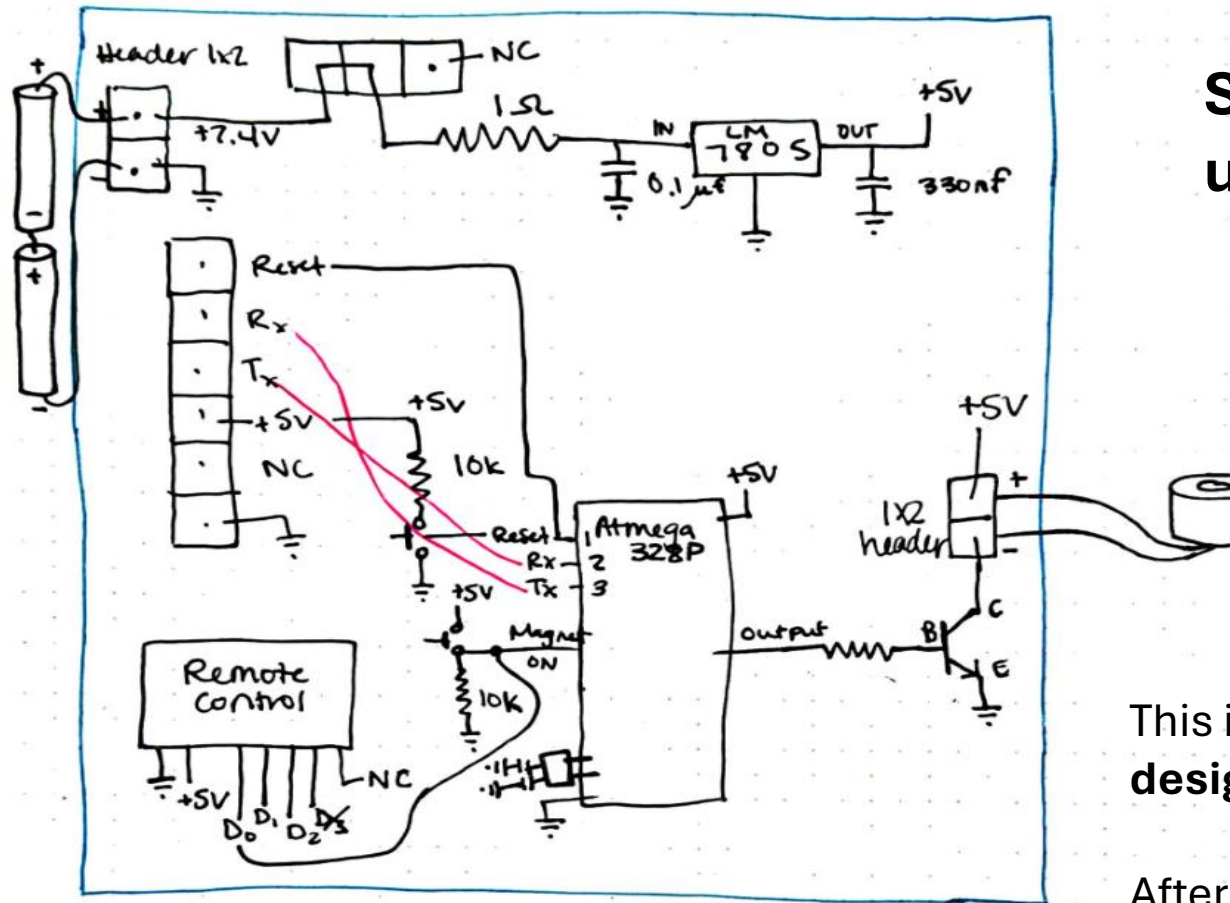


### Bottom Image of a PCB



Images courtesy of OSH PARK.com





**Step 1) Paper Design>>>>>>>will be used to create a Schematic Design**

This is the first step in creating a pcb, referred to as a **paper design**.

After learning and understanding the basics of how circuit functions , began the process of creating circuit board (pcb) .

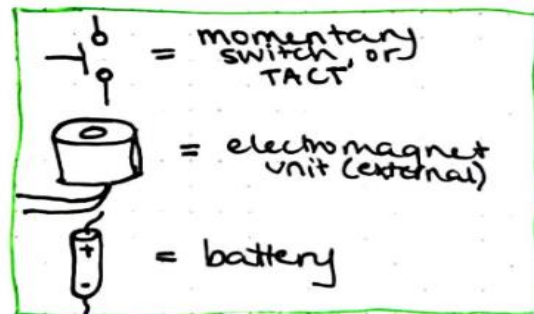
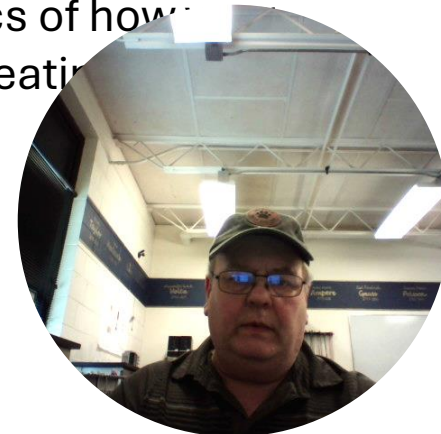
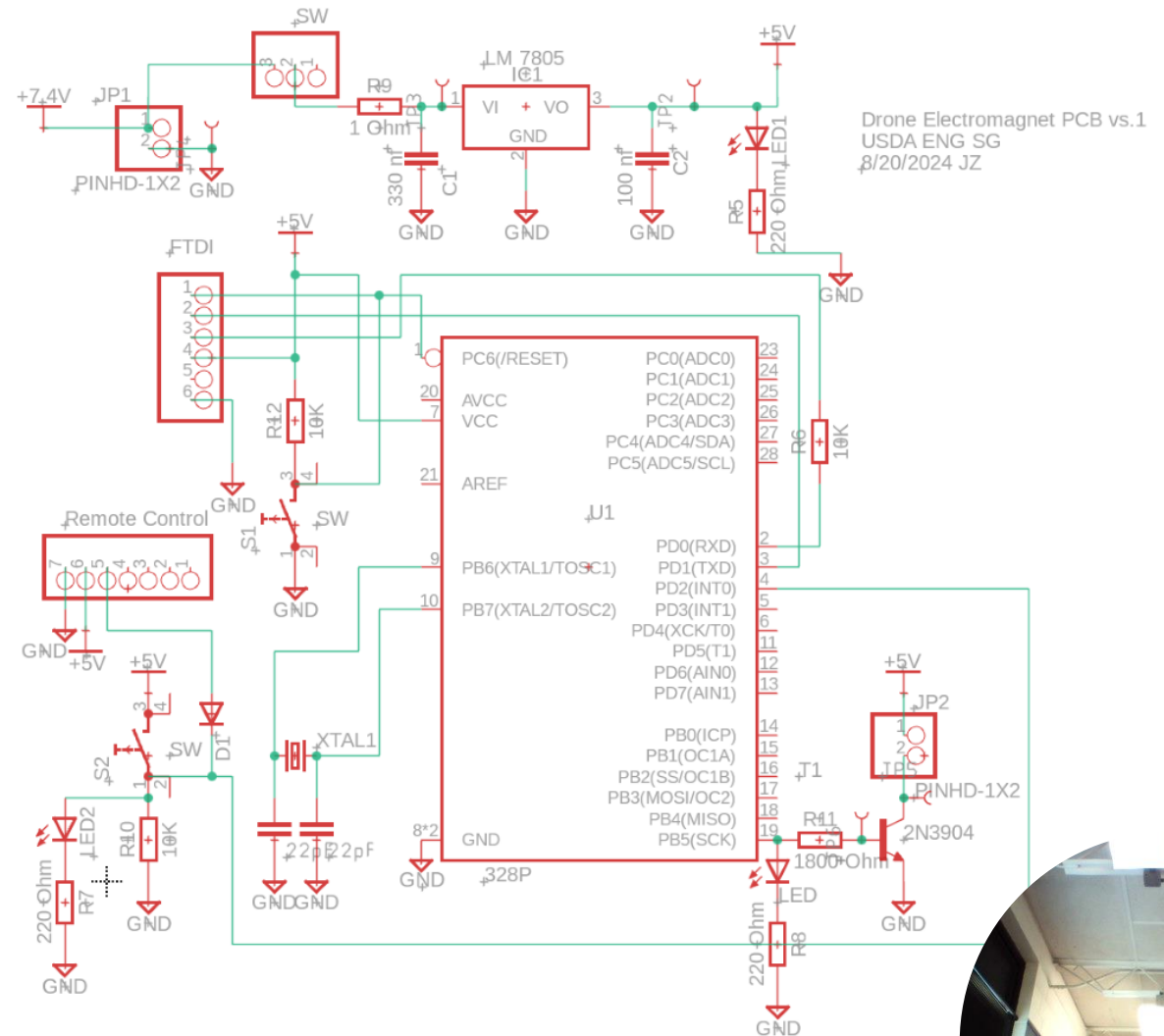


Image Courtesy of JZ



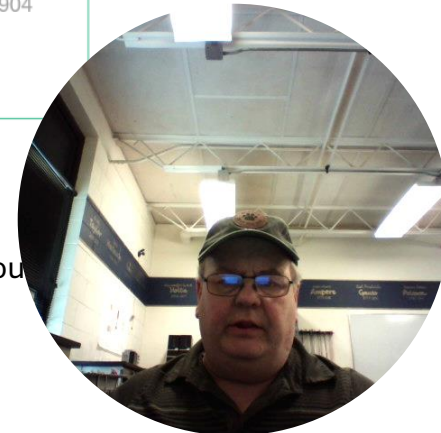
## Step 2) Paper Schematic converted to Fusion360 Schematic Design

The next step in the process was to transfer this paper schematic to the Fusion 360 Design Platform. This was done by selecting each component individually and verifying that it is correct for this project, then placing and connecting them in the correct locations to get a working circuit. It should resemble the schematic drawn in the previous step, but be a cleaner representation of the circuit itself. It is important to follow each of the PCB requirements to ensure your finished product works correctly and can be tested.

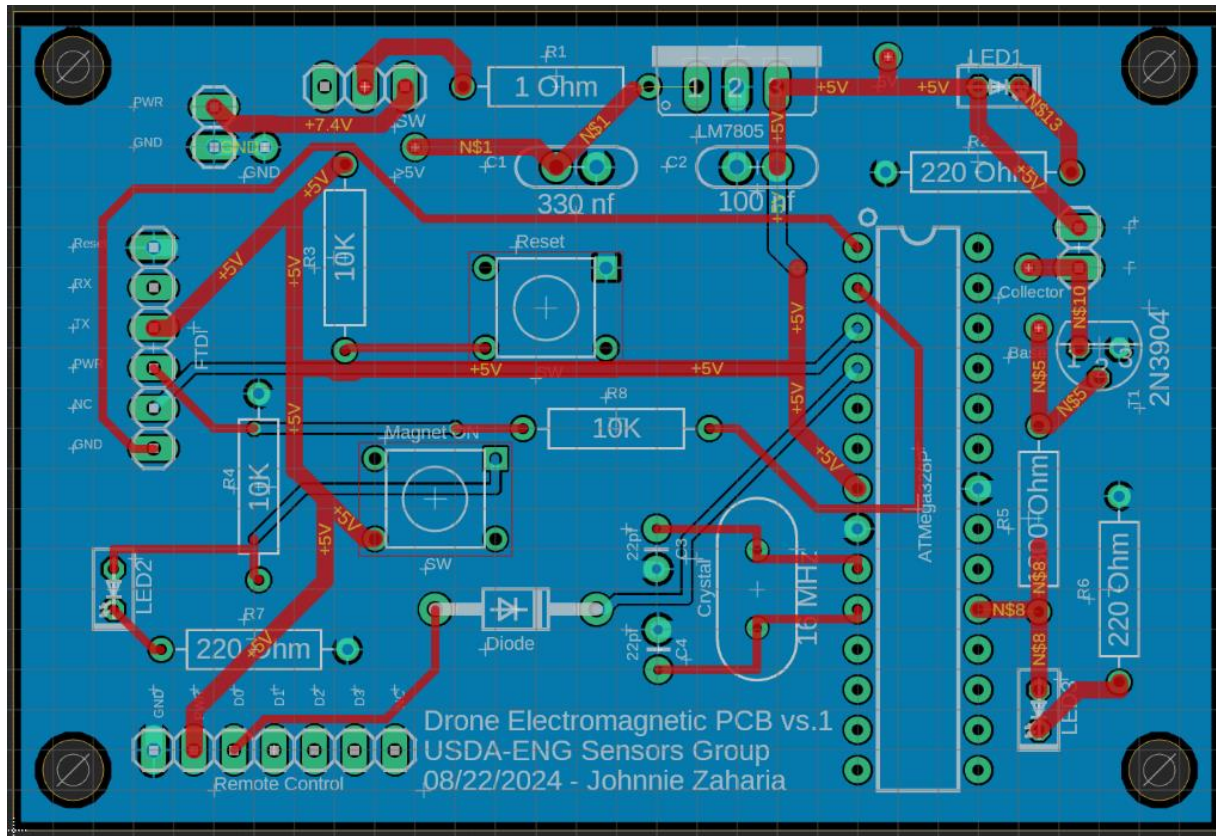


Fusion360 Schematic

Images Cou

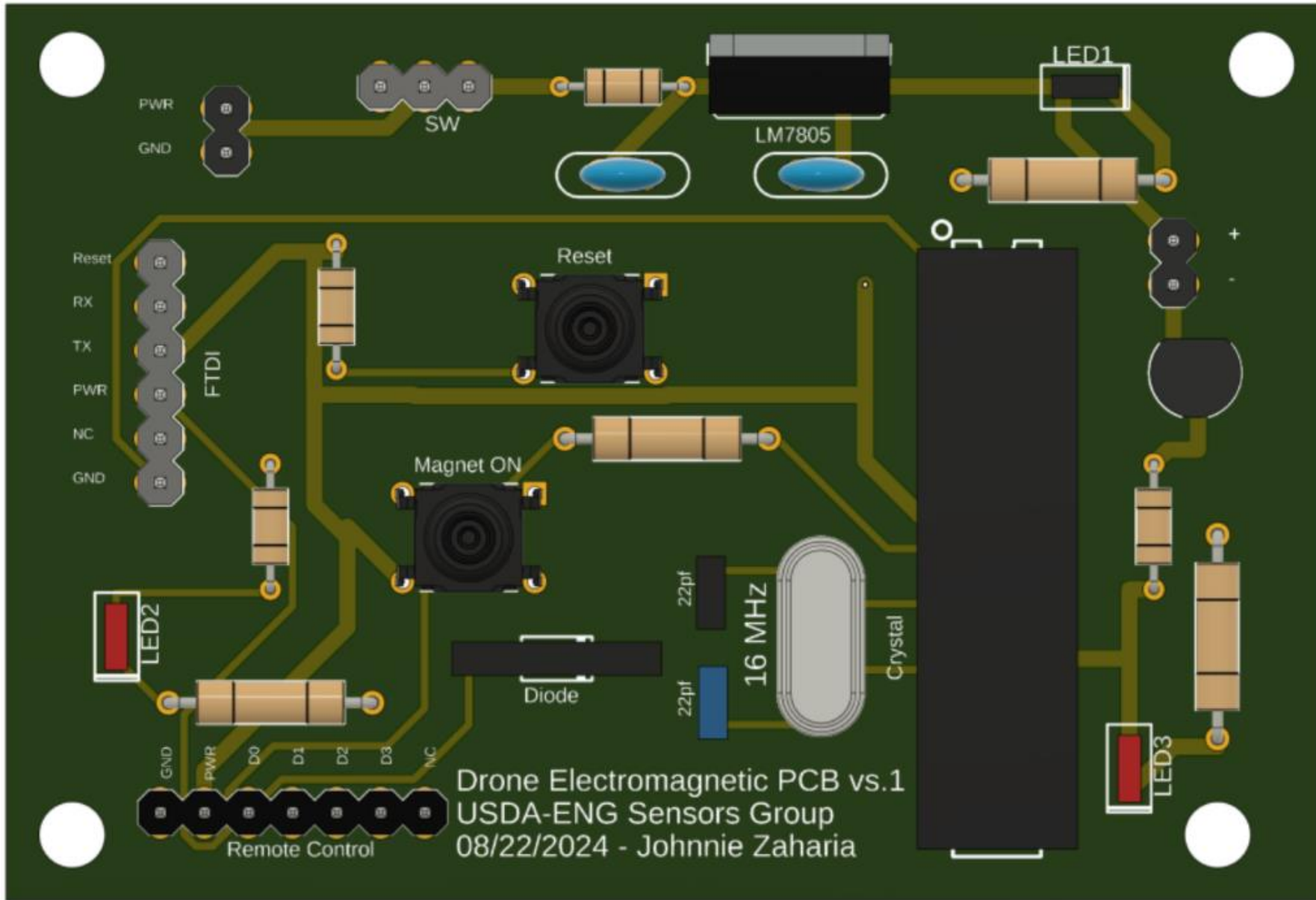






Once the schematic is verified and correct, you can move on to creating a PCB document and arranging your elements. Here is where it matters that you have chosen the correct components, because they need to fit into the designated space correctly so your pcb can be properly arranged. You will arrange the components in a compact and space-saving manner that is logical regarding the necessary wire connections between components (start with 3"x5" and work your way smaller). Then, once each of the pieces are where you want them, you will route the wires/connections (known as nets); this can be done manually, or with autorouter and then tweaked to your satisfaction. It is important that the diameter of the nets that are used for power or ground is wider (40mm) compared to the rest of the nets (20mm). Try to arrange the wires and utilize vias (transfer points from front to back of the board or vice versa) in order to prevent the circuit from shorting or encountering errors when it runs.





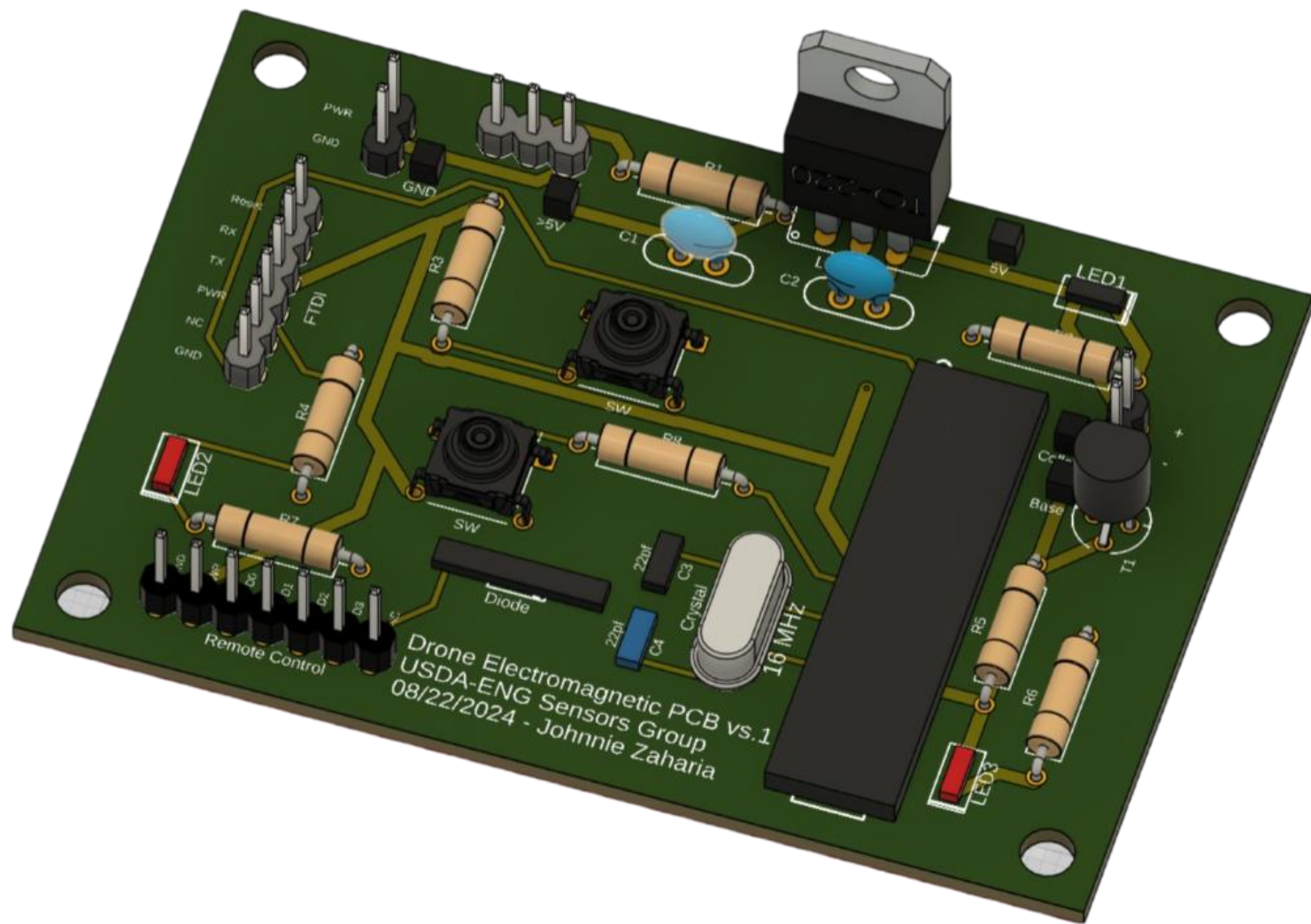
The final step is to push the PCB design to 3D to ensure the proper design. Then, it might be ordered and tweaked again. If more ordering is necessary that will be done until the product is correct for the intended purposes, when it will finally be tested and put to use on the applicable project. The 3D PCB design is visible below.

Image of PCB design 2D



Image courtesy of JZ





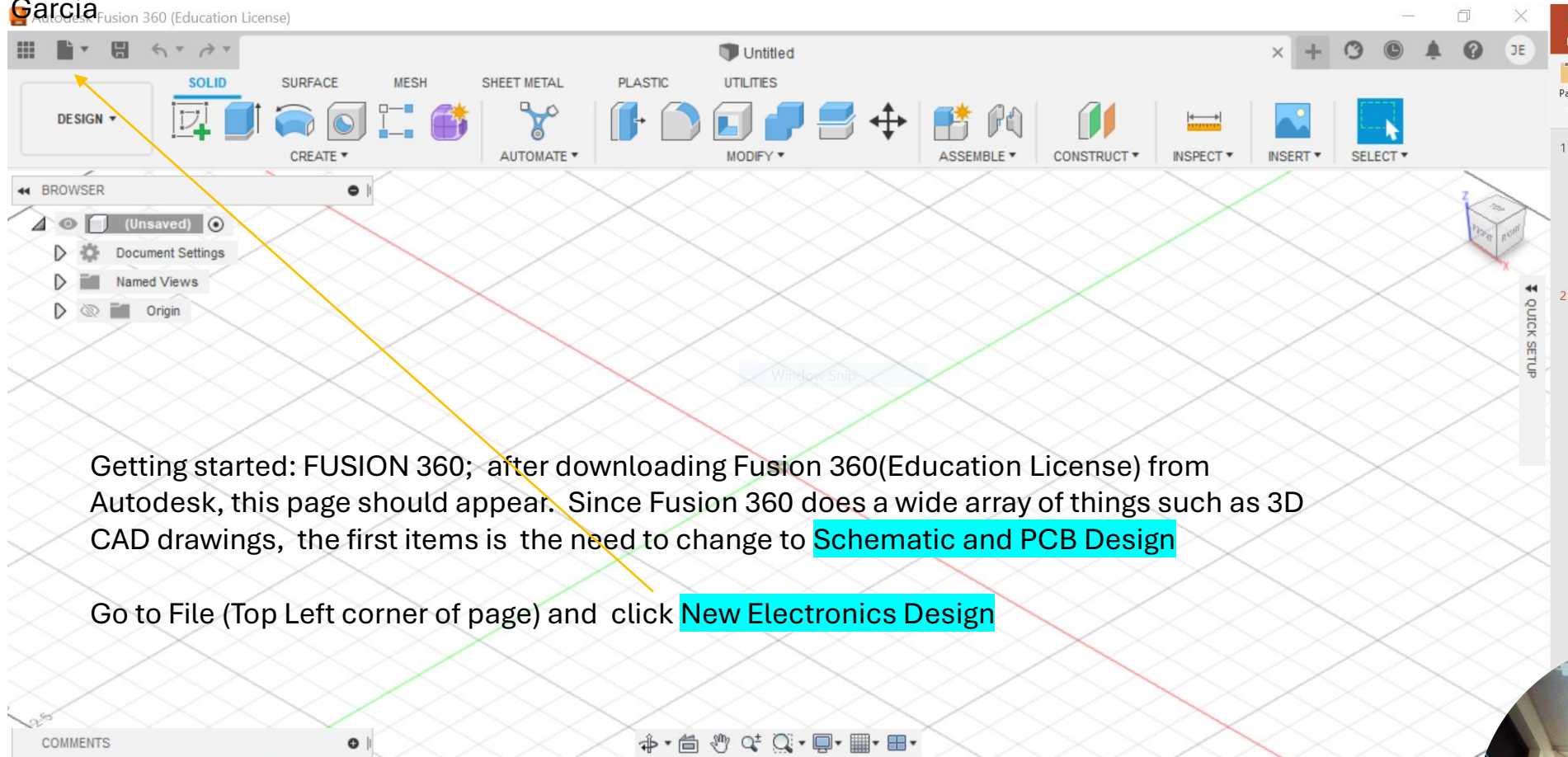
PCB Design Image after being pushed to 3D. Once populated (all components soldered in place) the board should look very close to this. You may order boards in Green, yellow, black. White, Red or Blue.

Image courtesy of JZ



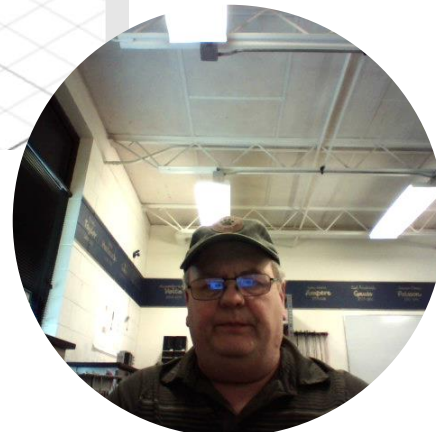
[https://www.youtube.com/watch?v=\\_jgUZeBiusw&list=PLmA\\_xUT-8UIL80Xm8Gxz98YNum3I9Glnr](https://www.youtube.com/watch?v=_jgUZeBiusw&list=PLmA_xUT-8UIL80Xm8Gxz98YNum3I9Glnr) by George

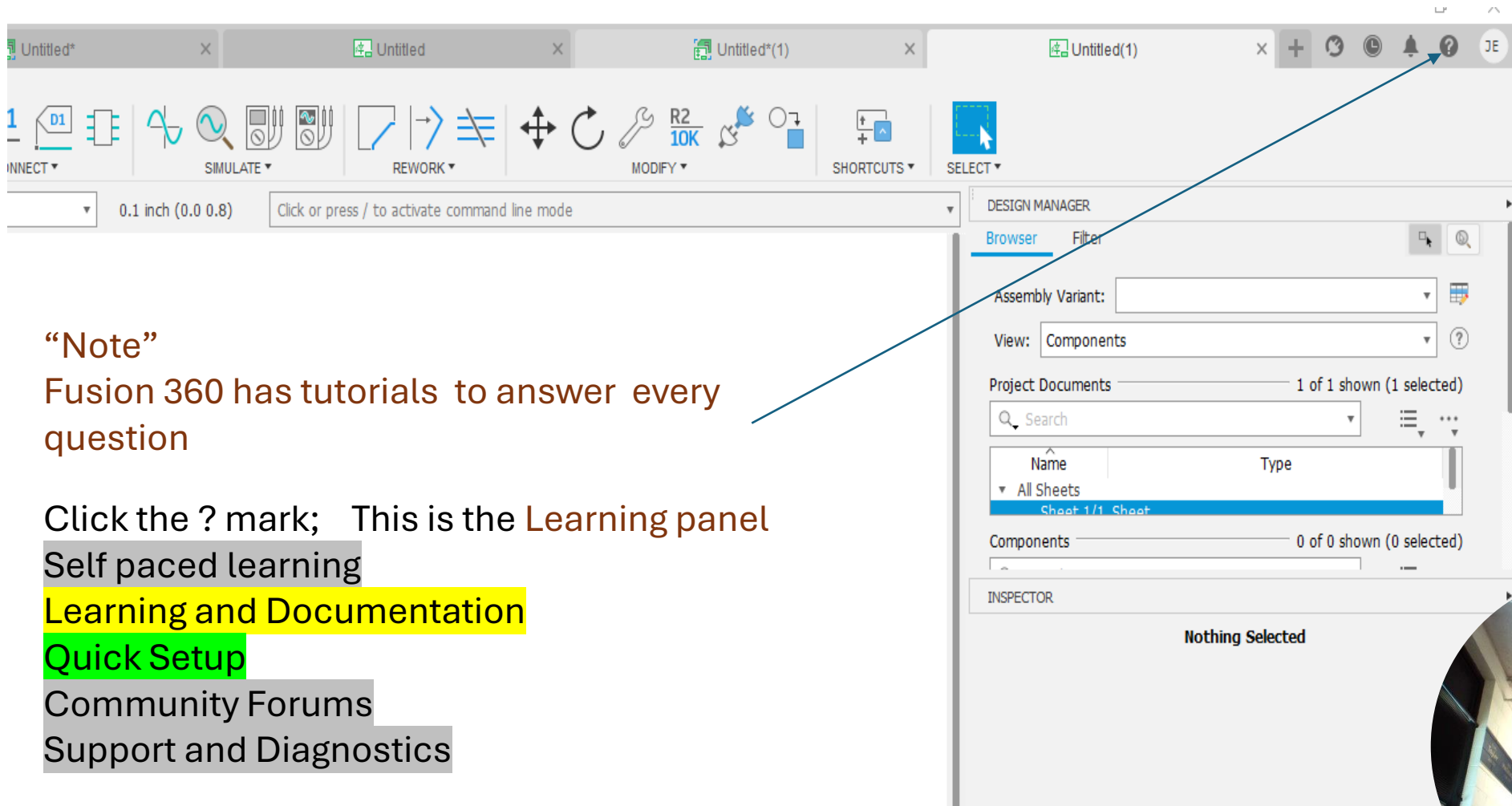
Garcia



Getting started: FUSION 360; after downloading Fusion 360(Education License) from Autodesk, this page should appear. Since Fusion 360 does a wide array of things such as 3D CAD drawings, the first items is the need to change to **Schematic and PCB Design**

Go to File (Top Left corner of page) and click **New Electronics Design**





“Note”

Fusion 360 has tutorials to answer every question

Click the ? mark; This is the Learning panel

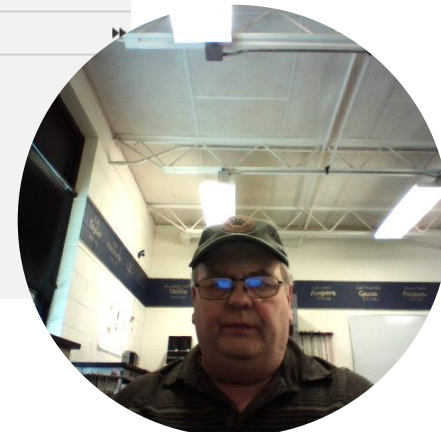
Self paced learning

Learning and Documentation

Quick Setup

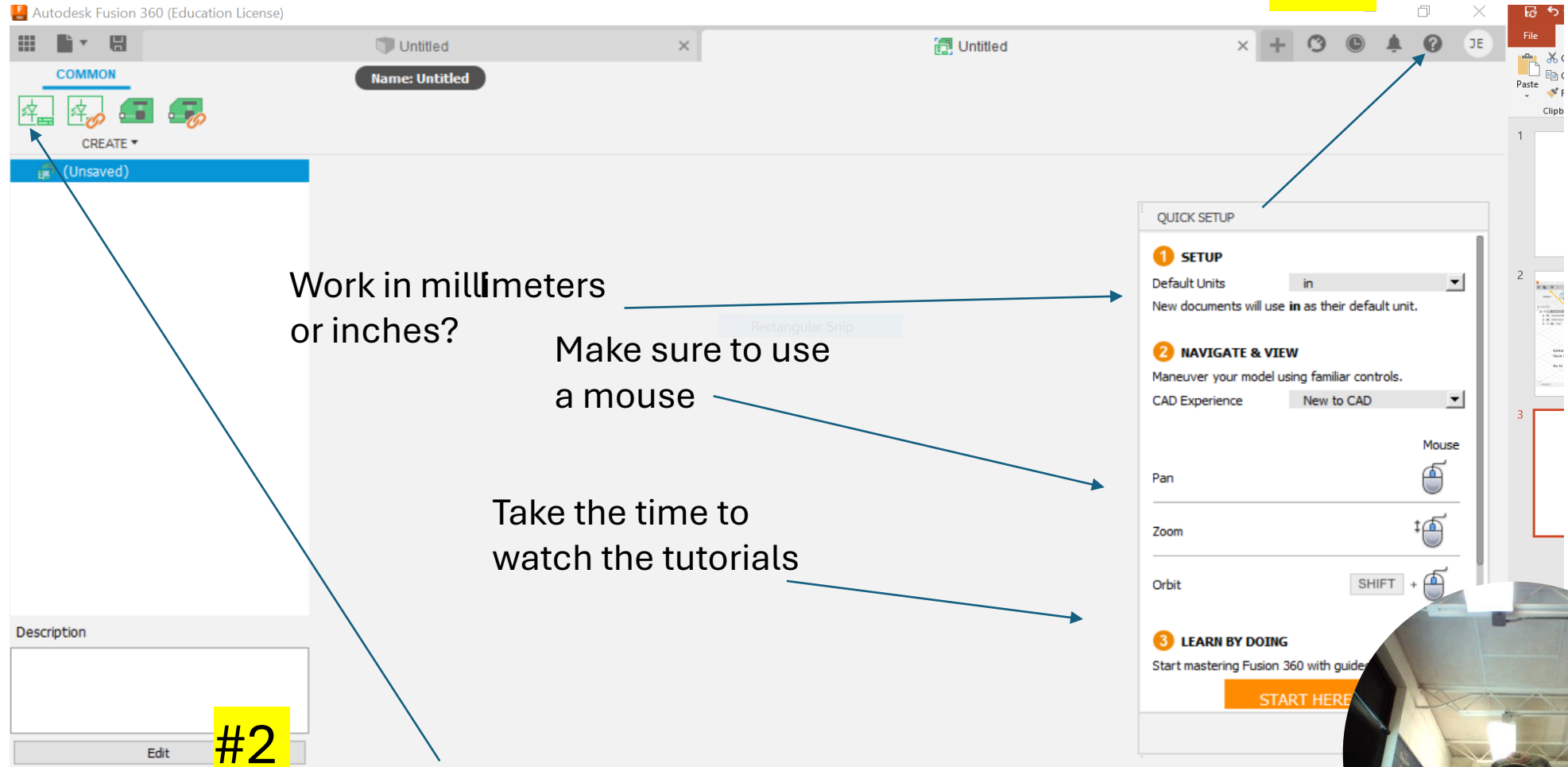
Community Forums

Support and Diagnostics



This page opens up to start a schematic

#1  
Note



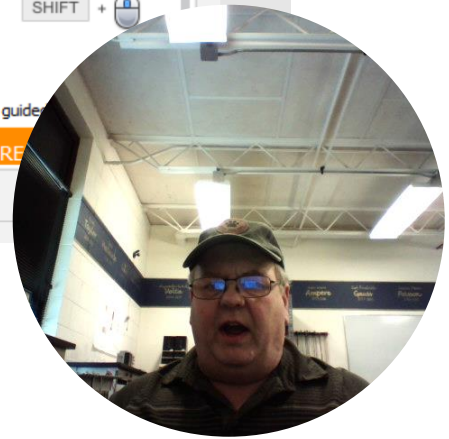
Work in millimeters  
or inches?

Make sure to use  
a mouse

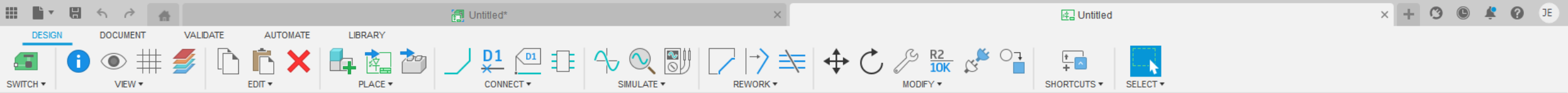
Take the time to  
watch the tutorials

#2  
Note

To create a  
schematic





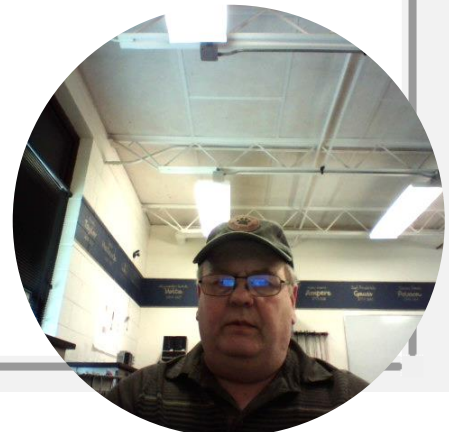


91 Nets

0.1 inch (-2.2 2.4)

Click or press / to activate command line mode

**Note1)** grid settings/ leave at 0.1”  
or change to 2.54mm as this is  
universal breadboard pitch- Pitch is  
the distance between the IC pins



DESIGN DOCUMENT VALIDATE AUTOMATE LIBRARY

SWITCH VIEW EDIT PLACE CONNECT SIMULATE REWORK MODIFY SHORTCUTS SELECT

91 Nets 0.1 inch (-1.4 1.6) Click or press / to activate command line mode

PLACE COMPONENTS

All Libraries

Filter...

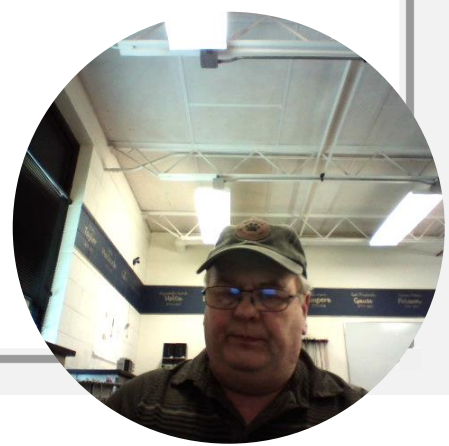
Compon...	Library	Variant
*1458	line...	D
*3080	line...	D
*317	line...	EMP
*337	line...	IMP
*4558	line...	D
*555	line...	D
*555	IC_...	_TSSOP8
*556	line...	D
*556	IC_...	_SOIC
*565H	line...	
*567	line...	D
*723	line...	D
*741	line...	D
*748	line...	D
+1.2V	Pow...	
+1.8V	Pow...	
+12V	Pow...	
+2.5V	Pow...	
+3.3V	Pow...	
+5V	Pow...	

2605 Components

Select components to place

To add parts, click place

Note the Library ICON!

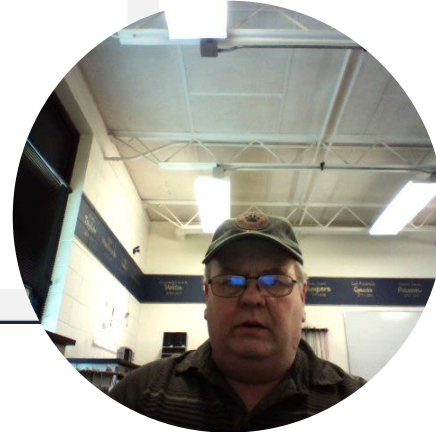


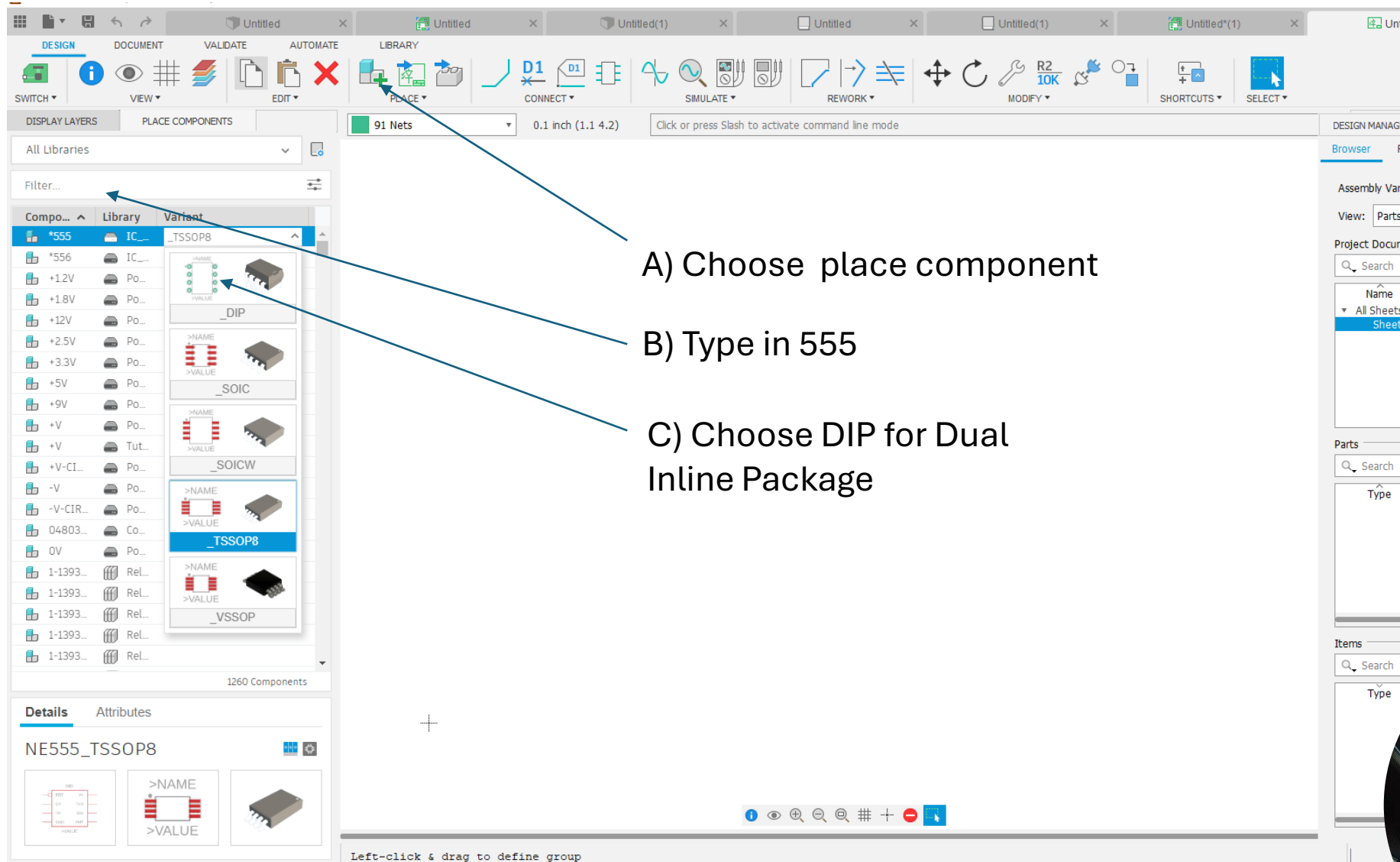
[https://www.youtube.com/watch?v=\\_jgUZeBiusw&list=PLmA\\_xUT-8UIL80Xm8Gxz98YNum3I9Glnr](https://www.youtube.com/watch?v=_jgUZeBiusw&list=PLmA_xUT-8UIL80Xm8Gxz98YNum3I9Glnr)  
Getting started by George Garcia of Fusion 360

**A) Open Library Manager**

**B) Library Manager shows- use the filters. Libraries in Blue are already open**

Library	Folder Name	Version	In Use
19inch	Eagle Pcb	3	<input type="checkbox"/>
40xx	Eagle Pcb	7	<input type="checkbox"/>
41xx	Eagle Pcb	3	<input type="checkbox"/>
45xx	Eagle Pcb	6	<input type="checkbox"/>
52101-101-REV-A_v16	Eagle Pcb	1	<input type="checkbox"/>
74ac-logic	Eagle Pcb	4	<input checked="" type="checkbox"/>
74ttl-din	Eagle Pcb	6	<input checked="" type="checkbox"/>
74xx-eu	Eagle Pcb	5	<input type="checkbox"/>
74xx-little-de	Eagle Pcb	6	<input type="checkbox"/>
74xx-little-us	Eagle Pcb	6	<input type="checkbox"/>
74xx-us	Eagle Pcb	3	<input checked="" type="checkbox"/>
751xx	Eagle Pcb	6	<input type="checkbox"/>
Airquality_Sensor	Eagle Pcb	1	<input type="checkbox"/>
Audio Connectors	Hetal @pcblayout	1	<input type="checkbox"/>
Audio-Devices	Fusion Electronics	3	<input checked="" type="checkbox"/>
Audio-Devices			<input checked="" type="checkbox"/>
Battery Chargers	Hetal @pcblayout	3	<input type="checkbox"/>
Battery Holder	Hetal @pcblayout	1	<input type="checkbox"/>
Battery_Holder	Fusion Electronics	4	<input type="checkbox"/>
Battery_Holder			<input checked="" type="checkbox"/>
BeagleBone_Black_...	Eagle Pcb	14	<input type="checkbox"/>
BeagleBone Blue R3	Eagle Pcb	49	<input type="checkbox"/>

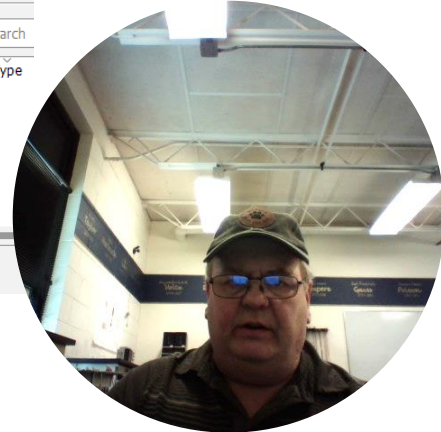




A) Choose place component

B) Type in 555

C) Choose DIP for Dual Inline Package

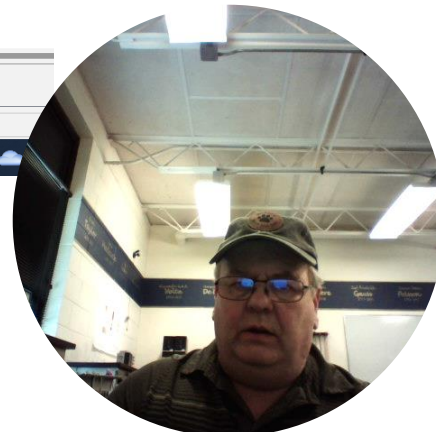


The screenshot displays the Altium Designer software interface. The top menu bar includes options like DESIGN, DOCUMENT, VALIDATE, AUTOMATE, LIBRARY, PLACE, CONNECT, SIMULATE, REWORK, MODIFY, SHORTCUTS, and SELECT. Below the menu bar, there are toolbars for DISPLAY LAYERS, PLACE COMPONENTS, and a status bar showing '91 Nets' and '0.1 inch (1.0 4.3)'. The left sidebar shows a component library with a search filter. The main workspace shows a component symbol for 'ICM7555\_DIP' with pins labeled RST, CV, TR, GND, V+, THR, DIS, and OUT. The right sidebar shows the DESIGN MANAGER with sections for Assembly Variant, Project Documents, Parts, and Items. The Windows taskbar is visible at the bottom.

**A) Will change to DIP package**

**B) Select drag and Drop on screen**

Left-click & drag to define group



DESIGN DOCUMENT VALIDATE AUTOMATE LIBRARY

SWITCH VIEW EDIT PLACE CONNECT SIMULATE REWORK MODIFY SHORTCUTS SELECT

91 Nets 0.1 inch (0.2 4.2) Click or press Slash to activate command line mode

DESIGN MANAGER  
Browser Filter  
Assembly Variant:  
View: Parts  
Project Documents  
Search  
Name  
All Sheets  
Sheet 1/1  
Parts  
Search  
Type Name  
Part IC1  
Items  
Search  
Type


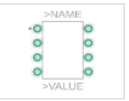

Filter...

Compo...	Library	Variant
*555	IC...	_DIP
*556	IC...	_SOIC
+1.2V	Po...	
+1.8V	Po...	
+12V	Po...	
+2.5V	Po...	
+3.3V	Po...	
+5V	Po...	
+9V	Po...	
+V	Po...	
+V	Tut...	
+V-CI...	Po...	
-V	Po...	
-V-CIR...	Po...	
04803...	Co...	
0V	Po...	
1-1393...	Rel...	
1-1393...	Rel...	
1-1393...	Rel...	
1-1393...	Rel...	
1-1393...	Rel...	
1-1393...	Rel...	

1260 Components

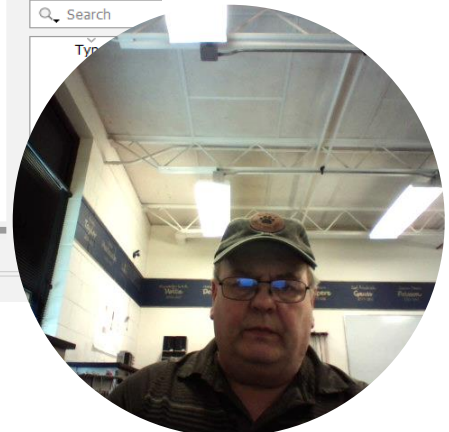
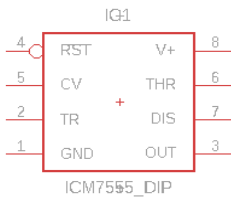
Details Attributes

ICM7555\_DIP



Left-click & drag to define group

A) Add resistors in your circuit- Type Resistors





The screenshot displays the Altium Designer software interface. At the top is a ribbon menu with tabs for DESIGN, DOCUMENT, VALIDATE, AUTOMATE, and LIBRARY. Below the ribbon are several toolbars including DISPLAY LAYERS, PLACE COMPONENTS, and a command line. The main workspace shows a component being placed, with a pinout diagram for ICM7555\_DIP. The component has 8 pins: 4 (RST), 5 (CV), 2 (TR), 1 (GND), 8 (V+), 6 (THR), 7 (DIS), and 3 (OUT). A yellow box highlights the text 'A) Type in resistors' with an arrow pointing to the search bar in the 'PLACE COMPONENTS' panel, which contains the text 'Resistors'. Another yellow box highlights the text 'B) Choose Style- Value is not important at this time, value can be changed' with an arrow pointing to the 'Variant' column in the component list. The 'Details' panel at the bottom left shows the component name 'ICM7555\_DIP' and its pinout diagram. The 'DESIGN MANAGER' panel on the right shows the project structure, including 'All Sheets' and 'Sheet 1/1 Sheet'. A circular inset image in the bottom right corner shows a man wearing a cap and glasses, looking at the camera.

DESIGN DOCUMENT VALIDATE AUTOMATE LIBRARY

SWITCH VIEW EDIT PLACE CONNECT SIMULATE REWORK MODIFY SHORTCUTS SELECT

DISPLAY LAYERS PLACE COMPONENTS 91 Nets 0.1 inch (5.9 3.7) Click or press Slash to activate command line mode

All Libraries

Resistors

Compon...	Library	Variant
1-21760...	Res...	
162376...	Res...	
162414...	Res...	
162415...	Res...	
163001...	Res...	
2-1623...	Res...	
7-21762...	Res...	
74*746	74x...	N
74*747	74x...	N
9-1879...	Res...	

10 Components

Details Attributes

ICM7555\_DIP

Left-click & drag to define group

DESIGN MANAGER

Browser Filter

Assembly Variant:

View: Parts

Project Documents

Name

All Sheets

Sheet 1/1 Sheet

Parts

Type	Name	Value
Part	IC1	ICM7555_I

Items

Type

Resistor

Filter...

Compon...	Library	Variant
R-US	Res...	AXIAL-7.2MM-PITCH
R	Res...	AXIAL-11.7MM-PITCH
PV36	Res...	AXIAL-7.2MM-PITCH
		CHIP-0402(1005-M...
		CHIP-0603(1608-M...
		CHIP-0805(2012-M...
		CHIP-1206(3216-M...
		CHIP-1210(3225-M...
		CHIP-2010(5025-M...

Details Attributes

R-US\_AXIAL-7.2M

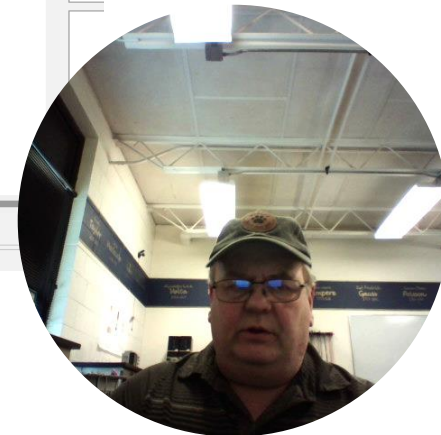
GS1  
RESISTOR  
>VALUE

ICM7555\_DIP

4 RST V+ 8  
5 CV THR 6  
2 TR + DIS 7  
1 GND OUT 3

Left-click & drag to define group

- A) Choose Resistor
- B) Under filter you can choose between Variants
- C) Variant is size & pitch, through hole or SMD choose Axial-7.2mm (highlighted in green)



Never Delete these- always need three

A) Select and insert Resistors wherever required

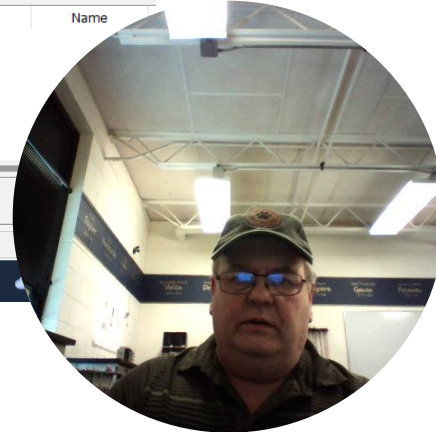
B) Double right click to rotate components

Move Command

Change Value Command

Left-click & drag to define group

Type	Name	Value
Part	IC1	ICM7555_DIP
Part	R1	Resistor
Part	R2	Resistor
Part	R3	Resistor
Part	R4	Resistor
Part	R5	Resistor
Part	R6	Resistor



**B) Connect your NETS**

**A) Add Power and Gnd Symbols**

Left-click to select object to move (Ctrl+right-click to move group)

DESIGN MANAGER

Browser Filter

Assembly Variant: [ ]

View: Parts

Project Documents 1 of 1 shown (1 selected)

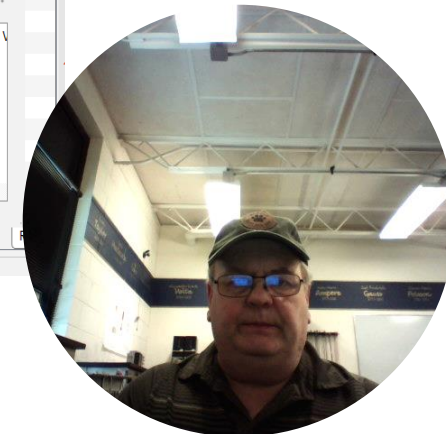
Name	Type
All Sheets	
Sheet 1/1 Sheet	

Parts 13 of 13 shown (0 selected)

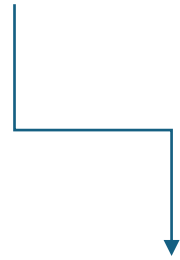
Type	Name	Value	Library	Devic
Part IC1	ICM7555_DIP	IC_Clock-Timing	_DIP (*)	
Part R1		Resistor	AXIAL-7	
Part R2		Resistor	AXIAL-7	
Part R3		Resistor	AXIAL-7	
Part R4		Resistor	AXIAL-7	
Part R5		Resistor	AXIAL-7	
Part R6		Resistor	AXIAL-7	

Items 0 of 0 shown (0 selected)

Type	Name	Parent
------	------	--------

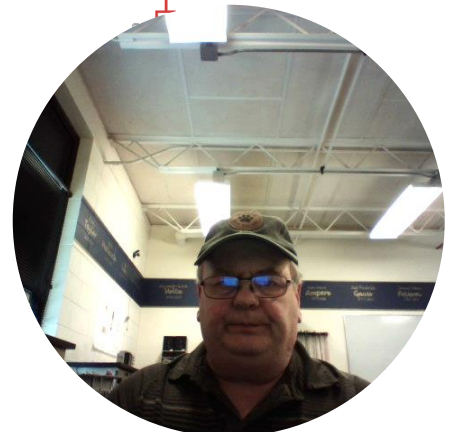


# Once Schematic completed switch to PCB Design



Component	Library	Variant
*555	IC_Clo...	_TSSOP8
*556	IC_Clo...	_SOIC
+12V	Power...	

1370 Components



DESIGN DOCUMENT VALIDATE AUTOMATE LIBRARY  
 WITCH VIEW ASSEMBLY VARIANT OUTPUT DRAW ATTRIBUTES SELECT  
 PLACE COMPONENTS 91 Nets 0.1 inch (-1.0 2.5) Click or press / to activate command line mode

Example libraries are now on library.io. Please swap libraries in designs that used local examples. [Click here](#) for more info.  
 Don't show again.

Tutorial - Fusion 360

Filter...

Component ^	Library	Variant
+V	Tutorial - ...	
C	Tutorial - ...	RADIAL-
GND	Tutorial - ...	
GND-BAR	Tutorial - ...	
GND-CIRCLE	Tutorial - ...	
JSTPH2	Tutorial - ...	
LED_CHIP	Tutorial - ...	RED-2
LM555N	Tutorial - ...	S-PDS
R-US	Tutorial - ...	CHIP-
VCC-BAR	Tutorial - ...	
VCC-CIRCLE	Tutorial - ...	
VDD-CIRCLE	Tutorial - ...	
VLP-300-R/F	Tutorial - ...	
VSS	Tutorial - ...	

16 Components

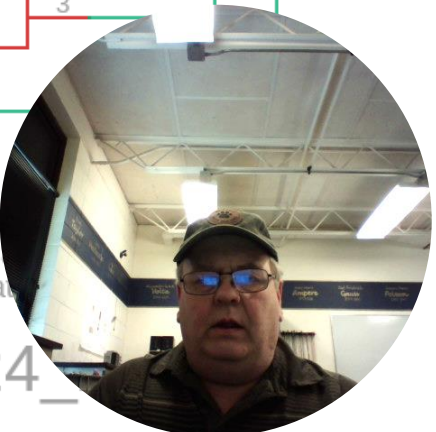
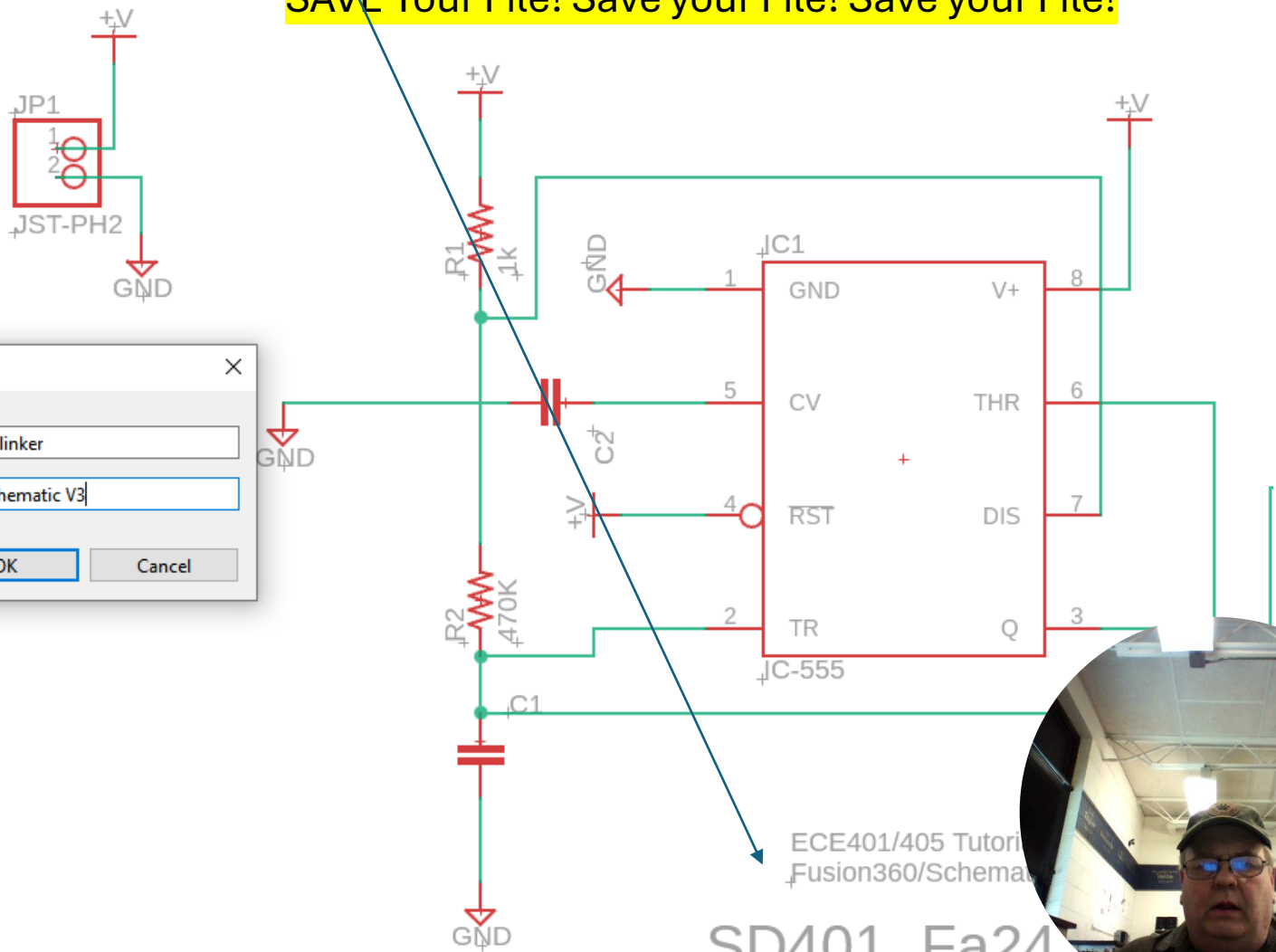
**SAVE Your File! Save your File! Save your File!**

Save

Version Description SD401Blinker

Milestone  Schematic V3

OK Cancel



ECE401/405 Tutorial  
Fusion360/Schematic  
SD401\_Fa24\_



END PART ONE

GETTING STARTED with FUSION360