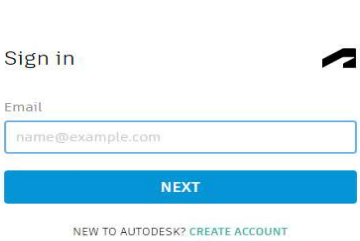
Getting Started: **Step by Step creating a PCB with Fusion 360**



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

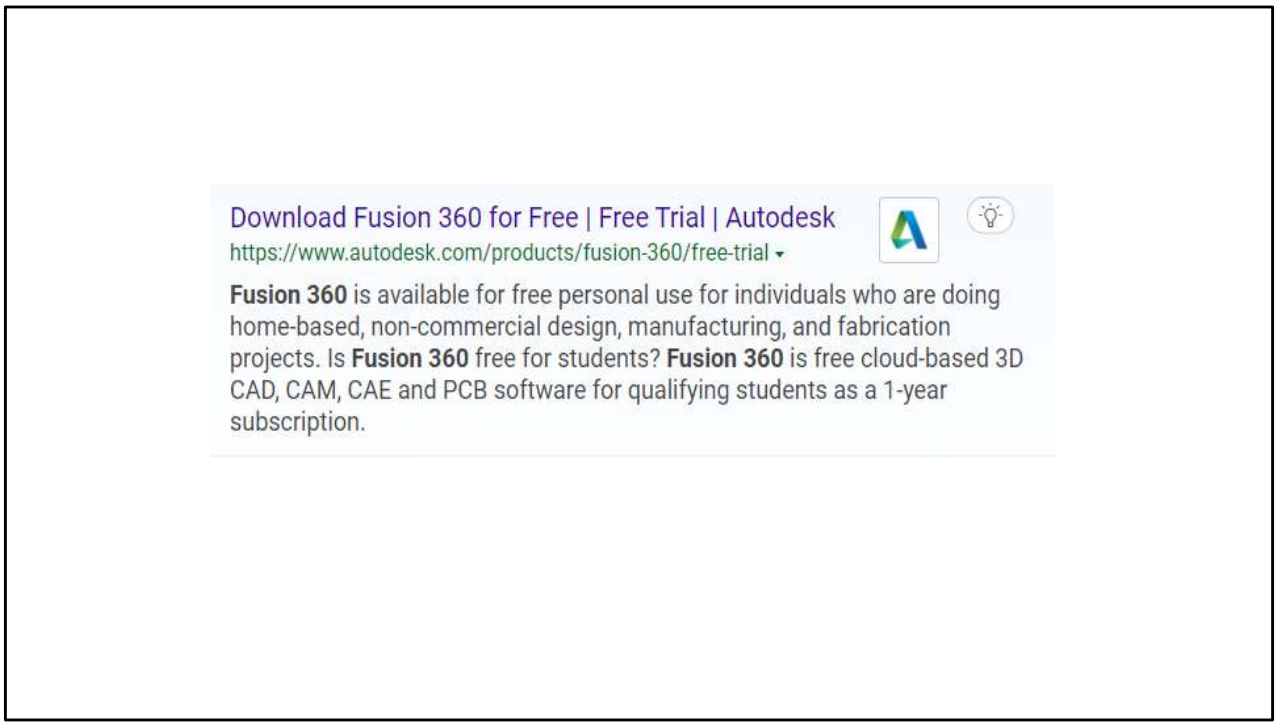
2) https://www.autodesk.com/products/fusion-360/education?AID=10282382&PID=100357191&SID=tuid%3A2982E00743266CAE26CCF26B47266A54&mktvar002=afc_us_deeplink&cjevent=909e6a5fa63211ed8367bd120a1c0e0b&affname=100357191_10282382

This PowerPoint was created to give you a step by step tutorial of how to create a printed circuit board from start to finish. It only shows the highlights of the process as you will have to find some information on your own as needed.
Fusion 360 has a higher learning curve than some other PCB design programs but it does have it's advantages and is used in Industry. It also is a program that let's you do 3D modeling creating files necessary for .stl and .obj files which are used for 3D printing.
#1)To begin with Fusion360 is a free CAD program to all students at NDSU with a ndsu email address
#2) Copy this link or go to Autodesk.com and follow instructions to setup a new account. It is good for one year and can easily be updated for another year still free to the user.

Here is a list of 11 best PCB Design Software:
Altium $330 per month
CircuitMaker Free
Eagle -Free
PCB Web Designer
Dip Trace

Solidworks PCB
Kicad
Easy EDA
Fusion 360 Free one year
PCB Artist- Digi-key- Advanced Circuits

Is Altium better than EAGLE?
Comparing these two software's, Eagle has better options when comes to integrated design or team collaboration. Better options are available in Eagle in case of hardware based project whereas on the other hand, the user interface of Altium is more reliable and powerful.
Altium Circuit Maker is a free PCB design tool. EAGLE offers a free version for personal use. This limited version includes 2 schematic sheets, 2 signal layers, and an 80 cm2 board area. Regular pricing for EAGLE/Fusion 360 is $1,555 (paid every 3 years), $545 annually, or $70 per month.



Another note from Autodesk about Fusion360…

AUTODESK FUSION360

Fusion360 Schematic Design Tutorial

https://www.youtube.com/watch?v=IqwHkB9lsUo

Fusion360 PCB Design Tutorial

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

or

https://www.youtube.com/watch?v=_jgUZeBiusw&list=PLmA_xUT-8UIL80Xm8Gxz98YNum3I9GInr A to Z by George Garcia

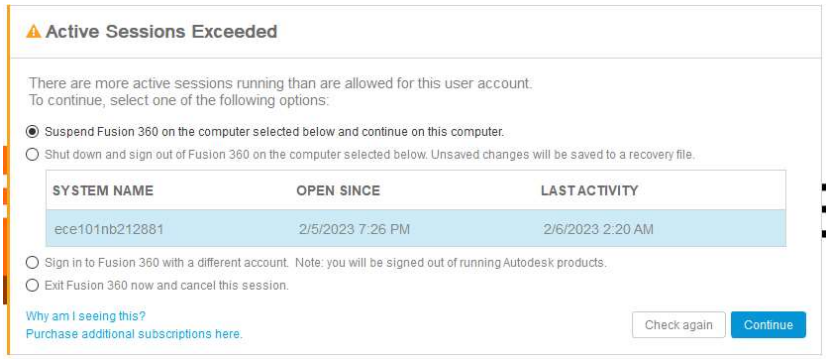
Fusion 360 is more advanced than Upverter, with the extra and more advanced libraries it has a higher level of complexity.

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

There are many more videos on Youtube such as
https://www.youtube.com/watch?v=IqwHkB9lsUo

As Fusion360 is more advanced than other programs such as Upverter, due to the complexity you must look at these design tutorials to get started and for further questions. Fusion360 answers most all questions via a short YouTube video. Highlighted in Blue is the Schematic Design tutorial, highlighted in green is the PCB design tutorial. You need to create a schematic design before you create a PCB design.

One account/ one active session at a time



Just a note on this: One active account at a time, if you should forget to logout on a computer at home or school and try to use another, this usually pops up. It is OK to suspend it, you will not lose any information. Just hit 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

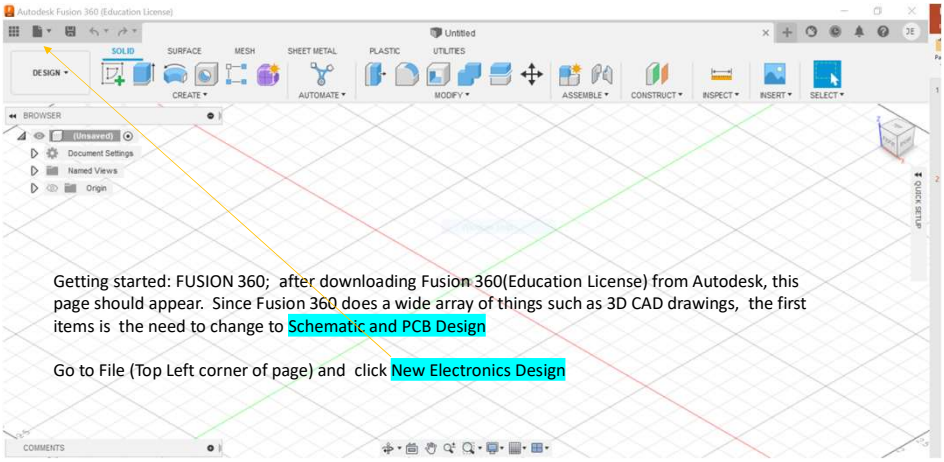
2) Parts list- Libraries- Create your own library (excel works) for complex schematics or it will develop a library for you as your Schematic is created by the parts you choose from within the Fusion 360 parts libraries. Because there are so many type of package designs Creating a known Parts file is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

3) PCB Design- Convert your Schematic to PCB Design

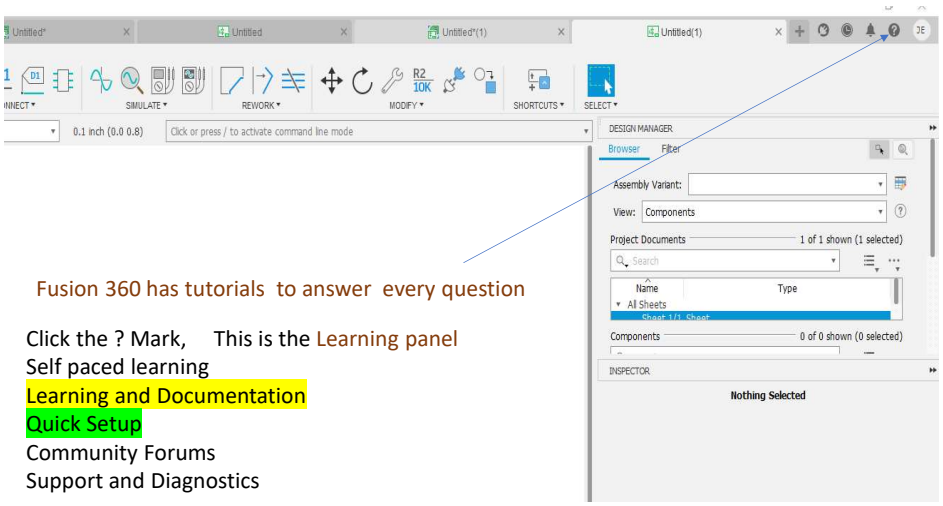
4) Create Gerber files for manufacturing- Drill and Cam files- as in a zip folder- labeled with your SD project #

These are four requirements to make a PCB….. Create a schematic, need a parts list, Create the PCB Design, Create the Gerber's for manufacturing

https://www.youtube.com/watch?v=_jgUZeBiusw&list=PLmA_xUT-8UlL80Xm8Gxz98YNum3I9GInr by George Garcia



To get started and after logging , this page should show up, take a note where the arrows go. As it says, first step is to change to Schematic and PCB Design… click New Electronics Design

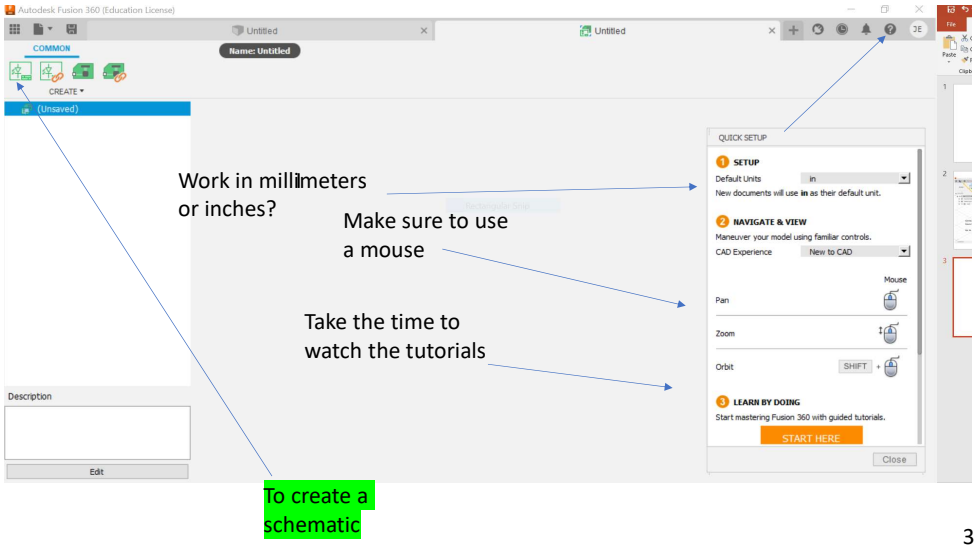


Fusion 360 has tutorials to answer every question

Click the ? Mark, This is the Learning panel
Self paced learning
Learning and Documentation
Quick Setup
Community Forums
Support and Diagnostics

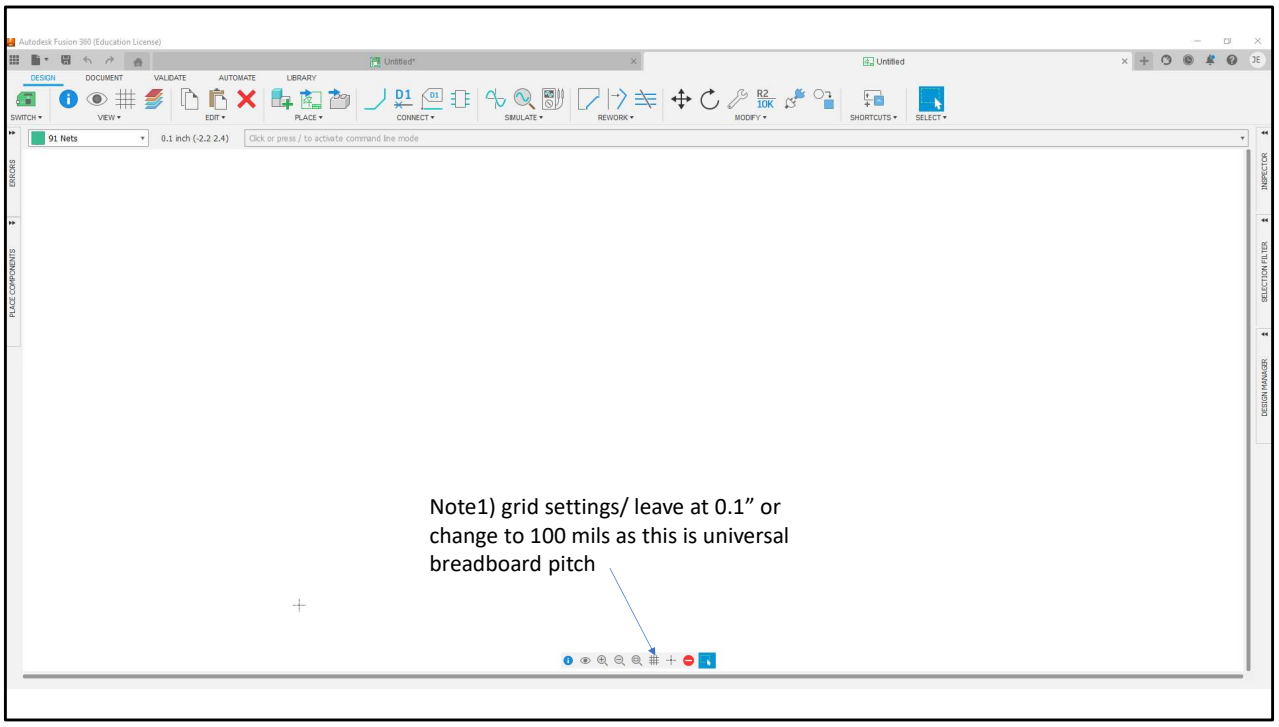
Clicking the ? Will help answer many question. Click on this to bring you to Learning and Documentation, or quick setup

This page opens up to start a schematic

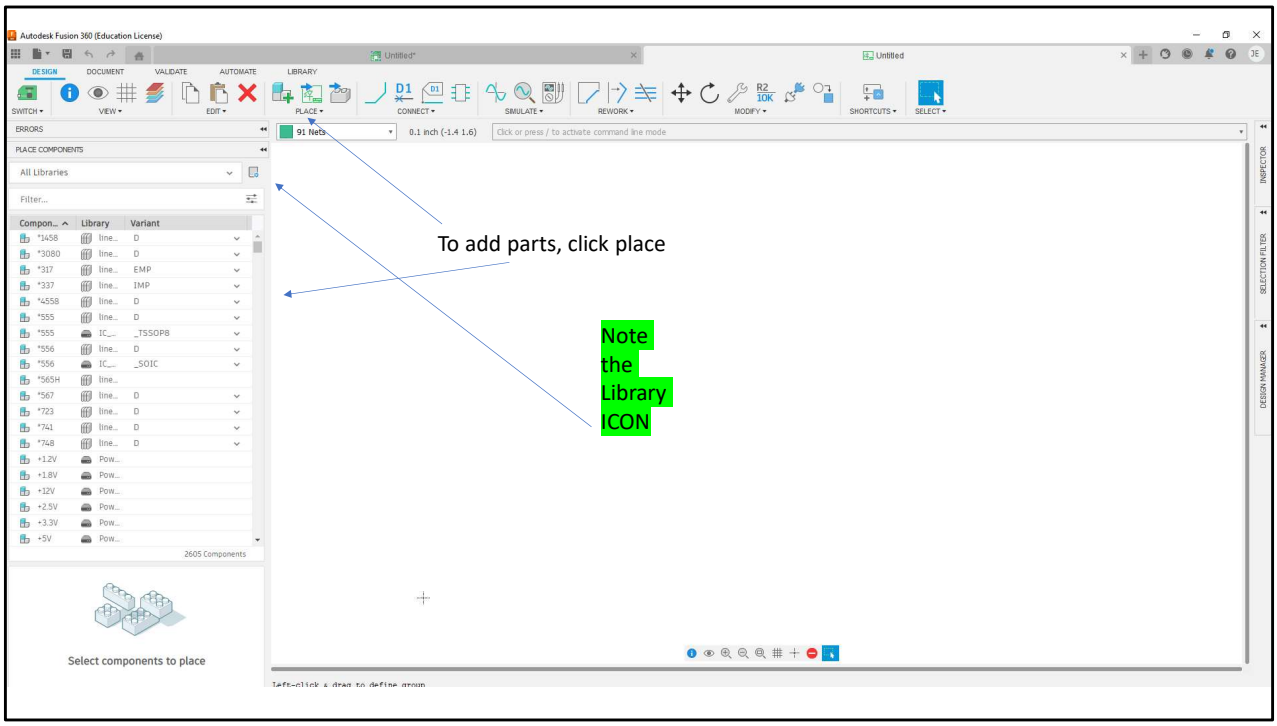


3

. To setup parameters such as working in mm or inches(mils) click the ? mark, click quick setup, this shows the 1) default units , 2) it asks are you ..New to Cad? And how to setup your mouse. "Note" Using Fusion 360 without a mouse is NOT a good idea! 3) Learn by Doing is also very helpful and has guided tutorials. Click and choose Electronics Design (back to the fundamentals) or ECAD Design , … Finally click on Schematic Icon to create a new schematic . Under Common you click the schematic Icon, later anything you put in the schematic can be seen by clicking the green Icon.

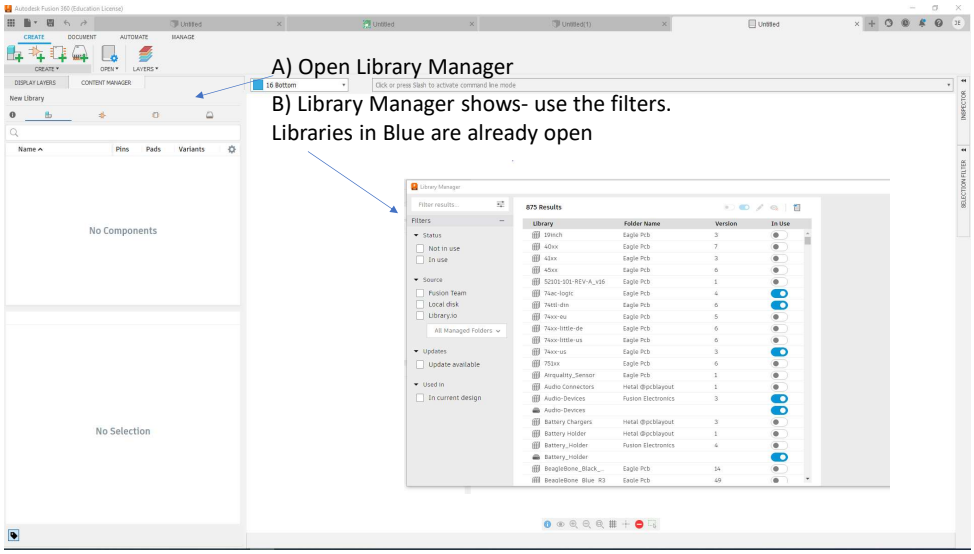


After clicking the Schematic Icon a blank screen shows up. Note 1) Users preference for dots, lines or no display,



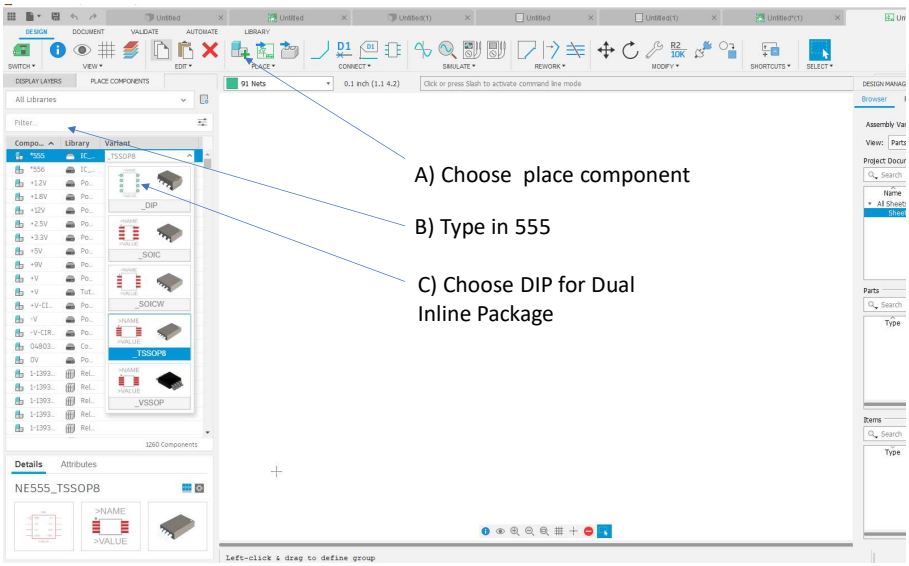
Click the Place under Library tab to add parts, Components and libraries listing shows. Here is where you access all the parts libraries in Fusion360

https://www.youtube.com/watch?v=_jgUZeBiusw&list=PLmA_xUT-8UlL80Xm8Gxz98YNum3I9GInr
Getting started by George Garcia of Fusion 360



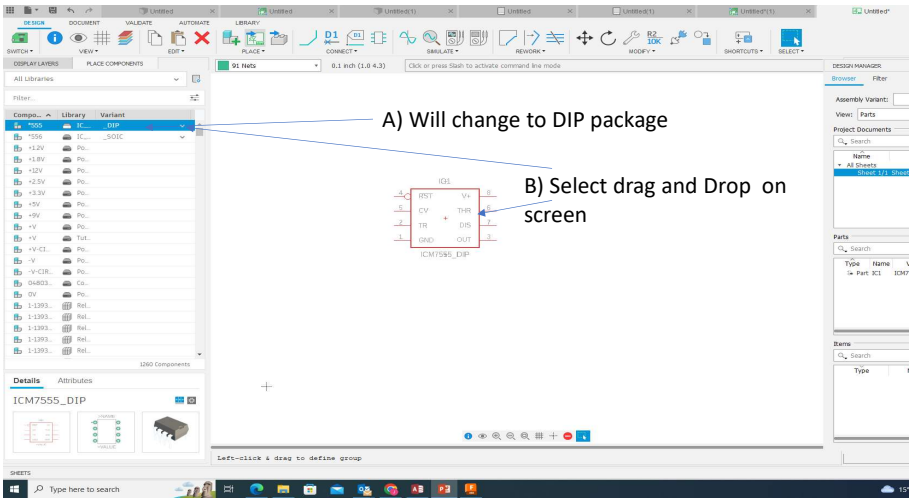
4

After clicking the library icon, all the libraries available to you open up, those in use are blue and already available, if you cannot find what you need you need to search for them. Due to the extensive library this can become challenging, but once found it will save them in a library called “In this Design”

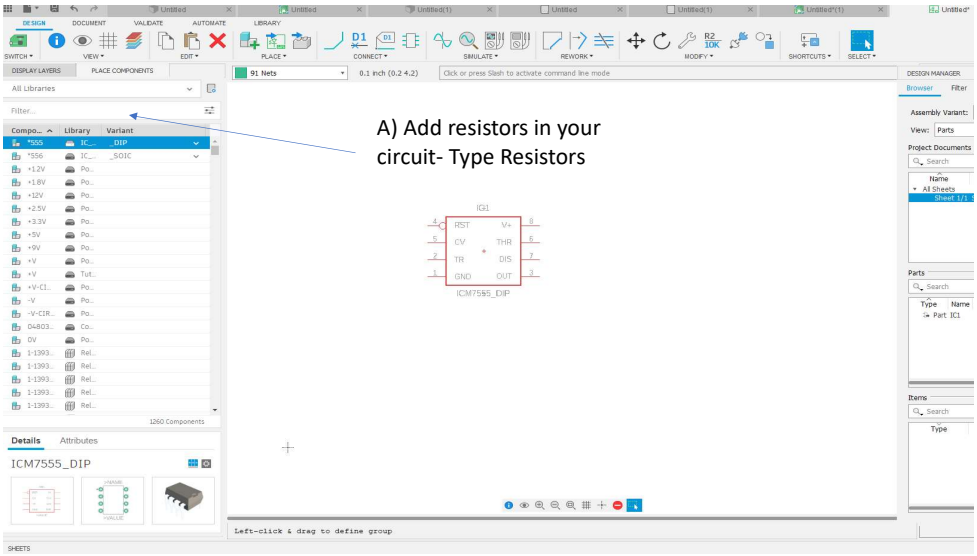


7

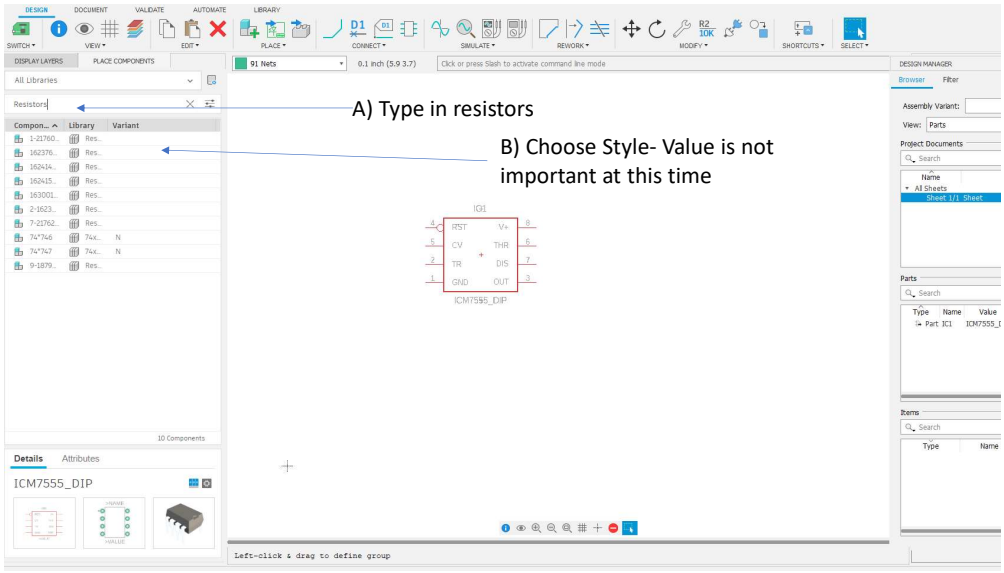
In this case we will create a circuit using a Timer IC (555 Timer) , note the different Variants under this library, choose a thru hole component, select , drag onto screen and drop



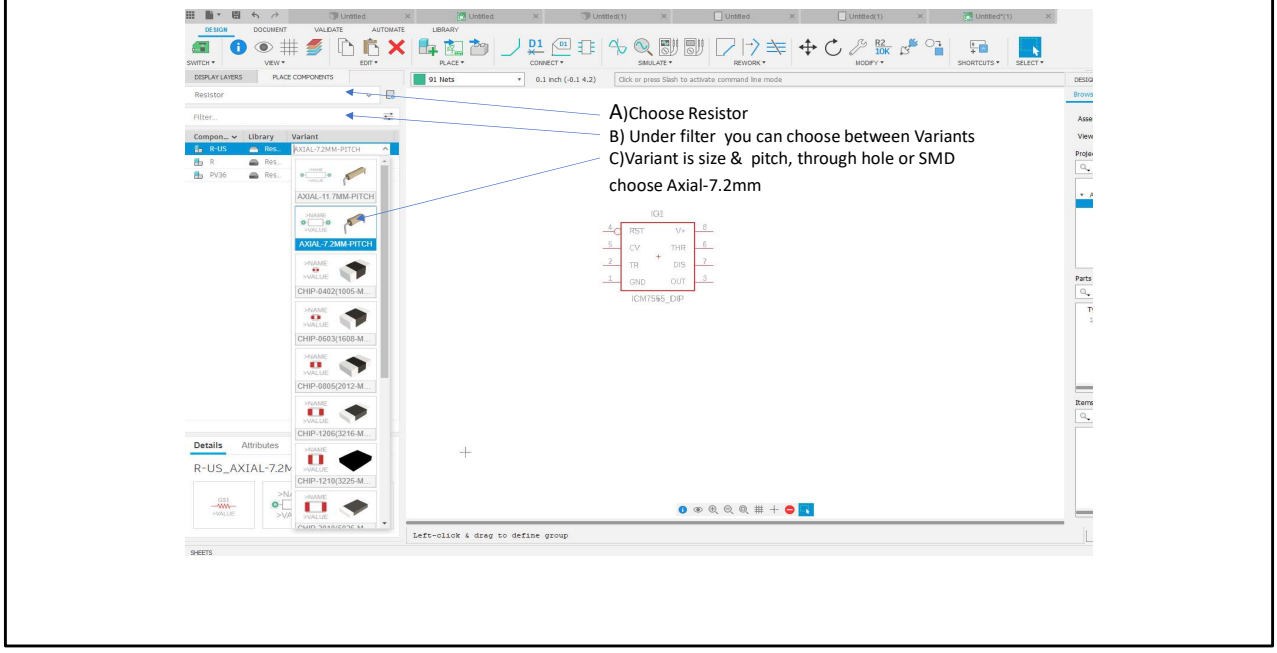
A) Then B) …… Drag and drop into the middle of your screen. Typically inputs are on the left, outputs on the right.



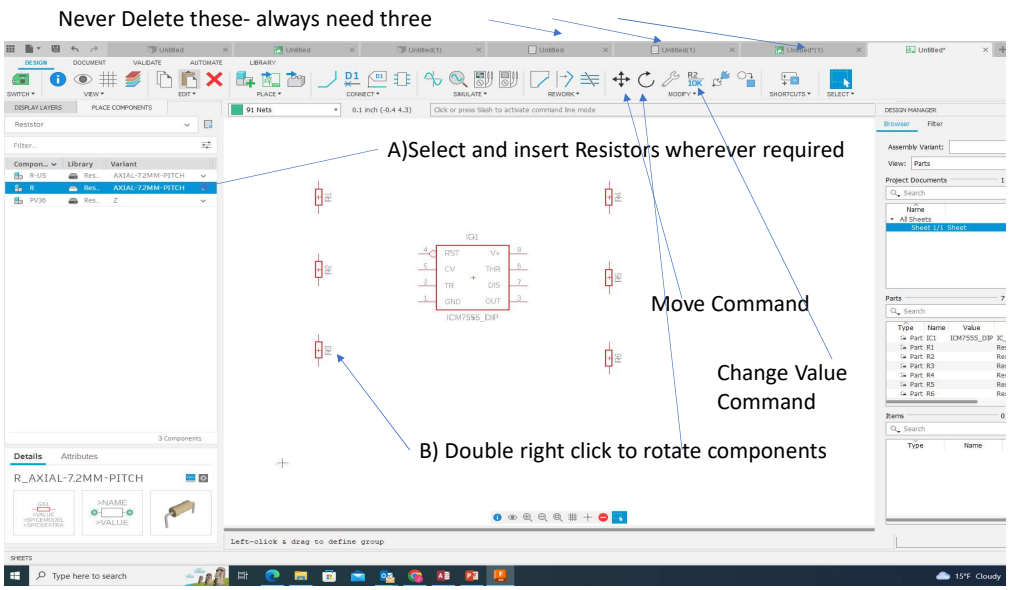
Add resistors to your circuit- do not be concerned about the value of the resistor, just the variant(size and pitch spacing)- type in resistor



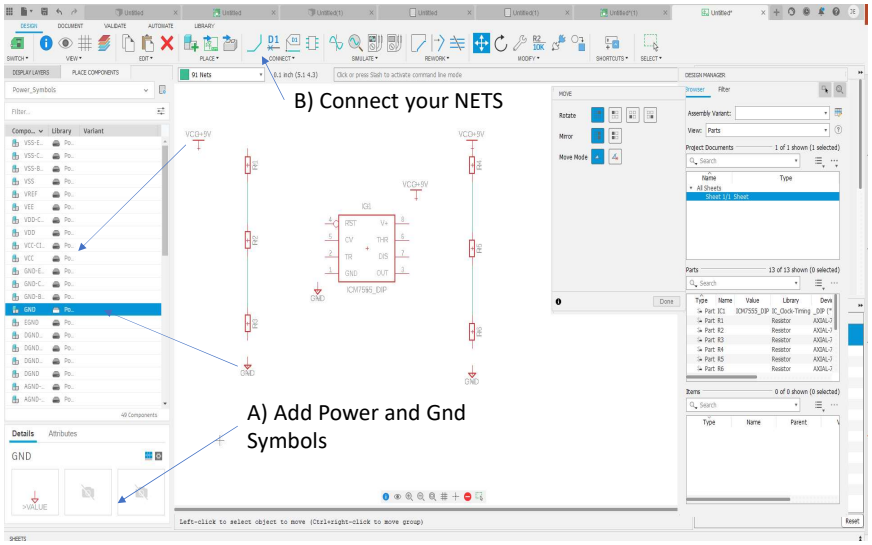
Note where “Resistors” was typed in, the screen will look like this… Choose next slide



Type “Resistor” in the top and this should show up, under Component -choose “R-US “ , then choose what is required, in ECE401 only through hole components are allowed, make sure the pitch and size is correct. Recommend AXIAL- 7.2MM- Pitch due to size constraints

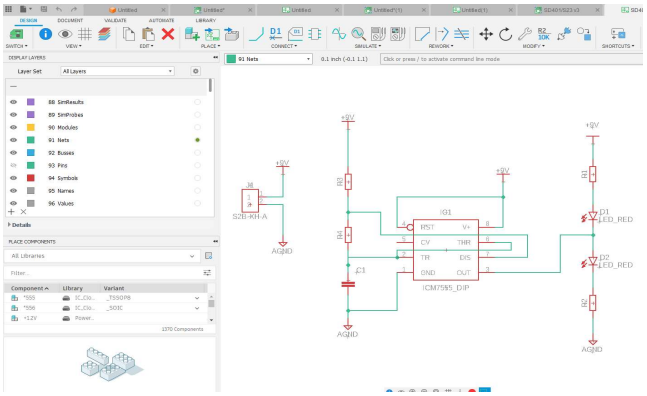


Easily change the value of components

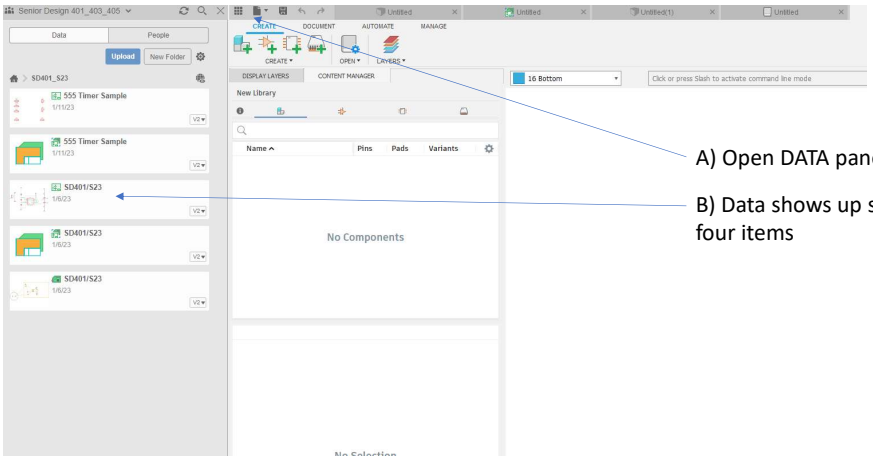


Add power and ground symbols; use the “Connect” icon line to connect components, these are called NETS, or Airwires, NETS tell the computer program where to add the copper traces from point A to B

Once Schematic completed switch to PCB Design



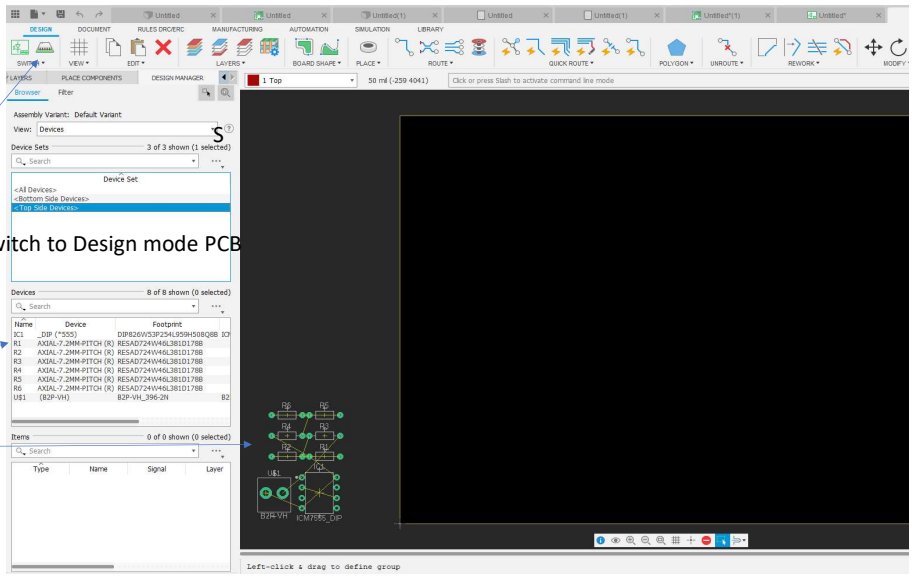
[PCB Layout Tutorial Walkthrough – YouTube](#) 0:32/ 4:38



6

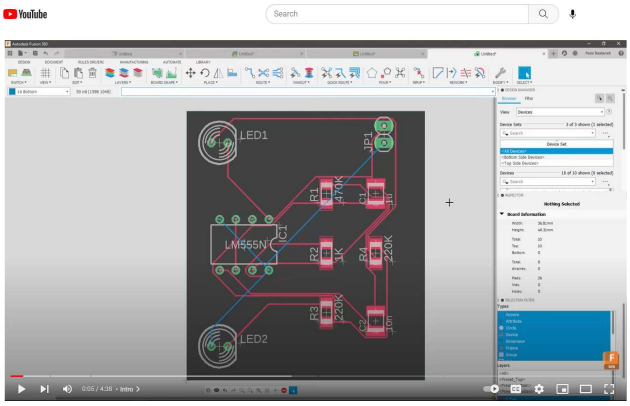
Open Data panel if you forgot where you saved everything to or forgot to save your design. This is your “Back up”

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



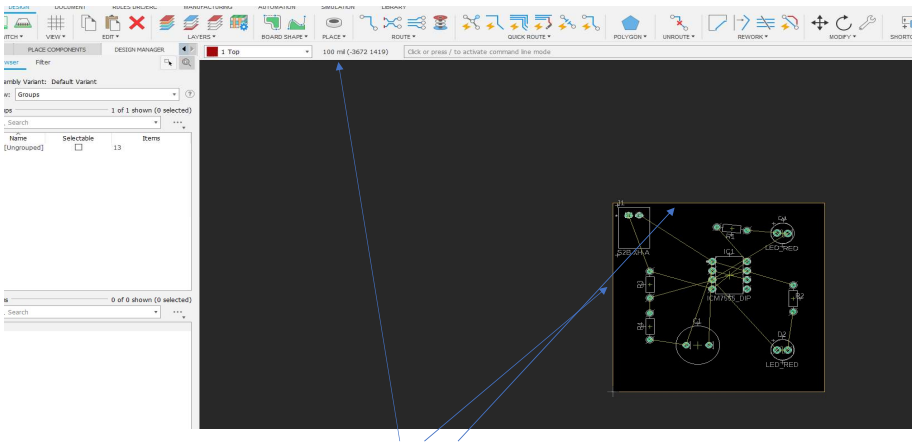
Part 2 Creating Circuit Board Layout

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



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

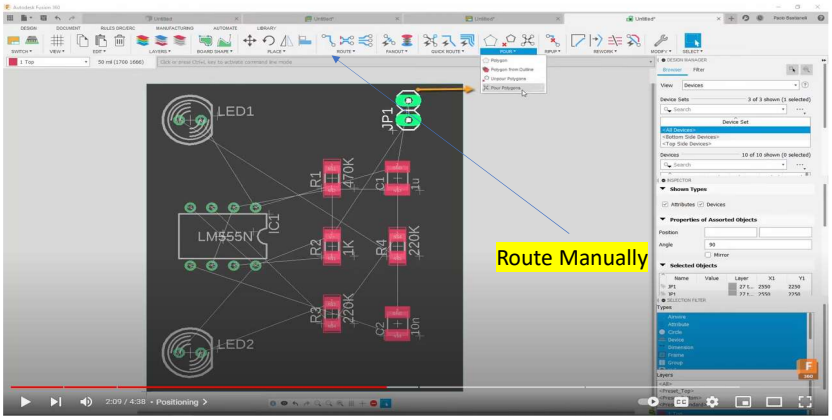
2D sim Components have been dragged into the Black square this is your PCB out line



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

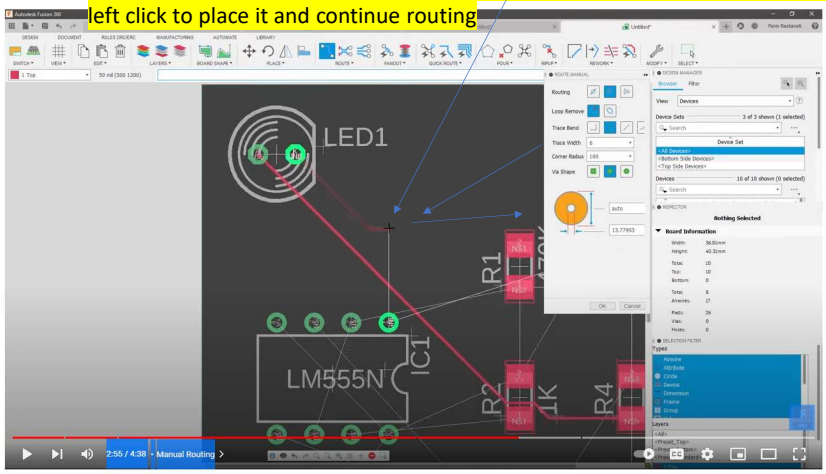
PCB Layout Tutorial Walkthrough – YouTube 0:42/4:38

To route traces manually and then automatically



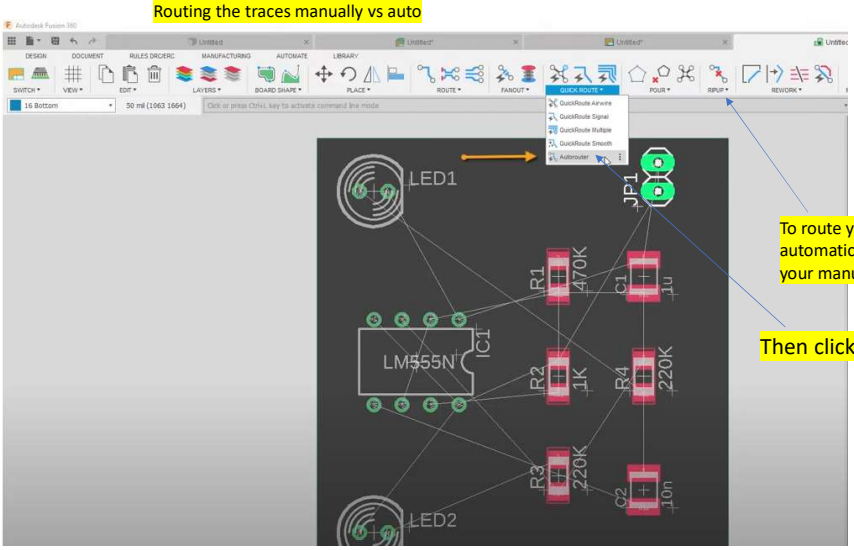
PCB Layout Tutorial Walkthrough – YouTube 2:32 mark

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

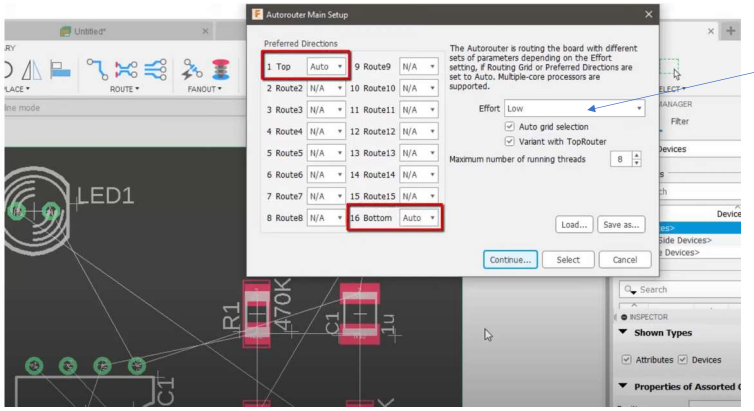


To route your traces automatically you may rip up your manual ones first.

Then click auto router

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

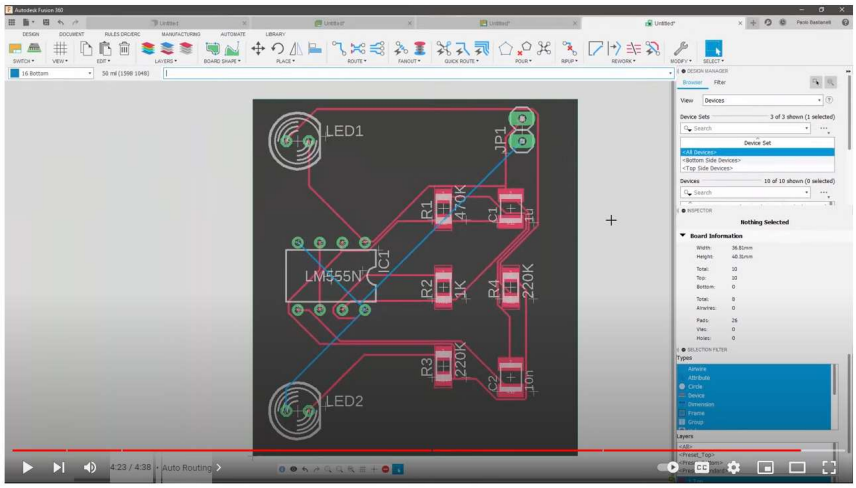
Two layer boards only top and bottom will be selected to auto route



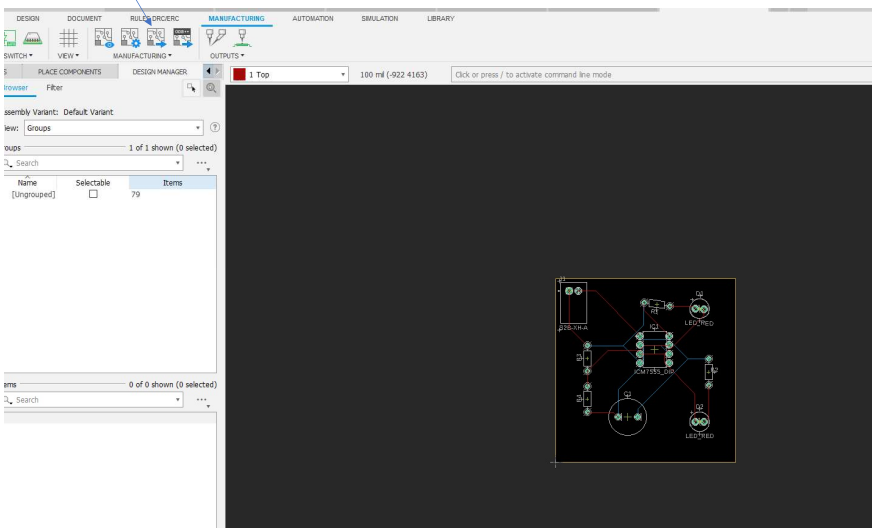
Change the effort to high to get a better selection of solutions

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

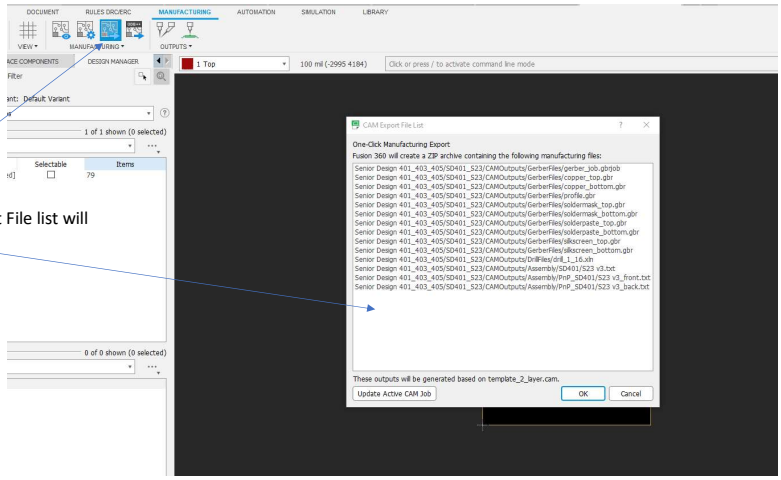
After choosing the solution with the least amount vias, the PCB is complete

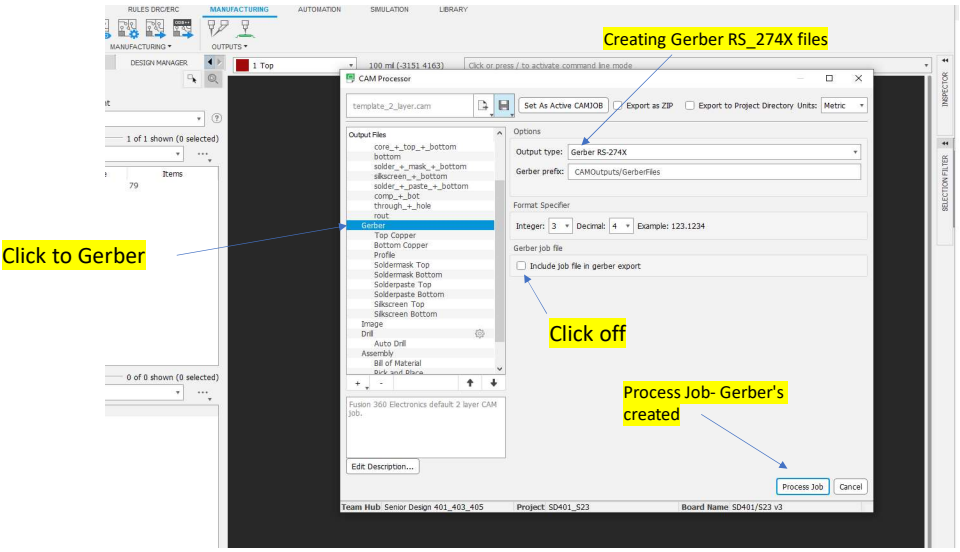


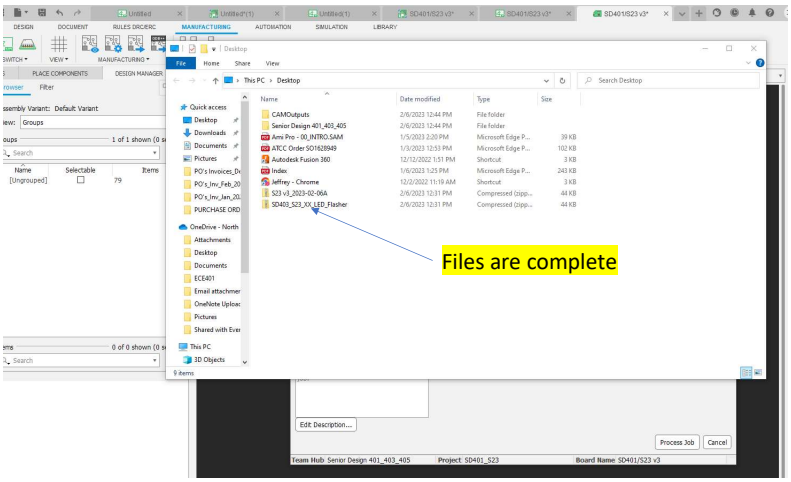
PCB Manufacturing- Creating Gerber Files



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

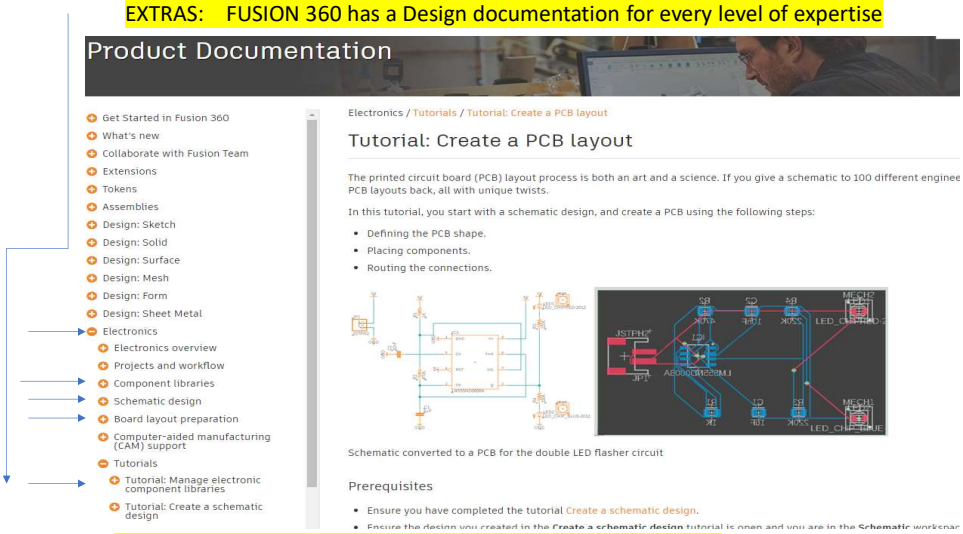



Creating Gerber RS_274X files
Click to Gerber
Click off
Process Job- Gerber's created
CAM Processor
Output Files
Gerber
Top Copper
Bottom Copper
Profile
Soldermask Top
Soldermask Bottom
Solderpaste Top
Solderpaste Bottom
Silkscreen Top
Silkscreen Bottom
Drill
Auto Drill
Assembly
Bill of Material
Options
Output type: Gerber RS-274X
Gerber prefix: CAMOutputs/GerberFiles
Format Specifier
Integer: 3 Decimal: 4 Example: 123.1234
Gerber Job file
Include job file in gerber export
Edit Description...
Process Job
Cancel

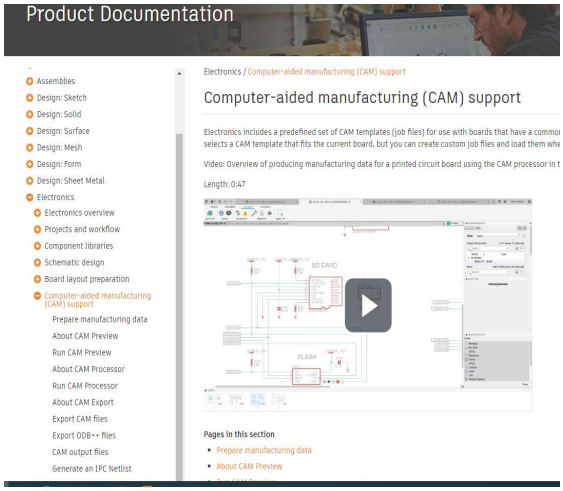


CAM Files are now Gerber files
Change File name to SD403_S23_XX_Flasher and
Email to Jeffrey.Erickson@ndsu.edu for verification and
ordering

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



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

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

Creating Gerber files