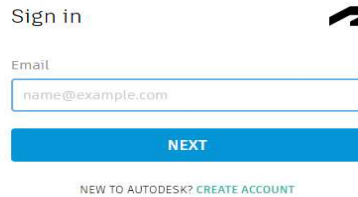


Getting Started: Step by Step creating a PCB with Fusion 360



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_deepink&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 its advantages and is used in Industry. It also is a program that lets 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.

Download Fusion 360 for Free | Free Trial | Autodesk

<https://www.autodesk.com/products/fusion-360/free-trial>



Fusion 360 is available for free personal use for individuals who are doing home-based, non-commercial design, manufacturing, and fabrication projects. Is **Fusion 360** free for students? **Fusion 360** is free cloud-based 3D CAD, CAM, CAE and PCB software for qualifying students as a 1-year subscription.

Another note from Autodesk about Fusion360...

AUTODESK FUSION360

Fusion360 Schematic Design Tutorial

<https://www.youtube.com/watch?v=lqwHkB9IsUo>

Fusion360 PCB Design Tutorial

<https://www.youtube.com/watch?v=VZ7BEocoyDA>

or

https://www.youtube.com/watch?v=jgUZeBiusw&list=PLmA_xUT-8UIL80Xm8Gxz98YNum3I9Glnr 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=lqwHkB9IsUo>

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

⚠ 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.](#)

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

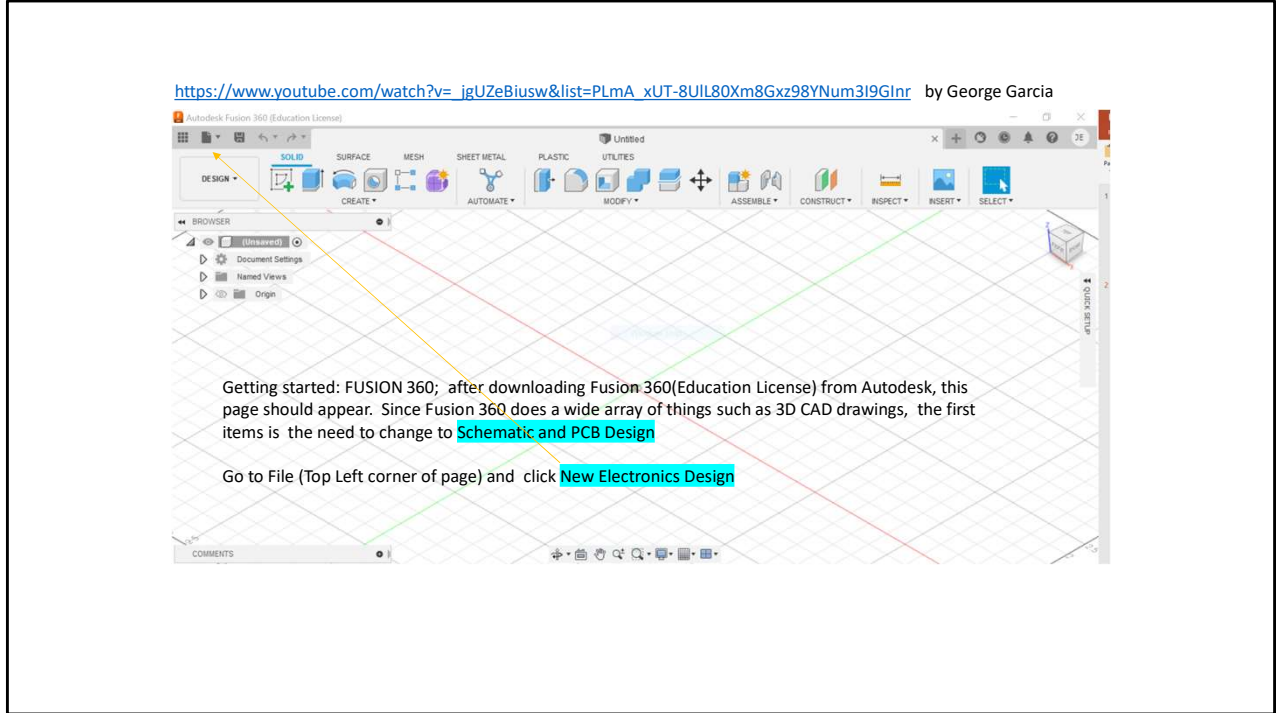
1) Schematic- After Breadboarding is finished and with the benefit of Circuit Lab schematic Create a schematic diagram

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

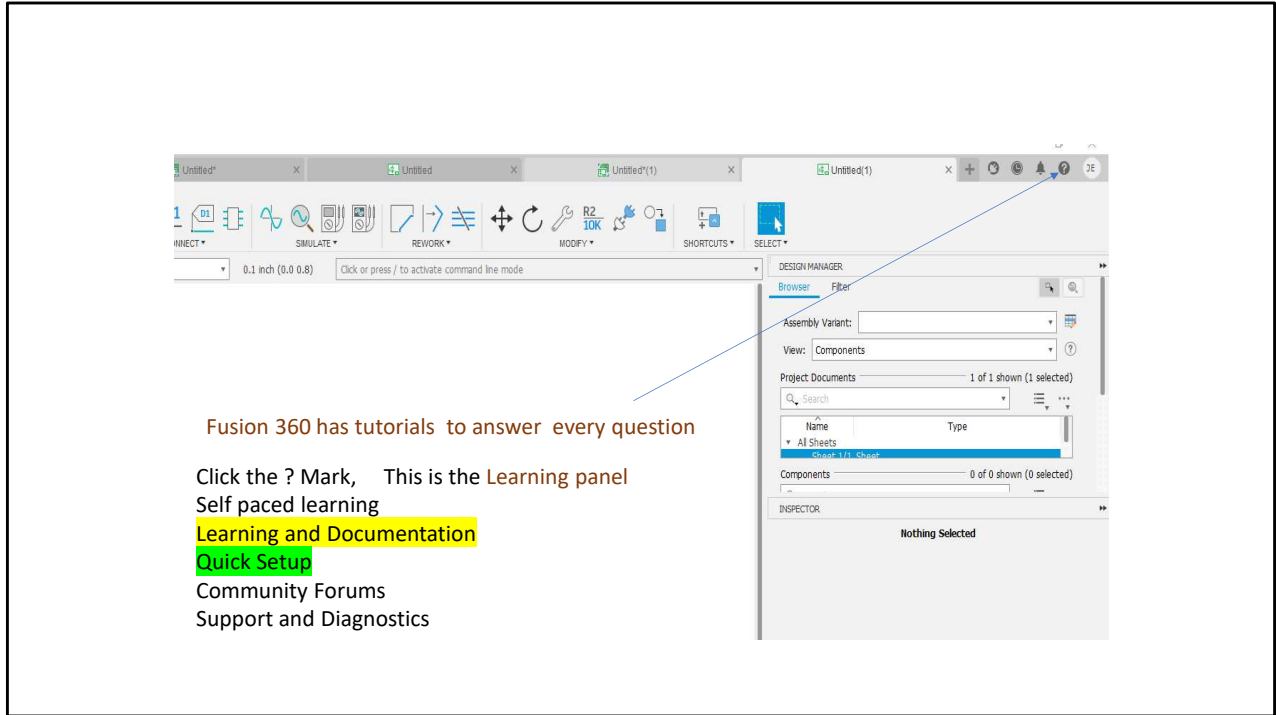
3) PCB Design- Convert your Schematic to PCB Design

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

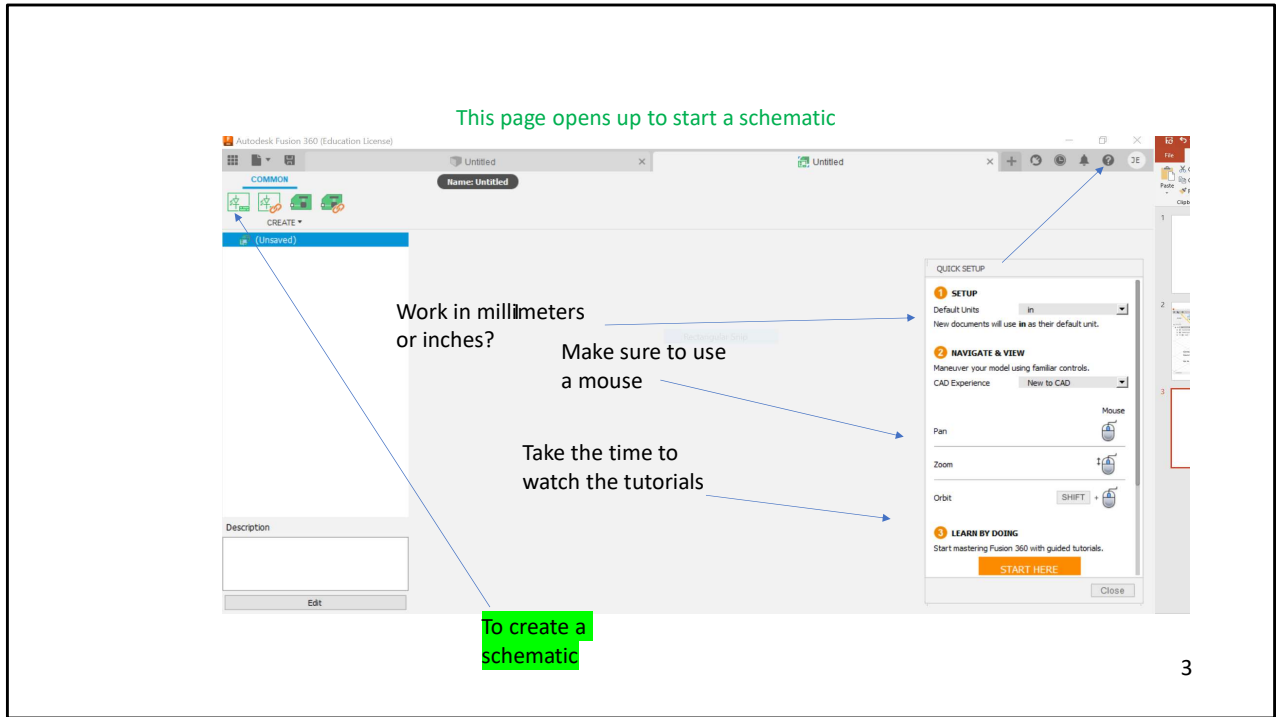
These are four requirements to make a PCB..... Create a schematic, need a parts list, Create the PCB Design, Create the Gerber's for manufacturing



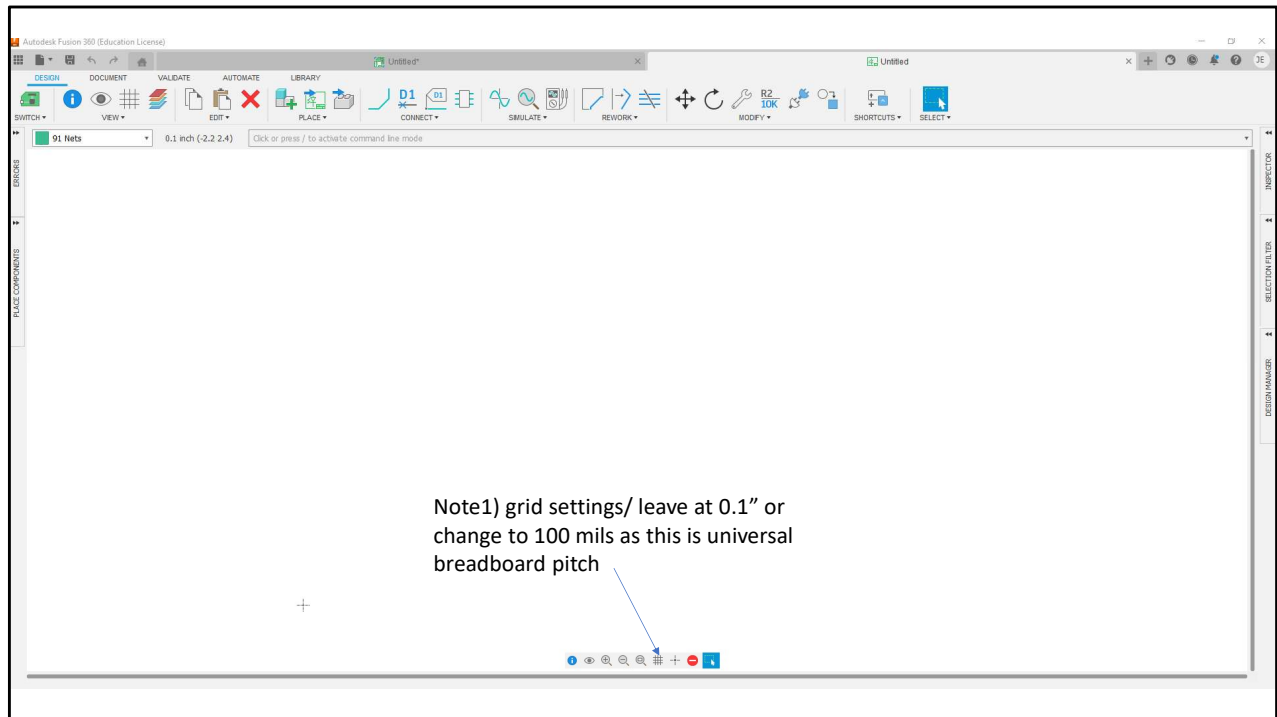
To get started and after logging , this page should show up, take a note where the arrows go. As it says, first step is to change to Schematic and PCB Design... click New Electronics Design



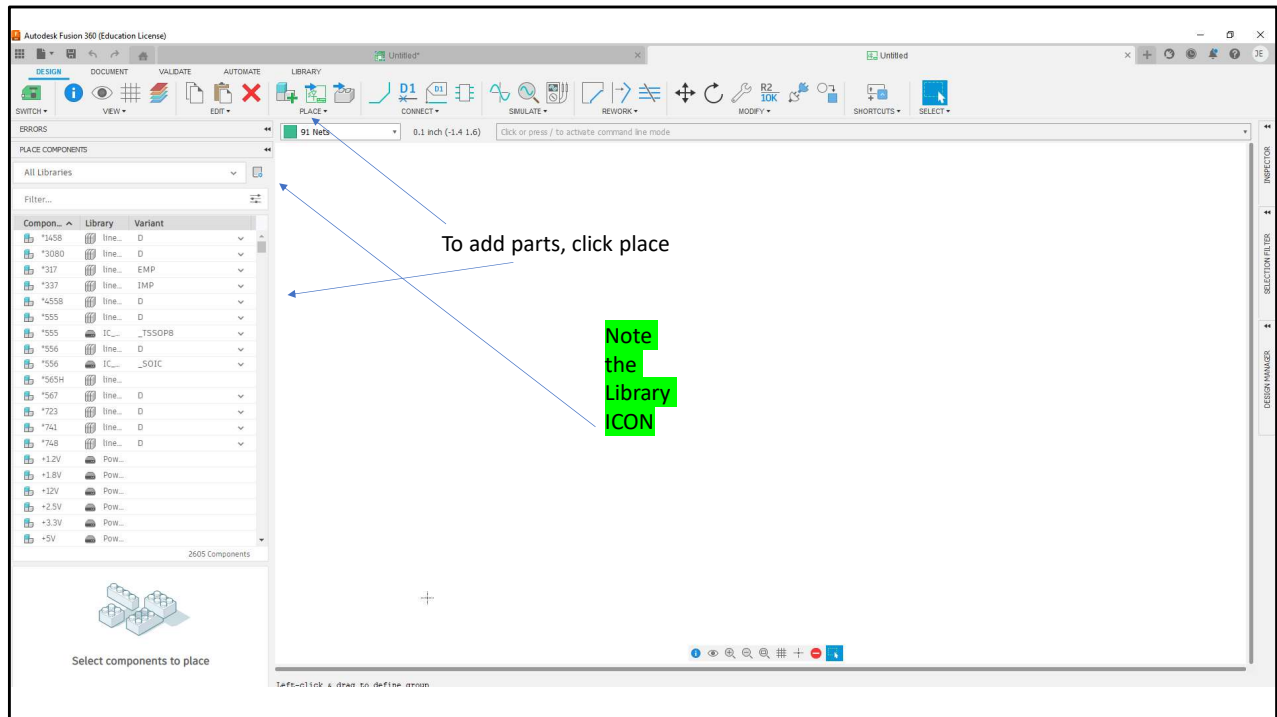
Clicking the ? Will help answer many question. Click on this to bring you to Learning and Documentation, or quick setup



. 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.

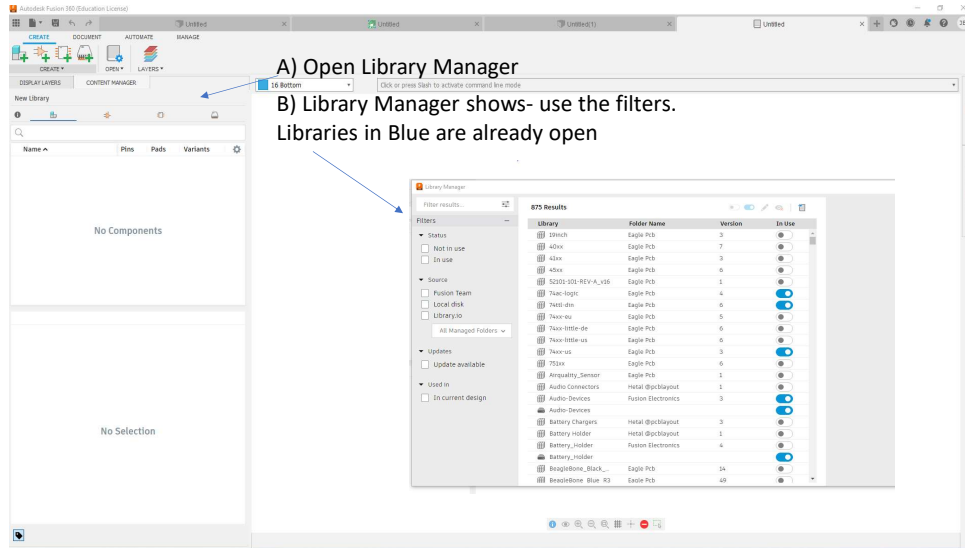


After clicking the Schematic Icon a blank screen shows up. Note 1) Users preference for dots, lines or no display,



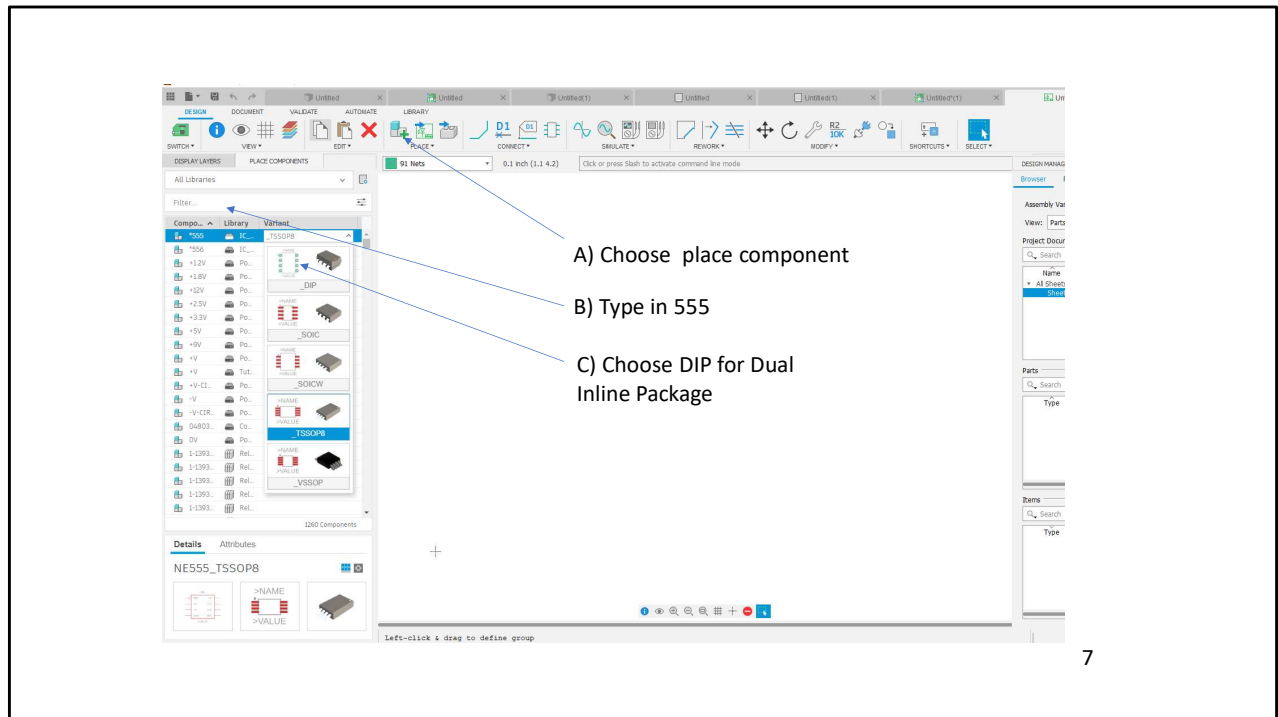
Click the Place under Library tab to add parts, Components and libraries listing shows. Here is where you access all the parts libraries in Fusion360

https://www.youtube.com/watch?v=jgUZeBlusw&list=PLma_xUT-8U1L80Xm8Gxz98YNum3j9Ginr
Getting started by George Garcia of Fusion 360

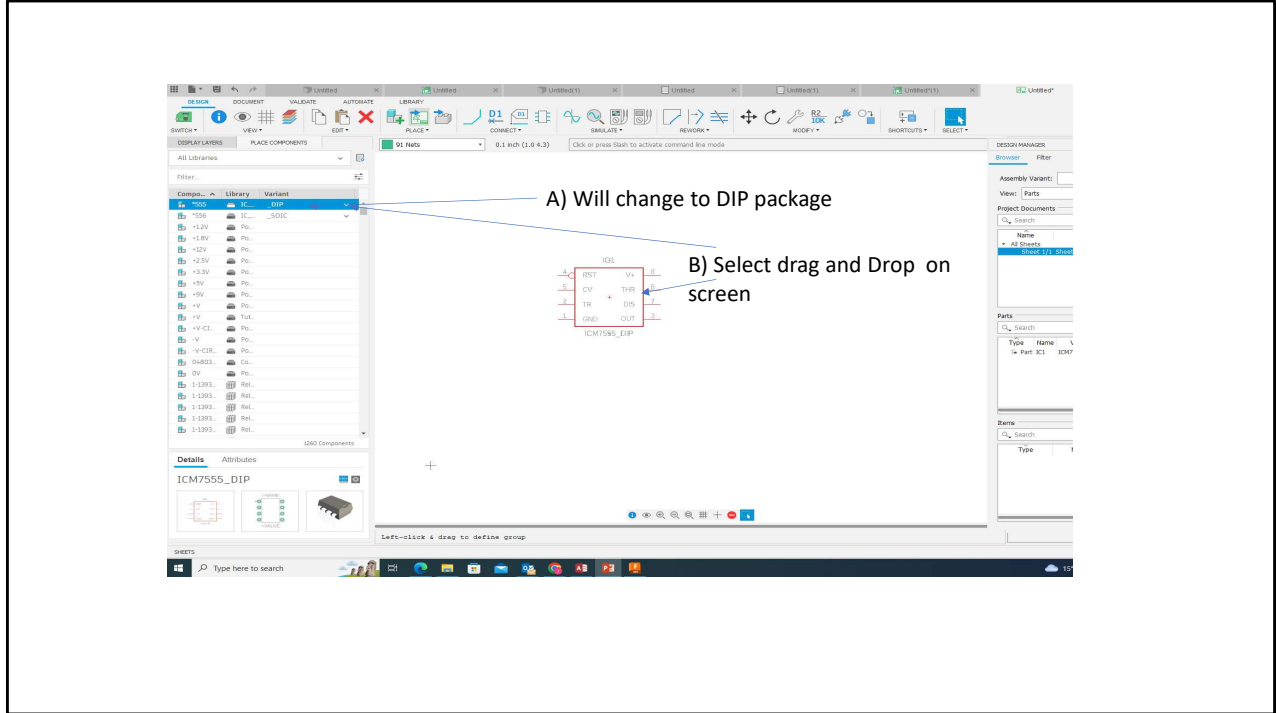


4

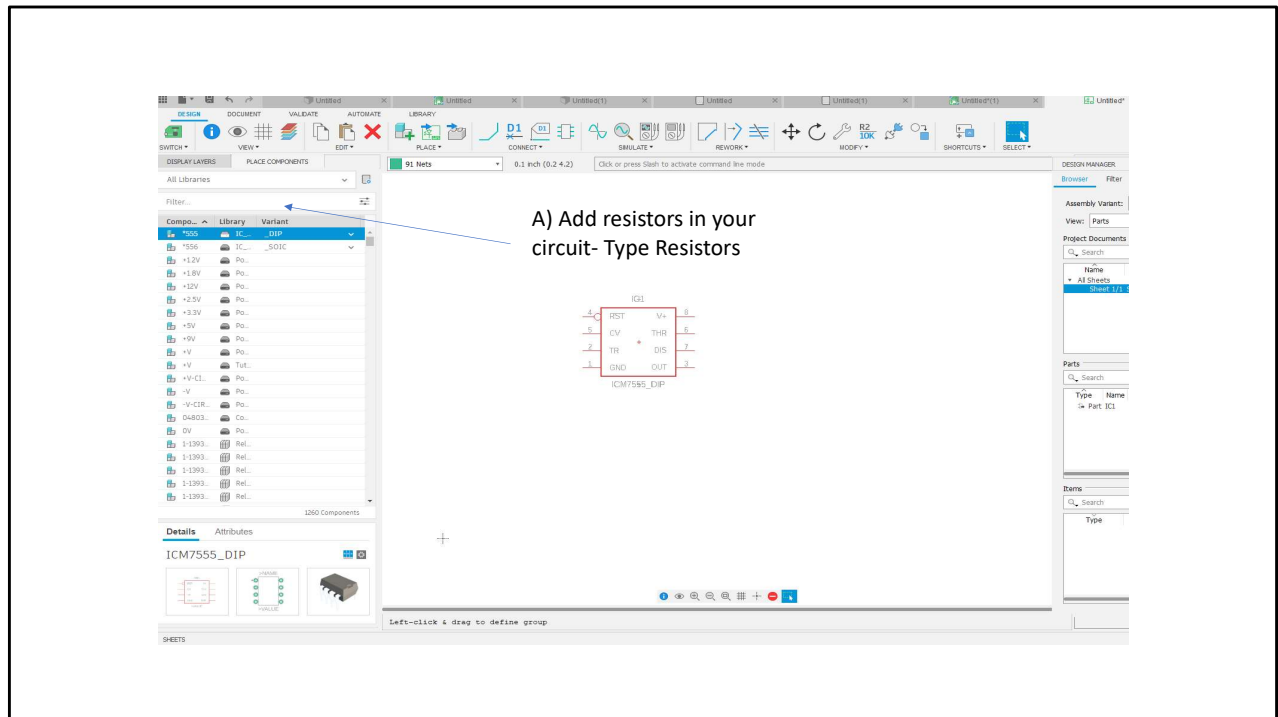
After clicking the library icon, all the libraries available to you open up, those in use are blue and already available, if you cannot find what you need you need to search for them. Due to the extensive library this can become challenging, but once found it will save them in a library called "In this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



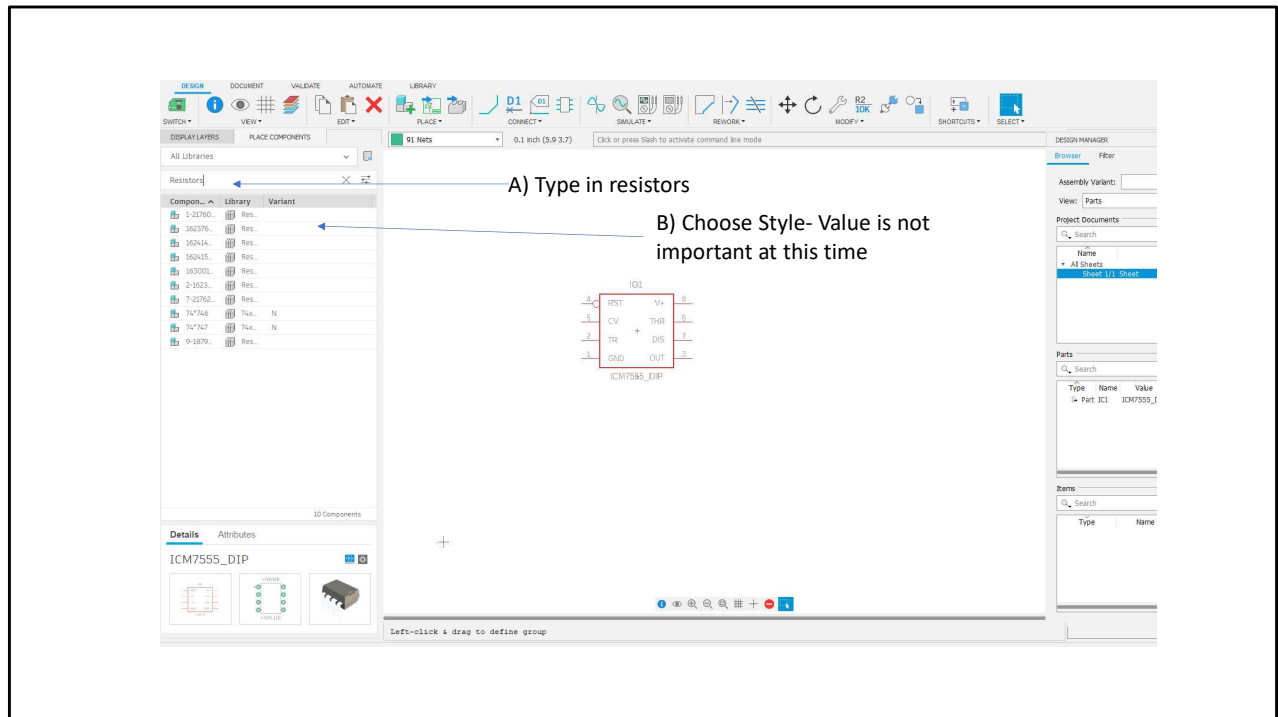
In this case we will create a circuit using a Timer IC (555 Timer) , note the different Variants under this library, choose a thru hole component, select , drag onto screen and drop



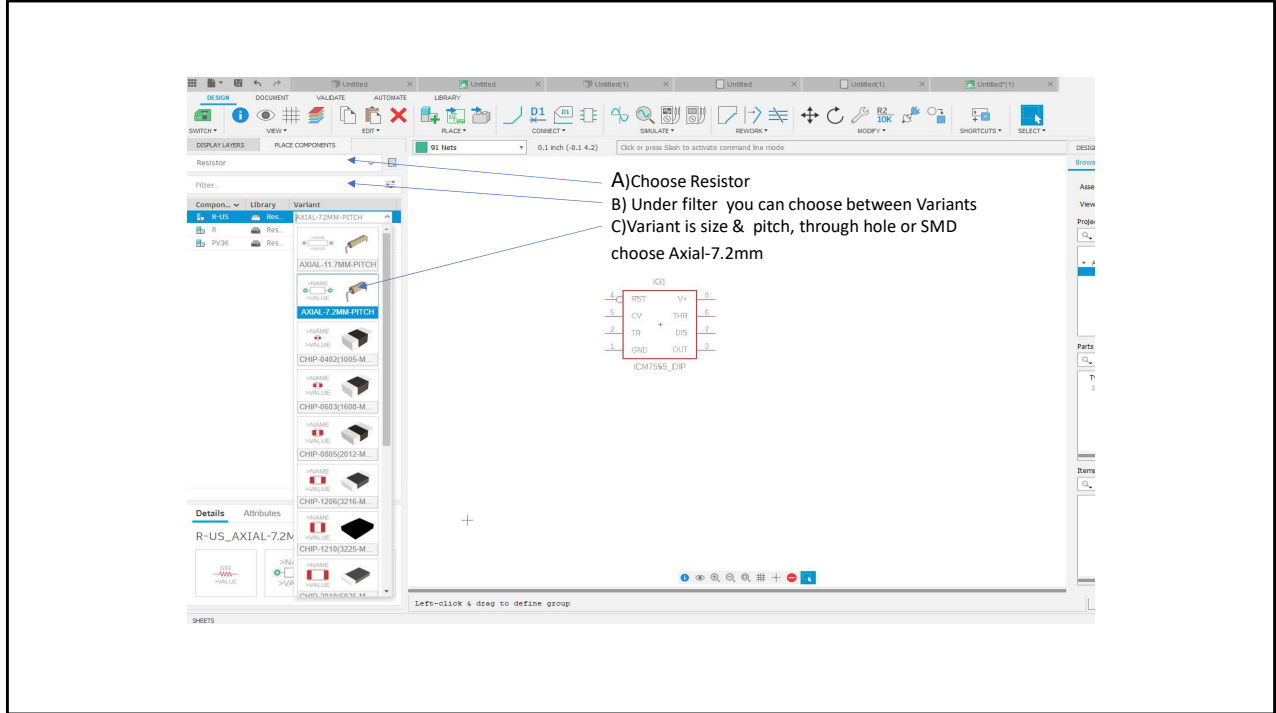
A) Then B) Drag and drop into the middle of your screen. Typically inputs are on the left, outputs on the right.



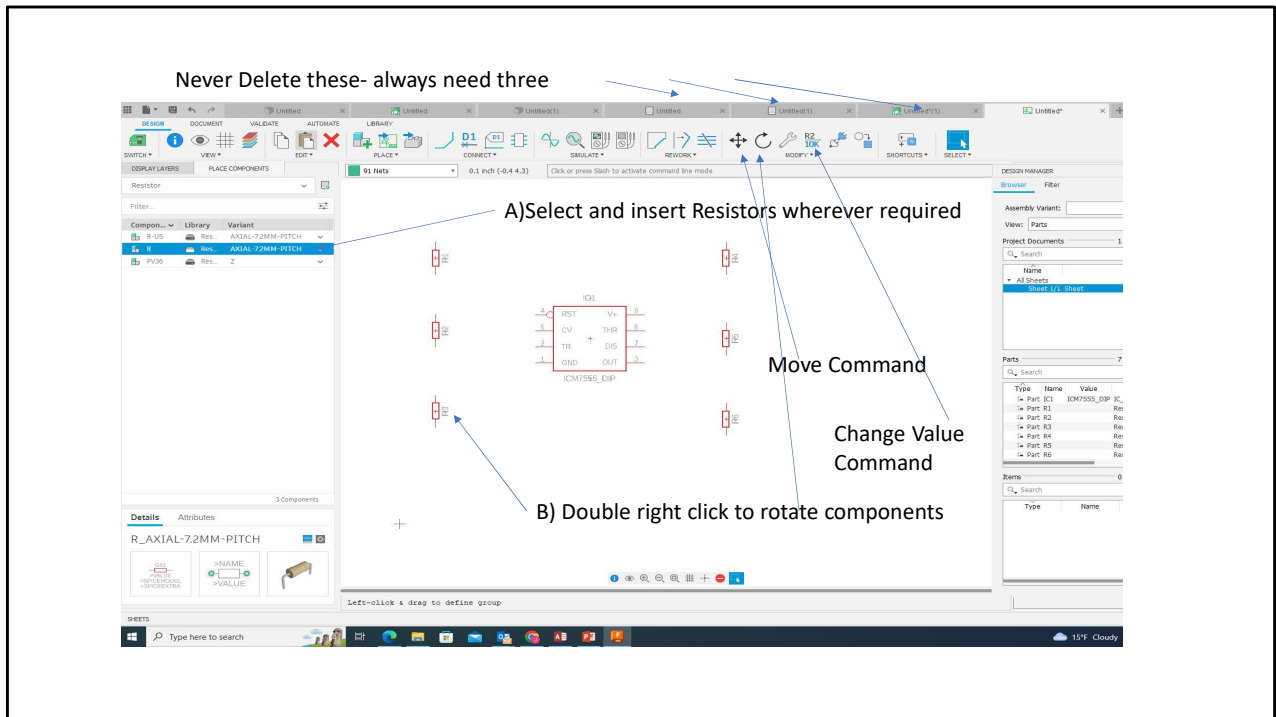
Add resistors to your circuit- do not be concerned about the value of the resistor, just the variant(size and pitch spacing)- type in resistor



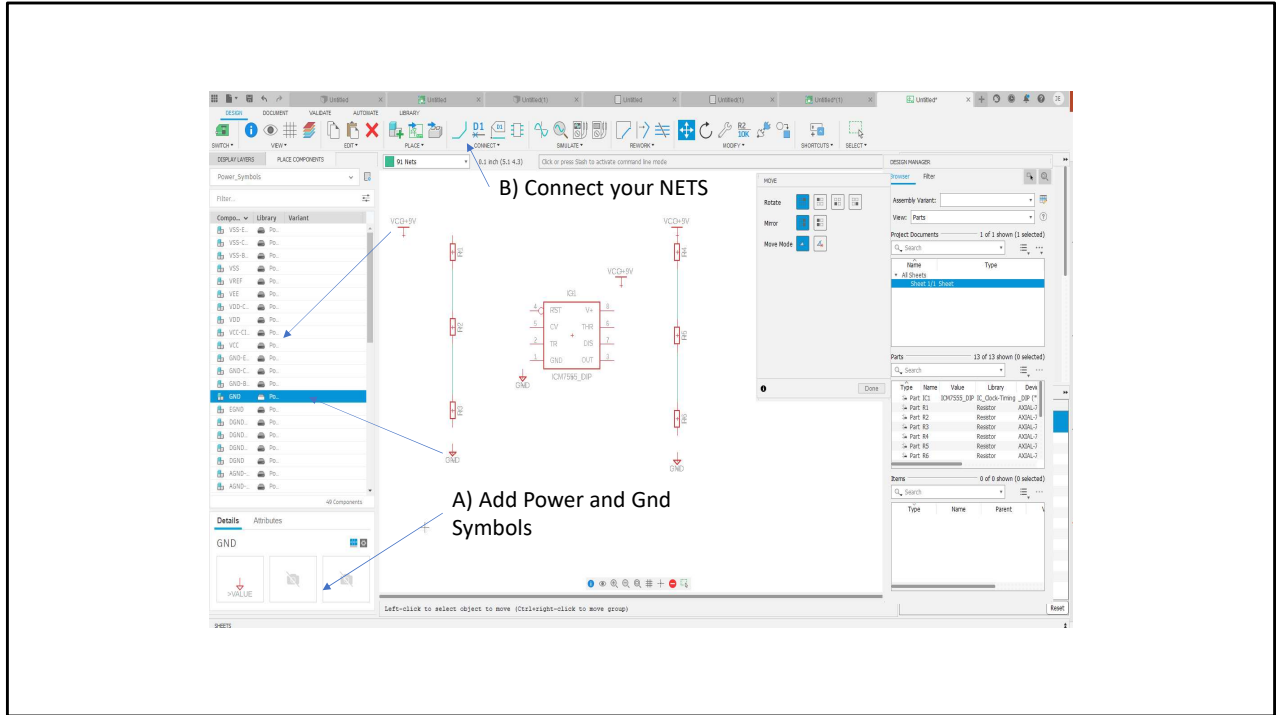
Note where “Resistors” was typed in, the screen will look like this... Choose next slide



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

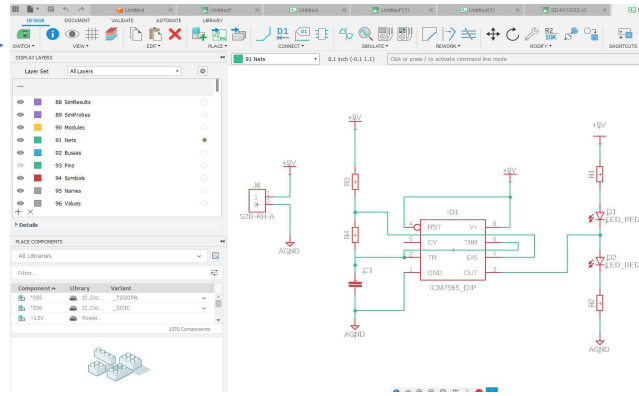


Easily change the value of components

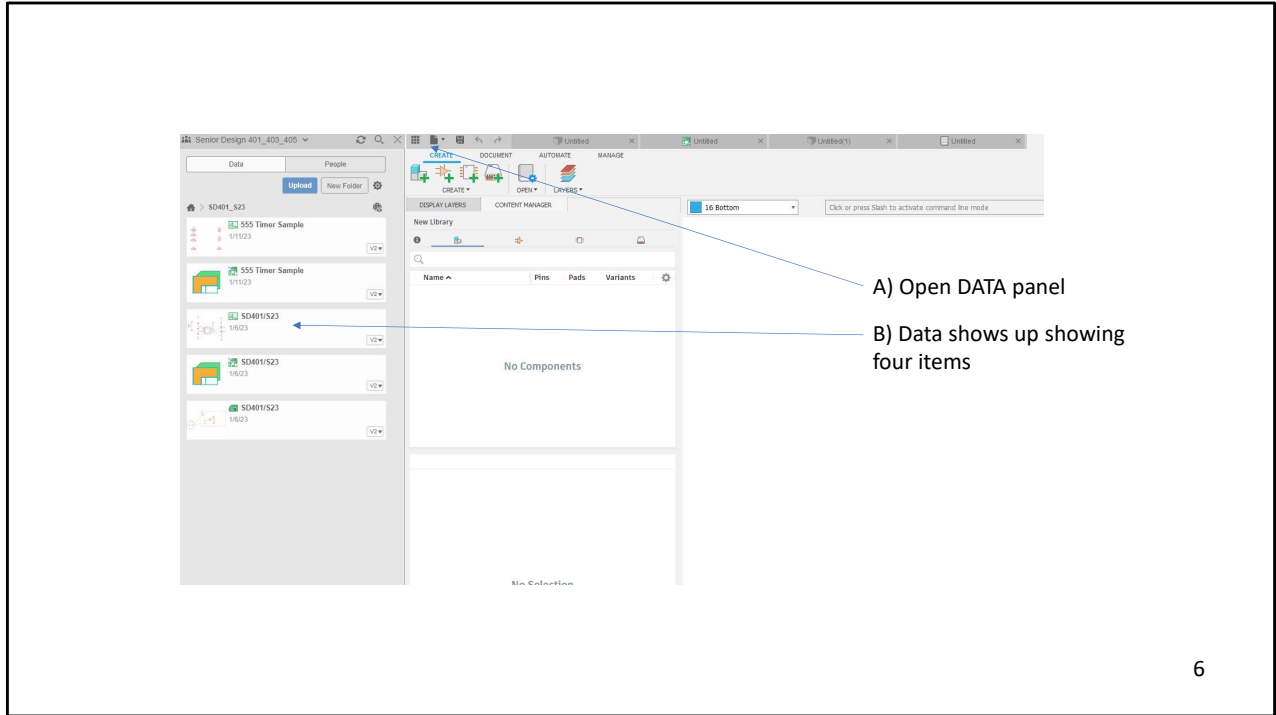


Add power and ground symbols; use the “Connect” icon line to connect components, these are called NETS, or Airwires, NETS tell the computer program where to add the copper traces from point A to B

Once Schematic completed switch to PCB Design



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



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

The screenshot shows the Altium Designer PCB layout environment. The top menu bar includes options like DESIGN, DOCUMENT, RULES DROPER, MANUFACTURING, AUTOMATION, SIMULATION, and LIBRARY. The main workspace is currently in a dark state, with a small schematic diagram visible in the bottom-left corner. The left-hand panels are active, showing the 'Browser' and 'Place Components' sections.

Annotations with arrows point to specific parts of the interface:

- An arrow points to the 'Browser' panel, with the text "Switch to Design mode PCB".
- An arrow points to the 'Devices' list in the 'Place Components' panel, with the text "List of components in Design".
- An arrow points to the 'Items' list in the 'Place Components' panel, with the text "2D Sim of components in design".

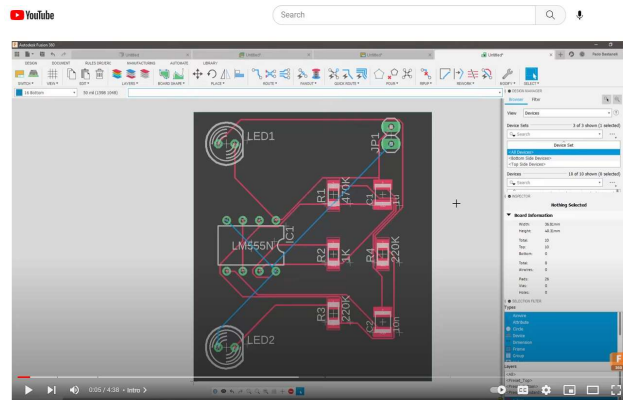
Name	Device	Footprint
R3	DIP (*NOS)	DIP824V03P034L0304S08Q8B ID
R1	AXIAL-7-20MM-PITCH (R)	RESAD724VW6L8E1D1788
R2	AXIAL-7-20MM-PITCH (R)	RESAD724VW6L8E1D1788
R3	AXIAL-7-20MM-PITCH (R)	RESAD724VW6L8E1D1788
R4	AXIAL-7-20MM-PITCH (R)	RESAD724VW6L8E1D1788
R5	AXIAL-7-20MM-PITCH (R)	RESAD724VW6L8E1D1788
R6	AXIAL-7-20MM-PITCH (R)	RESAD724VW6L8E1D1788
U81	ESP-W0	ESP-W0_20S-01 B2

Type	Name	Signal	Layer
------	------	--------	-------

Left-click & drag to define group

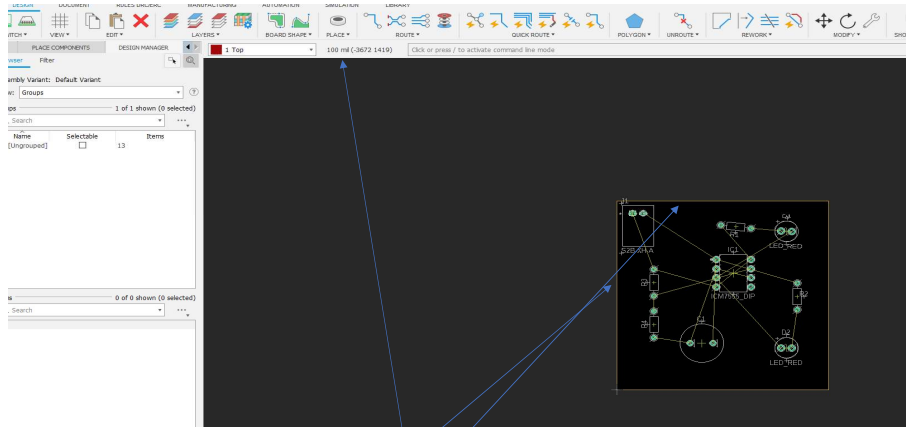
Part 2 Creating Circuit Board Layout

<https://www.youtube.com/watch?v=VZ7BFocaYD4>



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

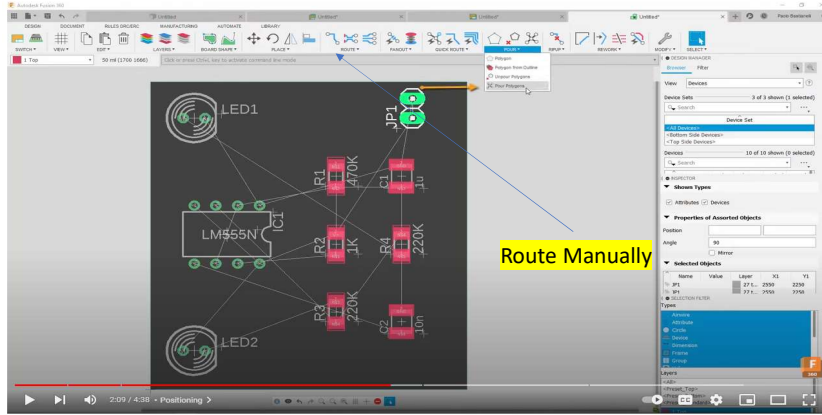
2D sim Components have been dragged into the Black square this is your PCB out line



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

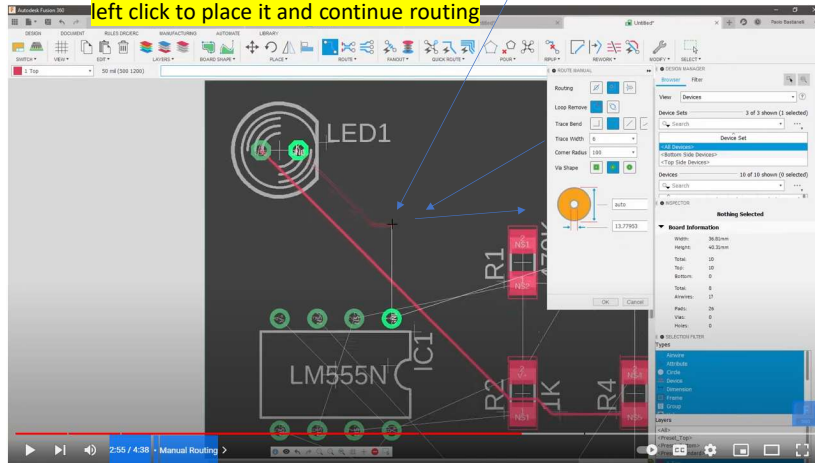
[PCB Layout Tutorial Walkthrough – YouTube](#) 0:42/4:38

To route traces manually and then automatically



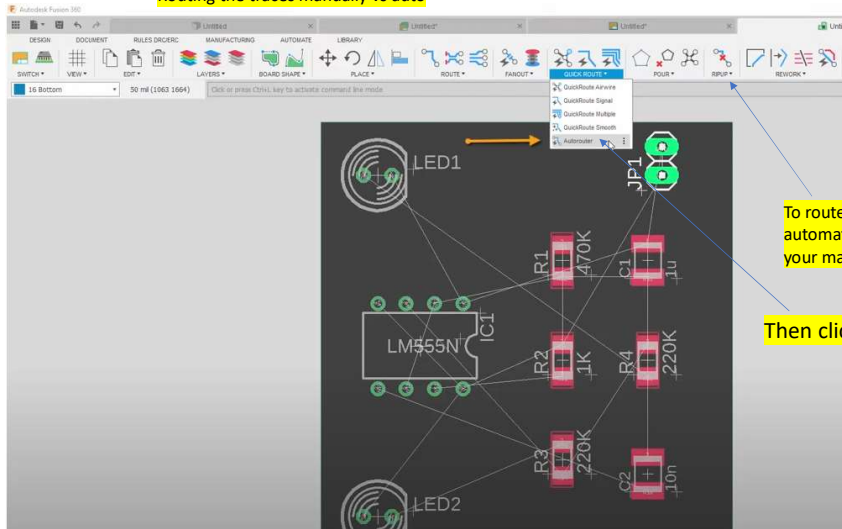
[PCB Layout Tutorial Walkthrough – YouTube](#) 2:32 mark

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

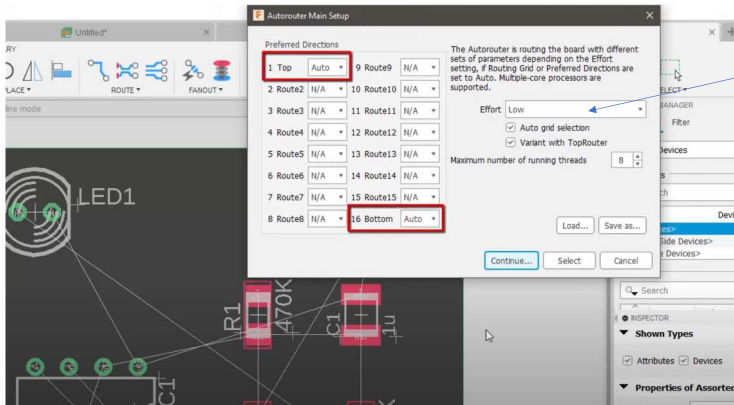


To route your traces automatically you may rip up your manual ones first.

Then click auto router

<https://www.youtube.com/watch?v=VZZBEccoYDA> 3:39/4:38

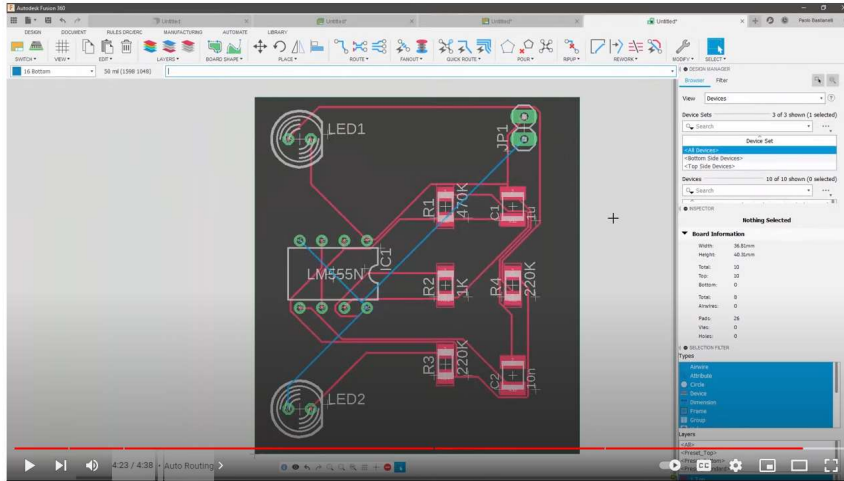
Two layer boards only top and bottom will be selected to auto route



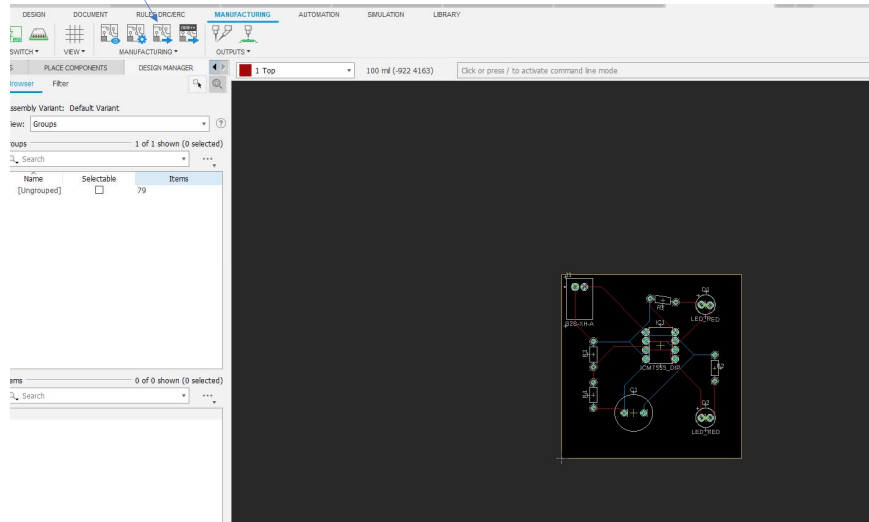
Change the effort to high to get a better selection of solutions

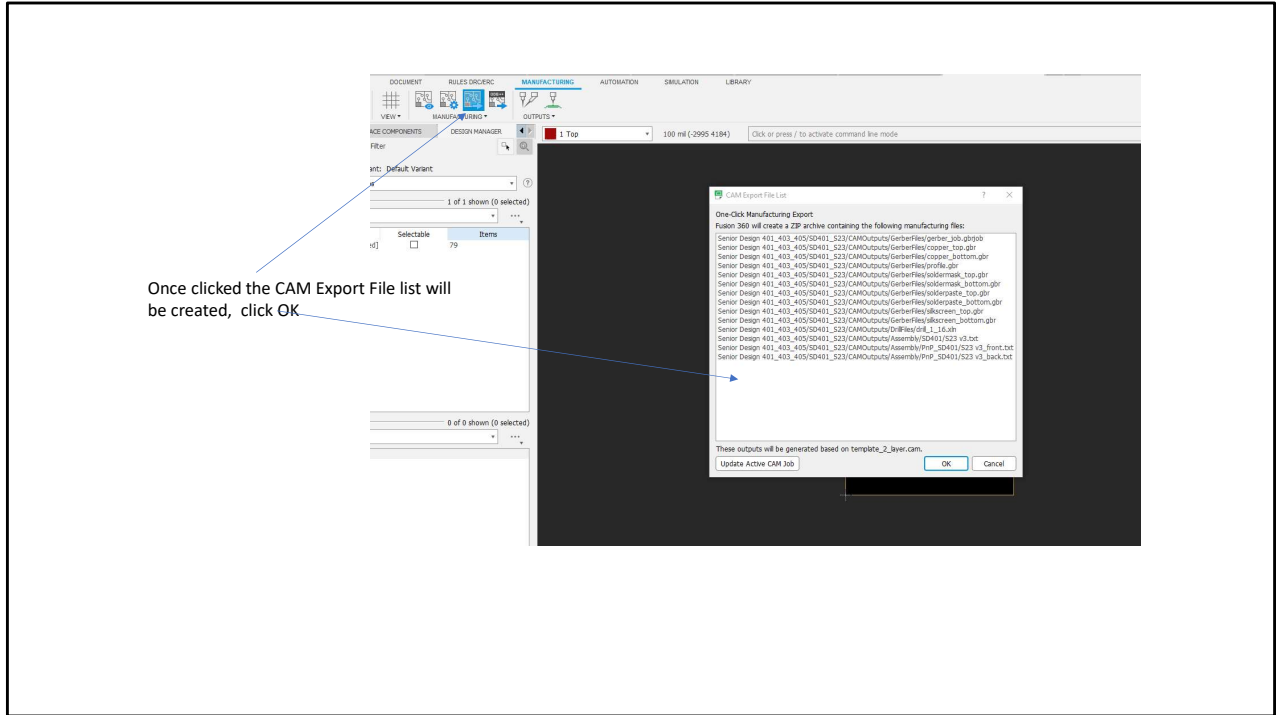
<https://www.youtube.com/watch?v=VZZBEocoYDA>

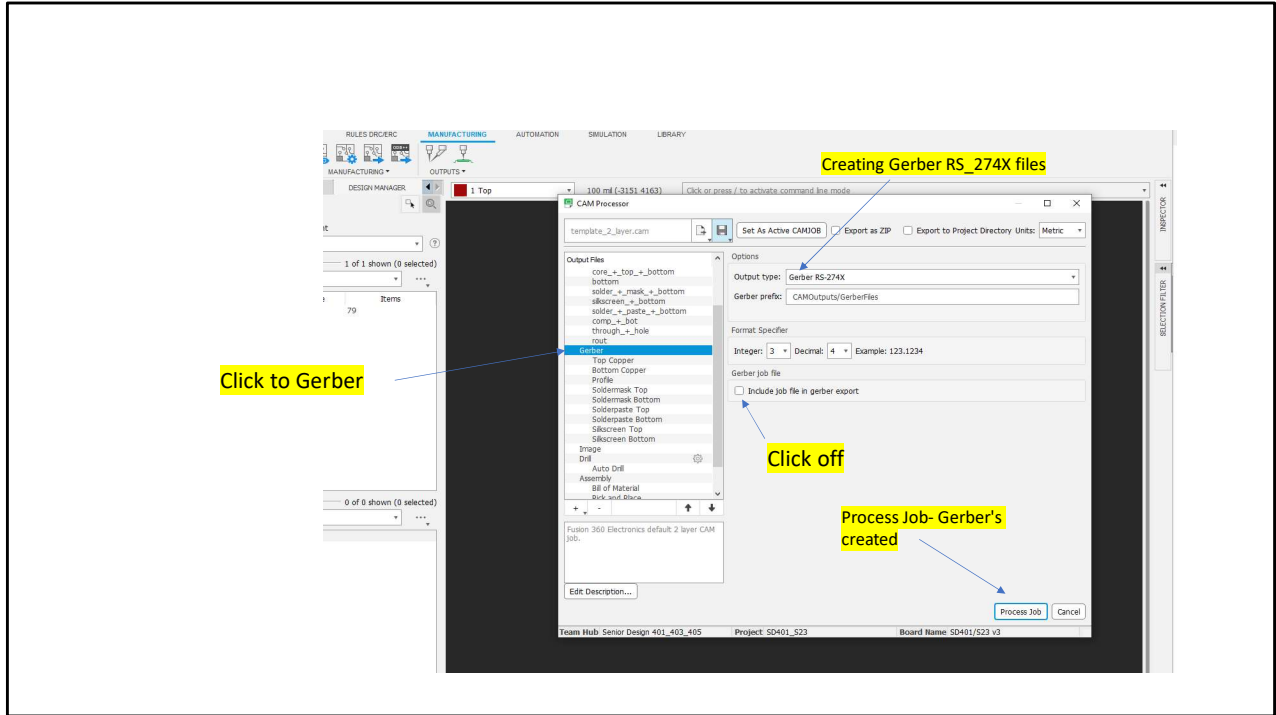
After choosing the solution with the least amount of vias, the PCB is complete

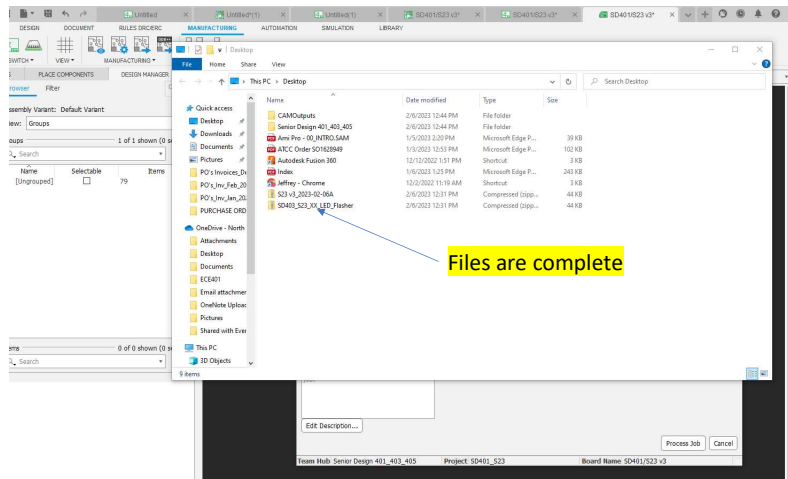


PCB Manufacturing- Creating Gerber Files









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

Product Documentation

- 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

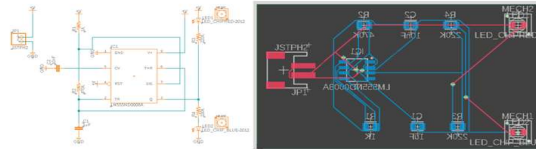
Electronics / Tutorials / Tutorial: Create a PCB layout

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 engineer 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** workspace.

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

Product Documentation

- 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
 - Prepare manufacturing data
 - About CAM Preview
 - Run CAM Preview
 - About CAM Processor
 - Run CAM Processor
 - About CAM Export
 - Export CAM files
 - Export ODB++ files
 - CAM output files
 - Generate an IPC Netlist

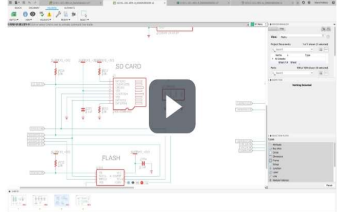
Electronics / Computer-aided manufacturing (CAM) support

Computer-aided manufacturing (CAM) support

Electronics includes a predefined set of CAM templates (job files) for use with boards that have a common selects a CAM template that fits the current board, but you can create custom job files and load them when

Video: Overview of producing manufacturing data for a printed circuit board using the CAM processor in t

Length: 0:47



Pages in this section

- Prepare manufacturing data
- About CAM Preview