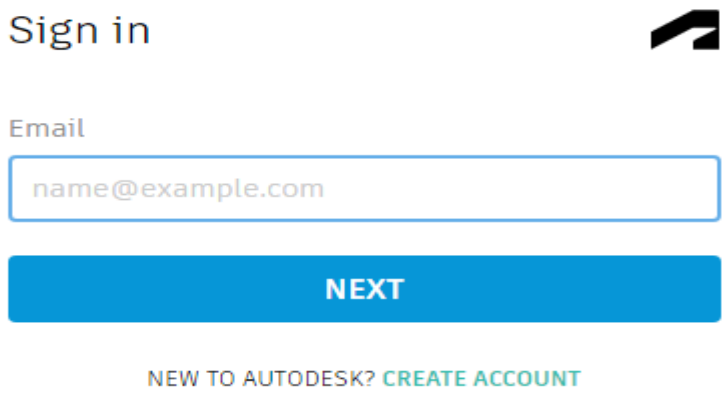
Getting Started: **Step by Step creating a PCB with Fusion 360**



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

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.

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

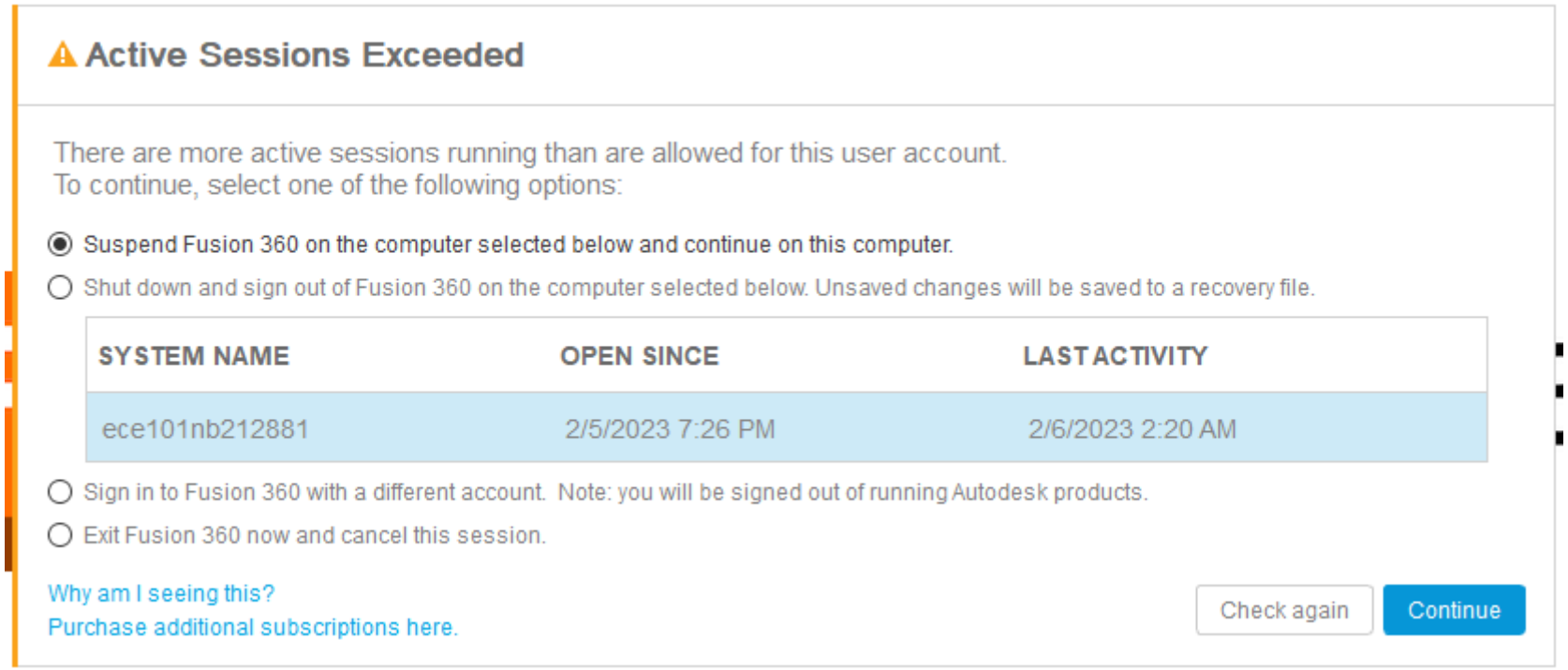
https://www.youtube.com/watch?v=_jgUZeBiusw&list=PLmA_xUT-8UlL80Xm8Gxz98YNum3I9GInr A to Z by George Garcia

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

One account/ one active session at a time



What does it take to make a PCB? Four steps

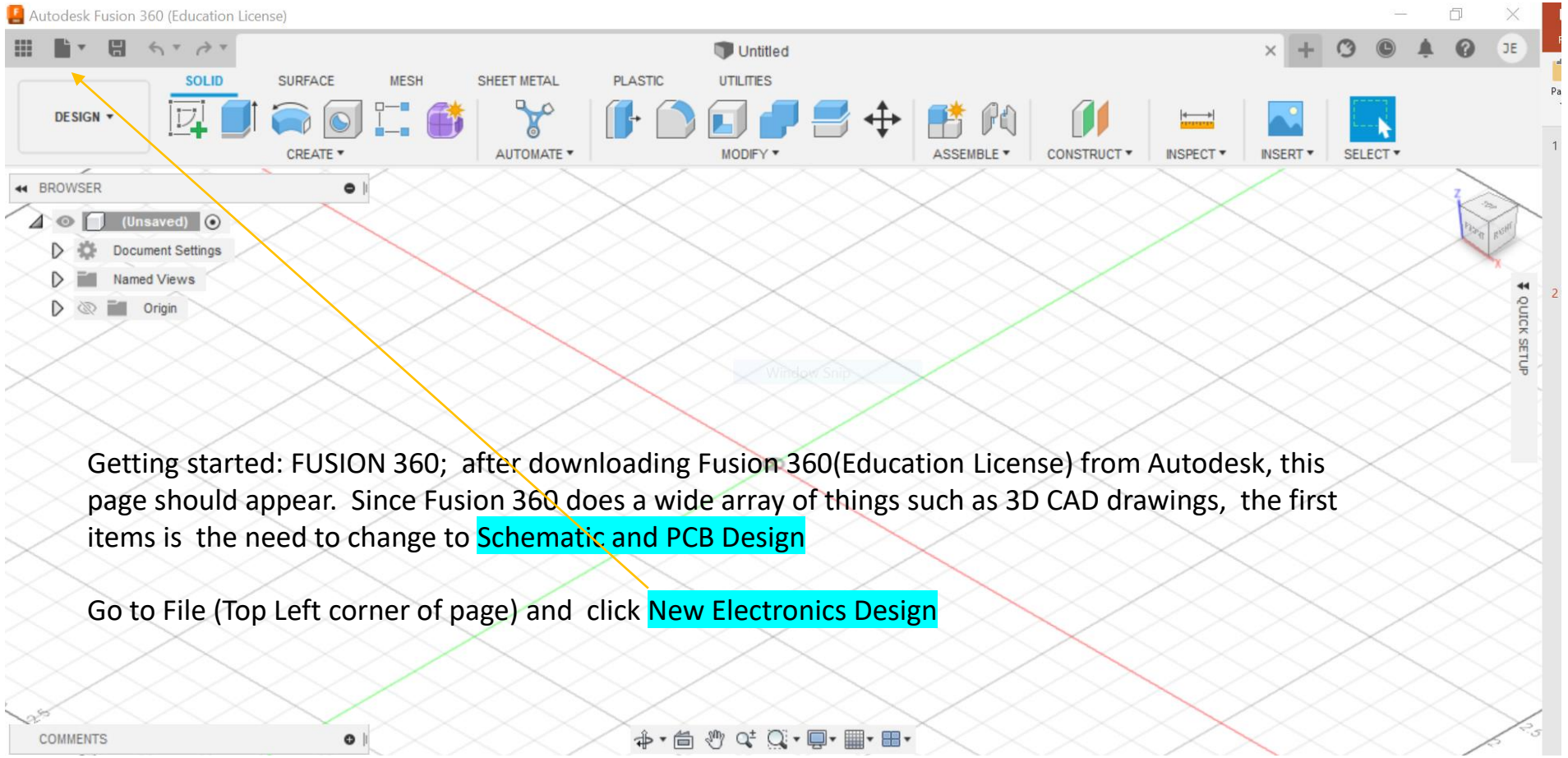
1) Schematic- After Breadboarding is finished and with the benefit of Circuit Lab schematic Create a schematic diagram

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

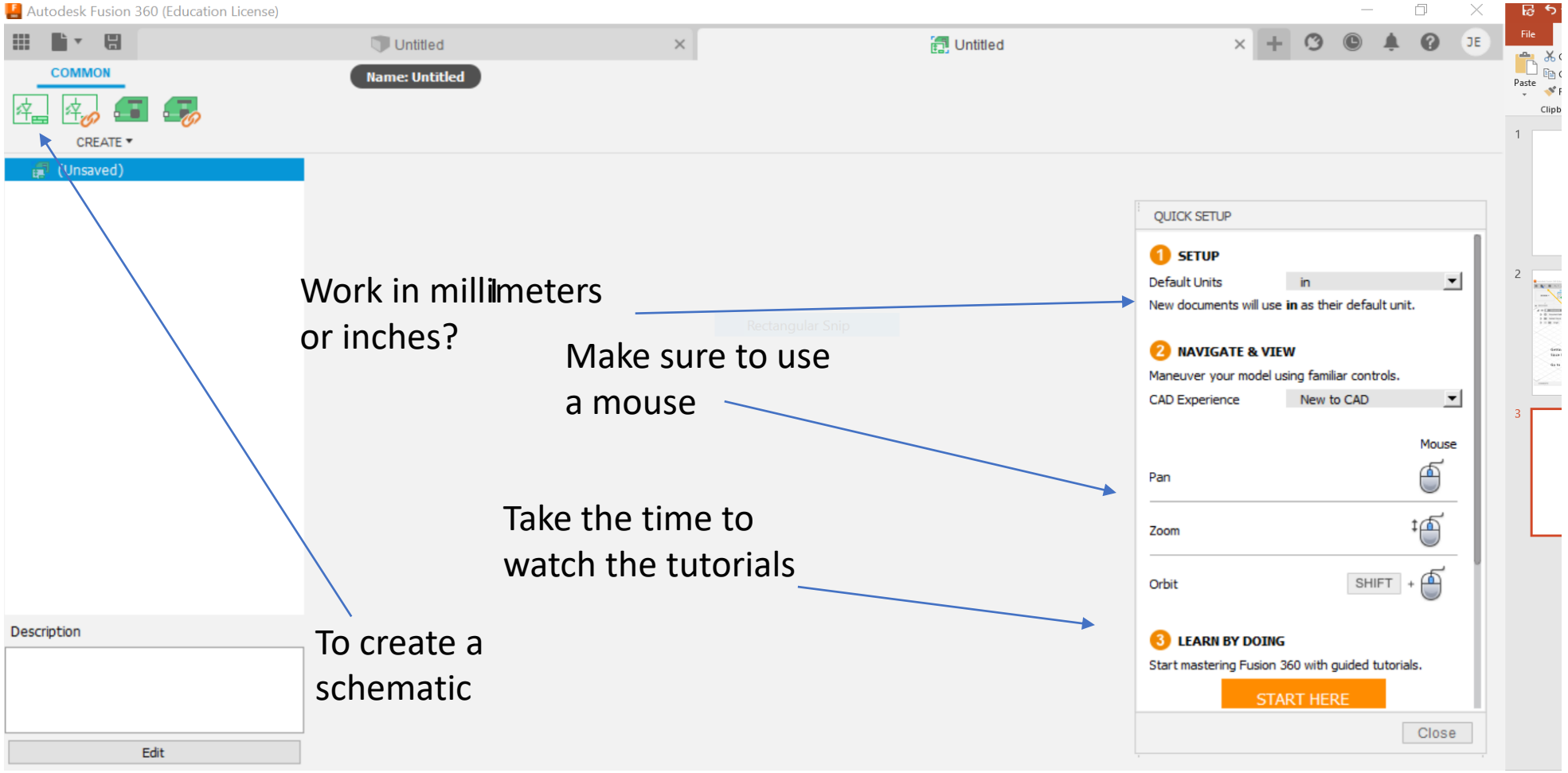
3) PCB Design- Convert your Schematic to PCB Design

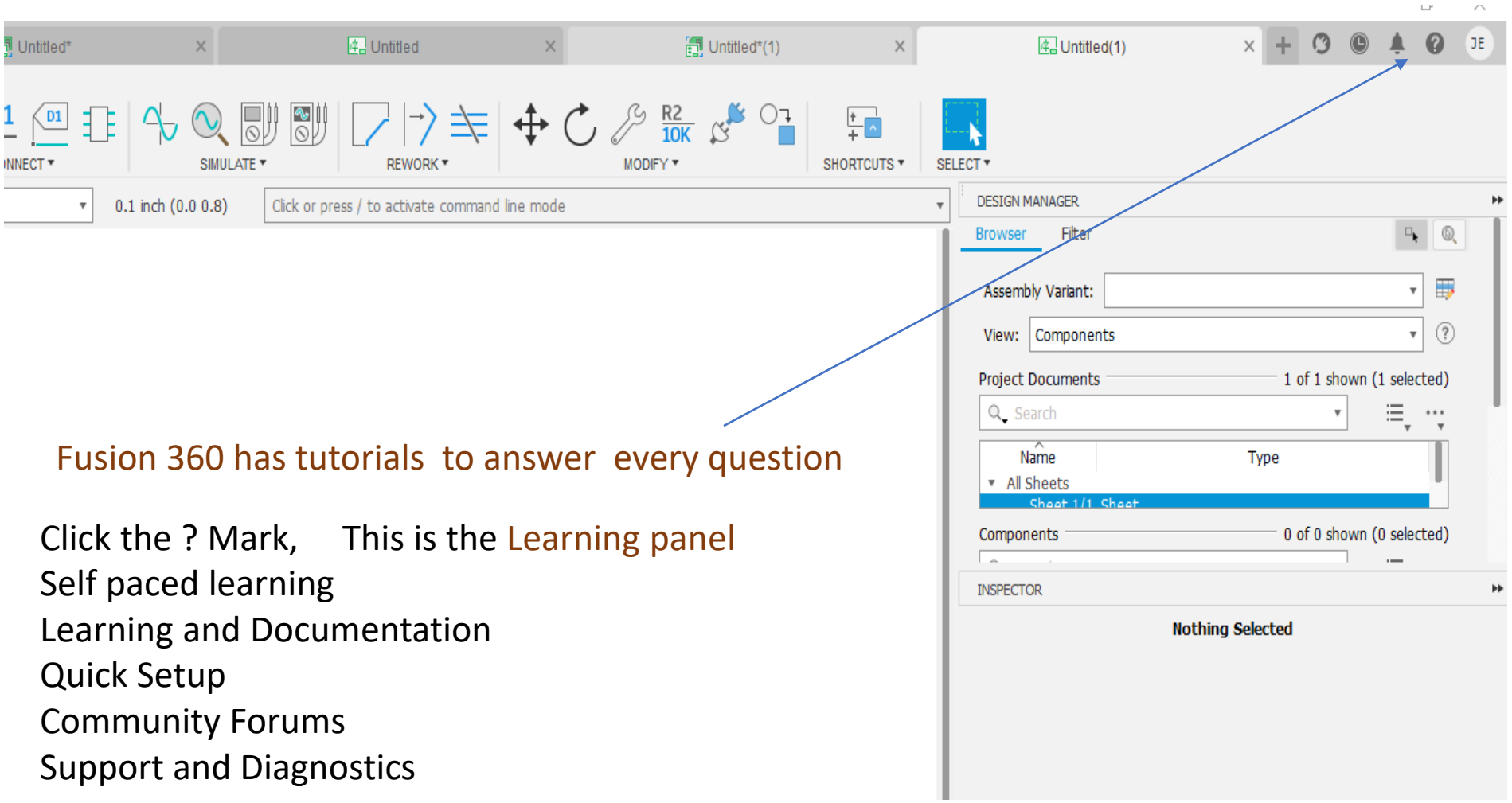
4) Create Gerber files for manufacturing- Drill and Cam files- as in a zip folder- labeled with your SD project #

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



This page opens up to start a schematic

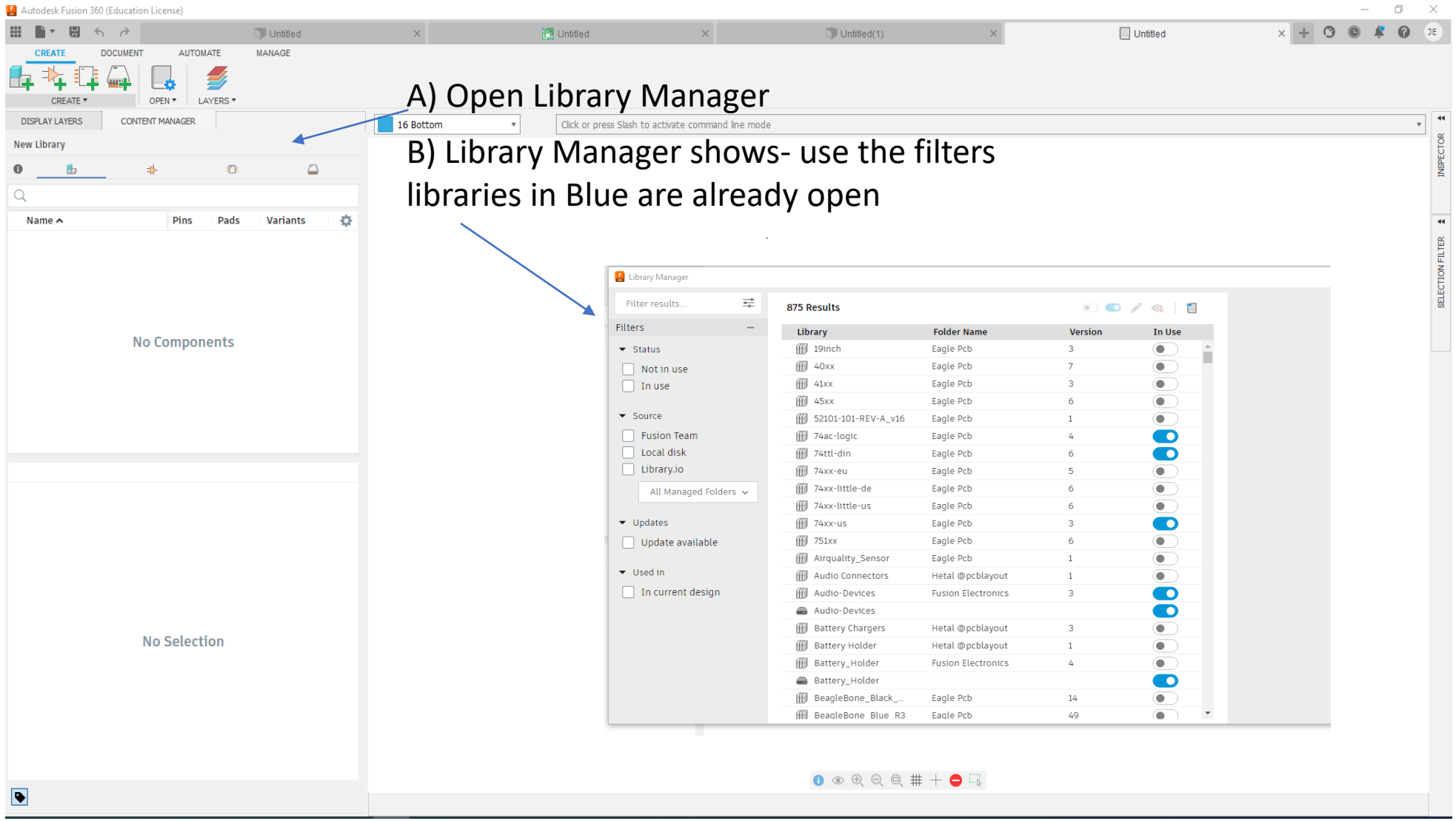




Fusion 360 has tutorials to answer every question

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

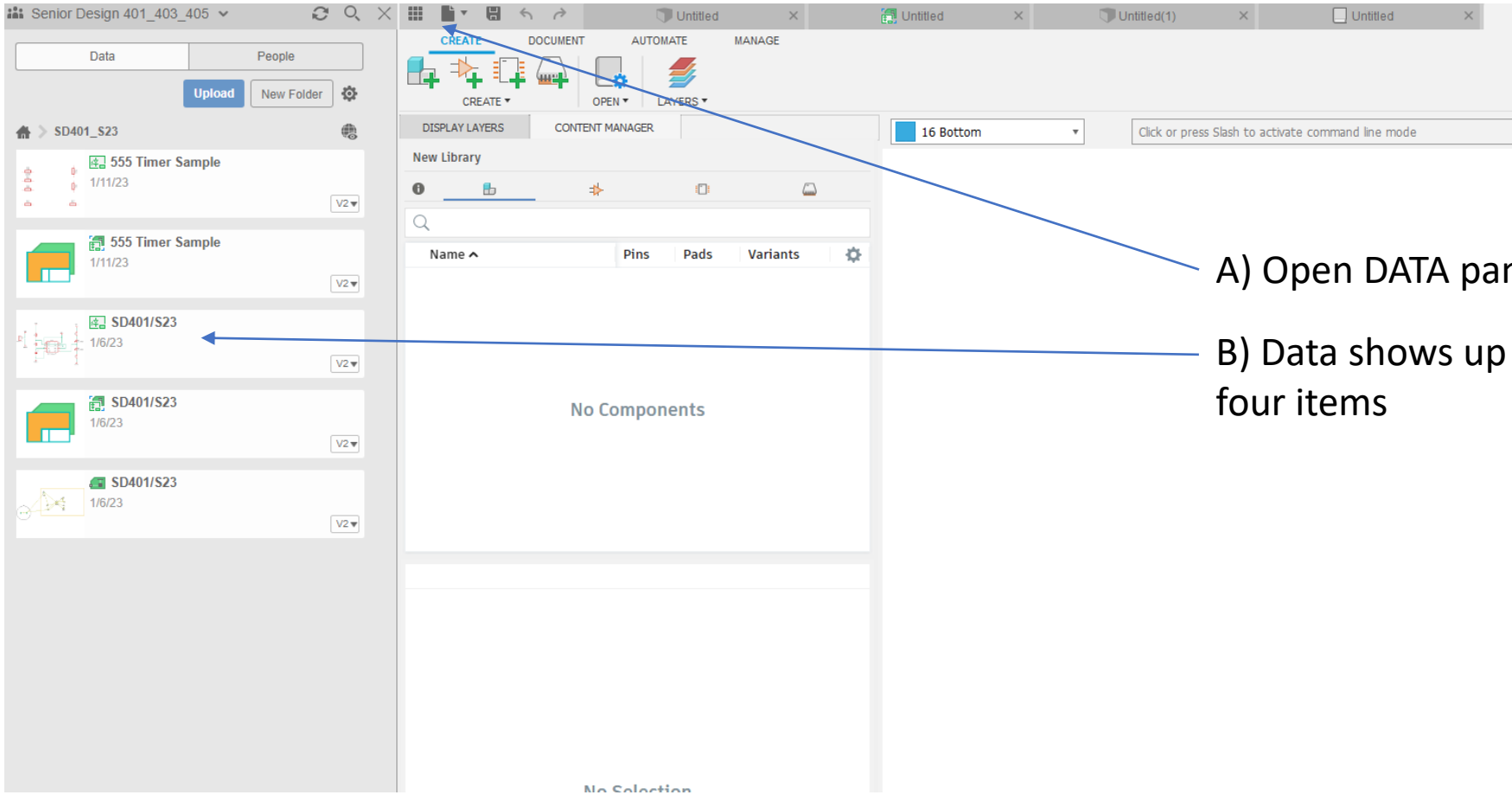
https://www.youtube.com/watch?v=_jgUZeBiusw&list=PLmA_xUT-8UlL80Xm8Gxz98YNum3I9GInr
Getting started by George Garcia of Fusion 360



Video to create your own Library by George Garcia- Fusion 360

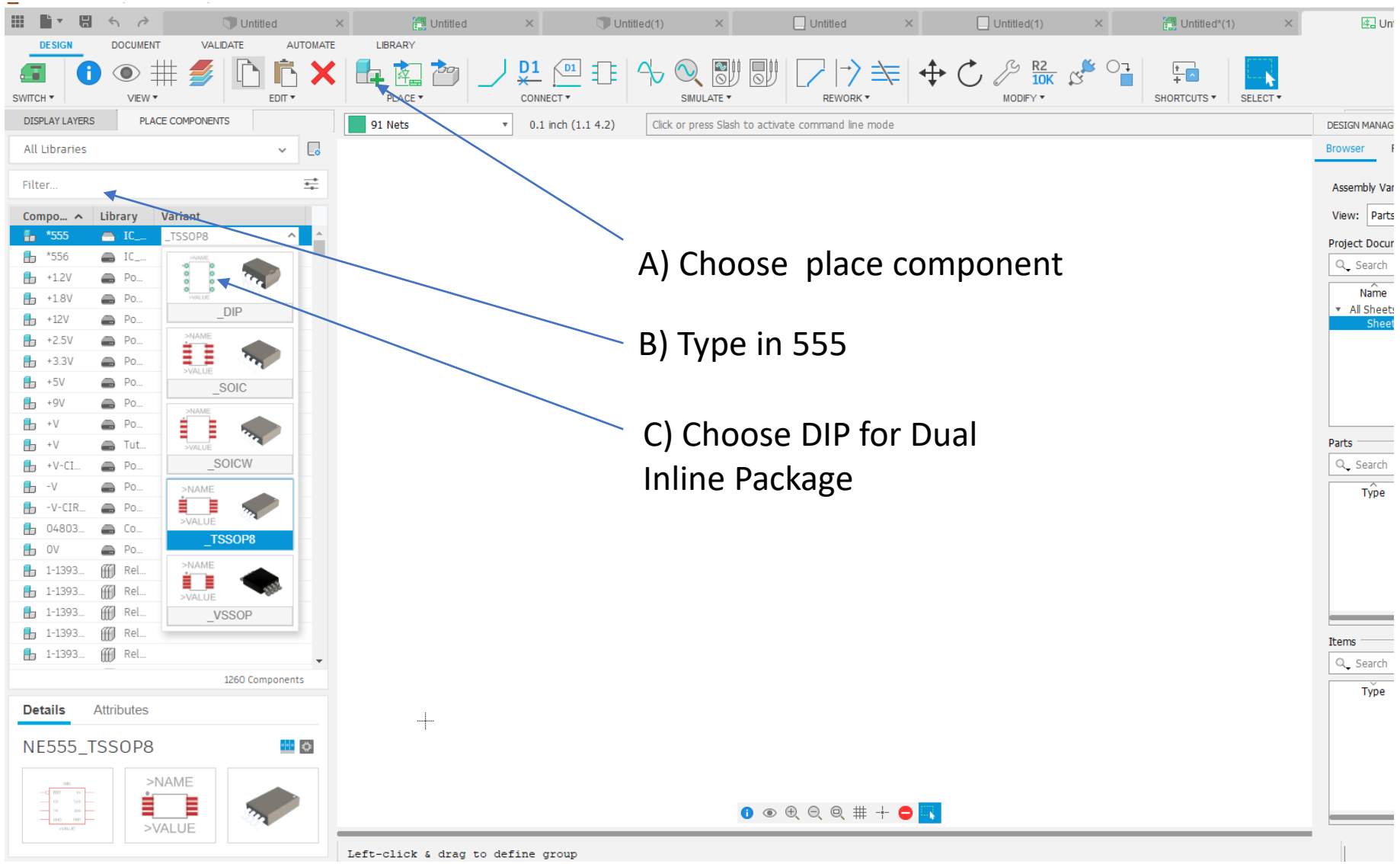
https://www.google.com/search?q=Create+your+own+librarie+in+Fusion360&rlz=1C1GCEB_enUS1034US1034&oq=Create+your+own+librarie+in+Fusion360&aqs=chrome..69i57j33i10i160j33i10i299.16607j0j7&sourceid=chrome&ie=UTF-8#fpstate=ive&vld=cid:635269fa,vid:xNIEXCimRSg,st:78

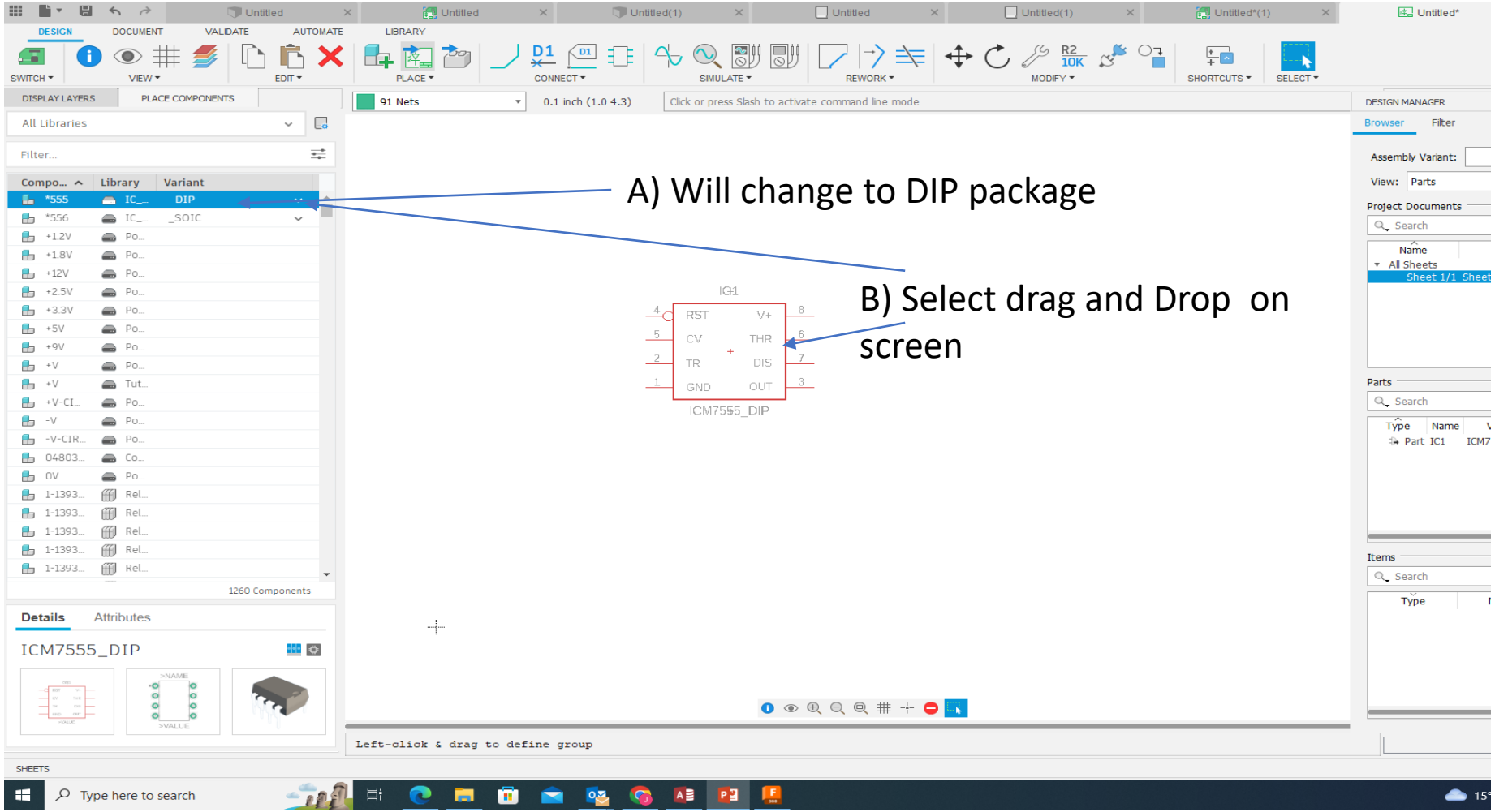
Fusion 360 Create Library
Fusion 360: Using Toolholder Solid Models in the CAM Tool Library - NYC CNC.
...
To create a new, custom-defined library:
Go to the Manufacture workspace.
Select Manage -> Tool Library.
Right-Click on the Local folder.
Select New Tool Library.

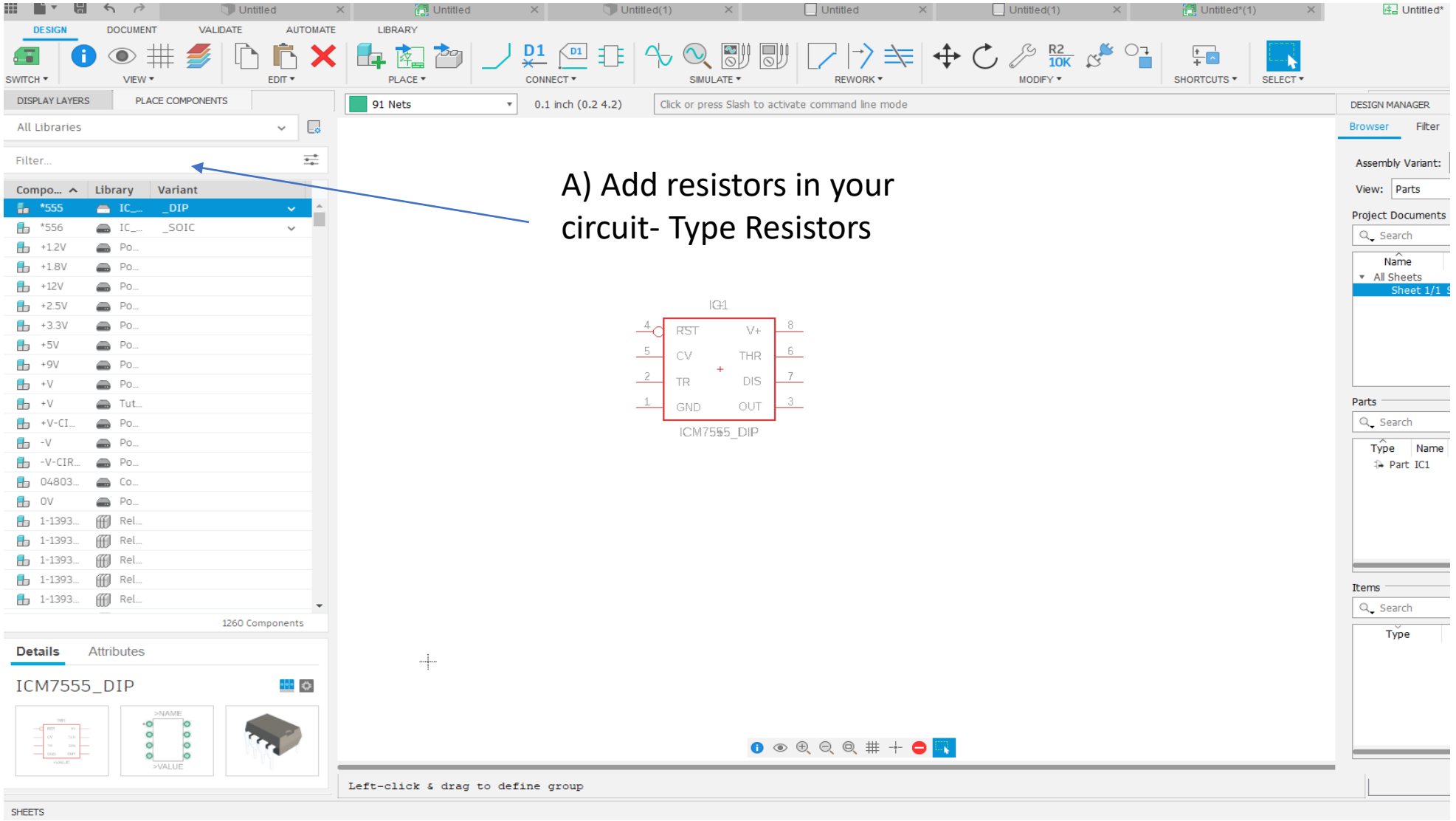


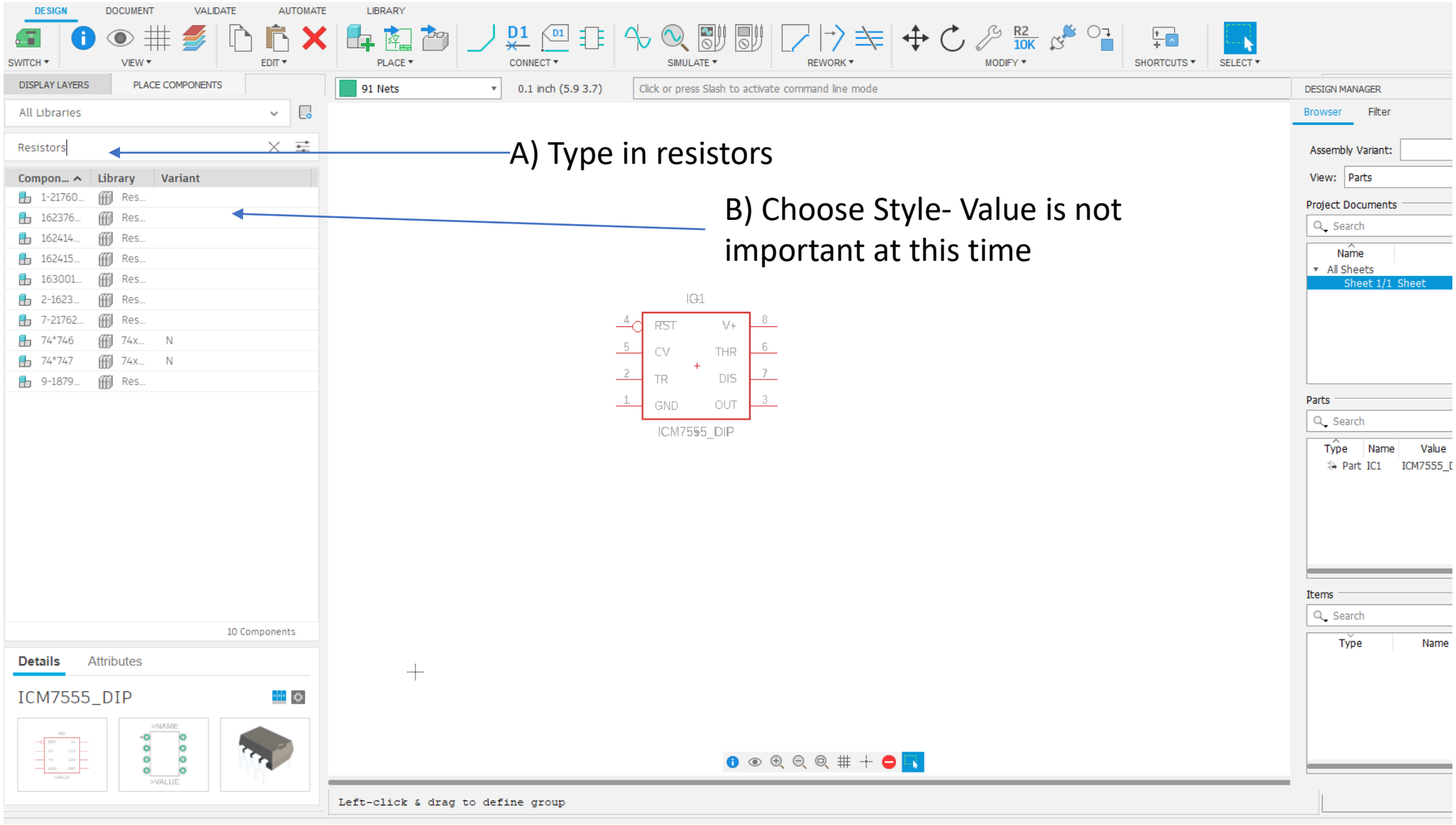
A) Open DATA panel

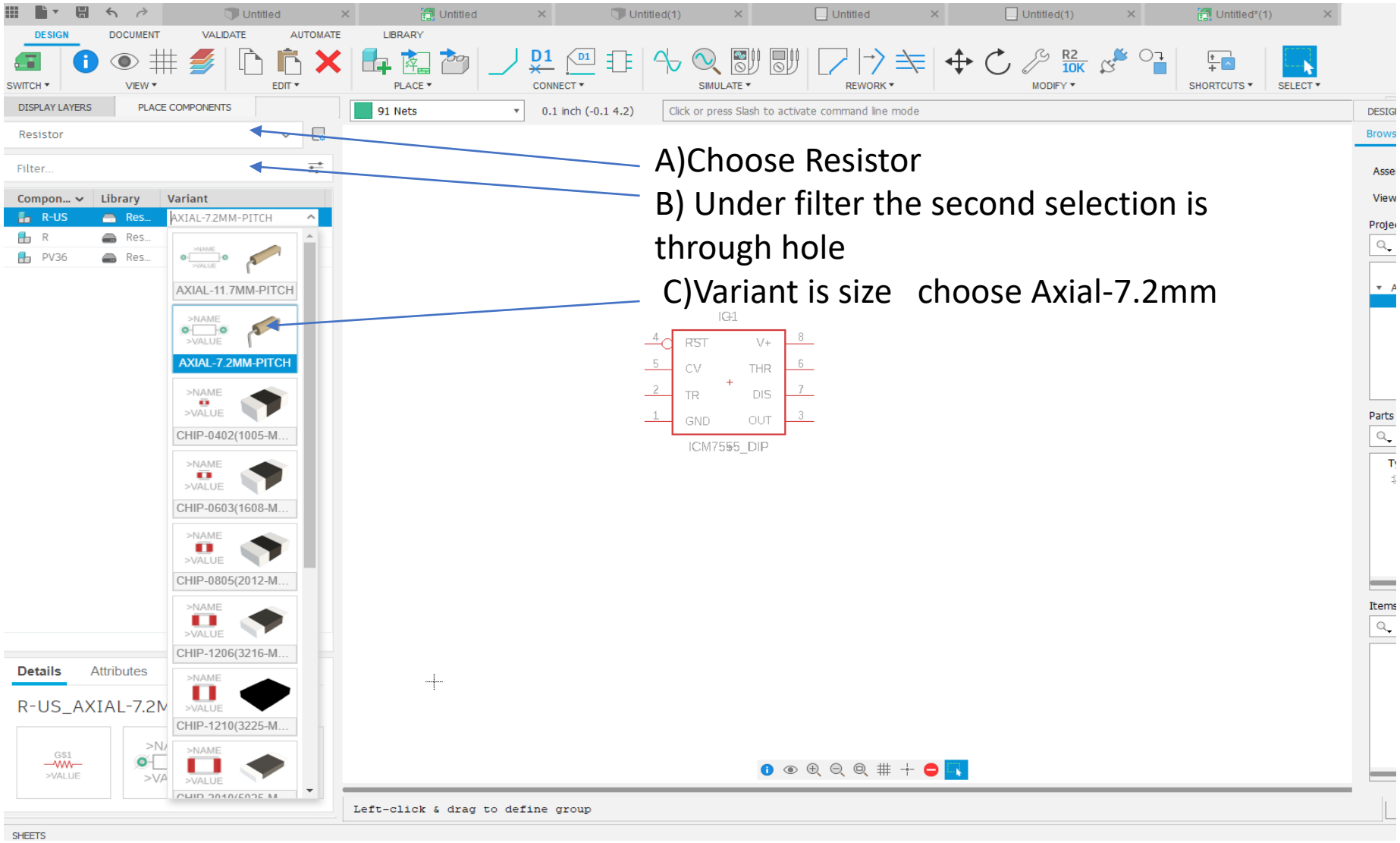
B) Data shows up showing four items


A) Choose place component
B) Type in 555
C) Choose DIP for Dual Inline Package

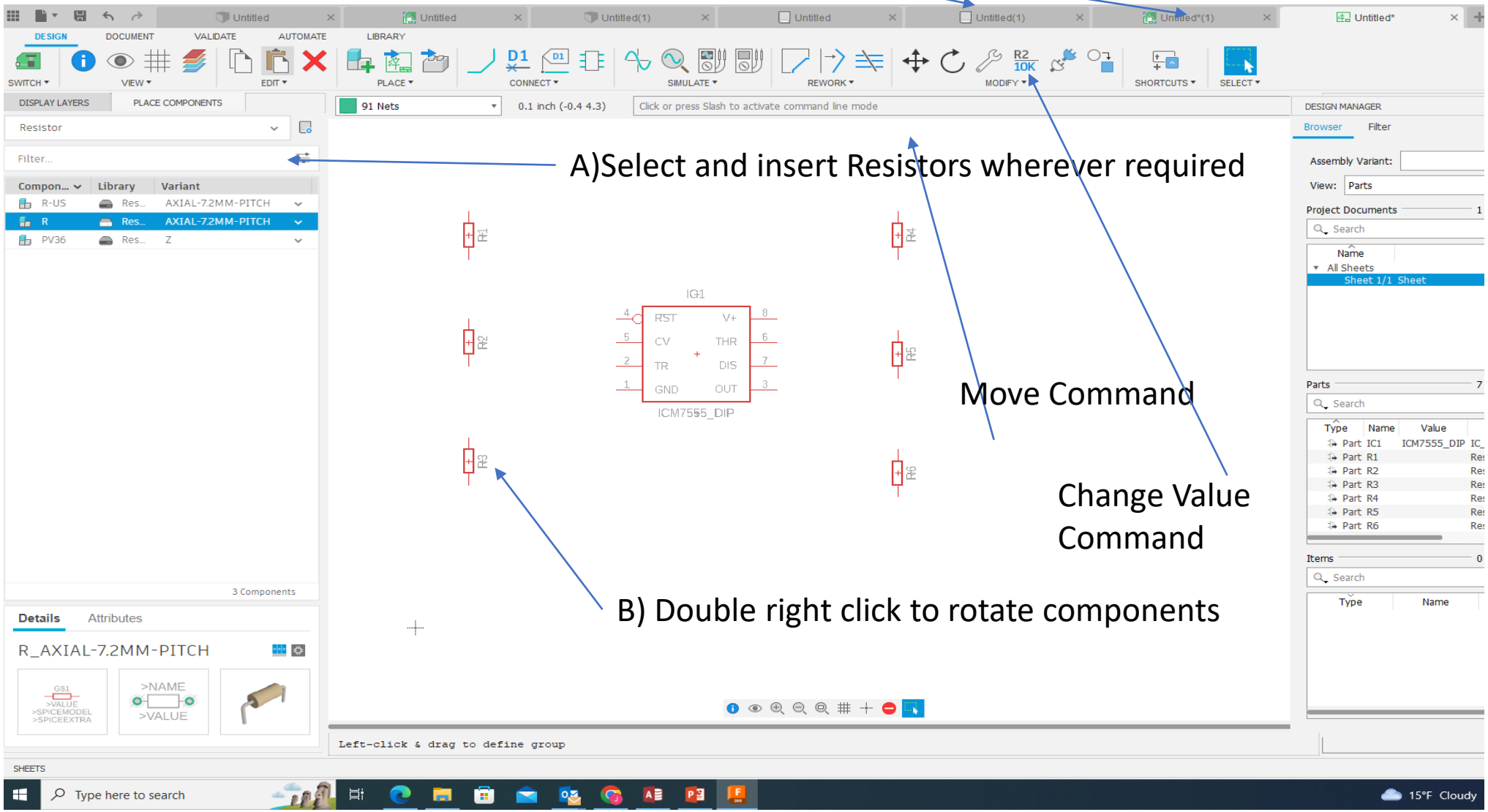

A) Will change to DIP package
B) Select drag and Drop on screen
IC1
RST
CV
TR
GND
V+
THR
DIS
OUT
ICM7555_DIP
Left-click & drag to define group

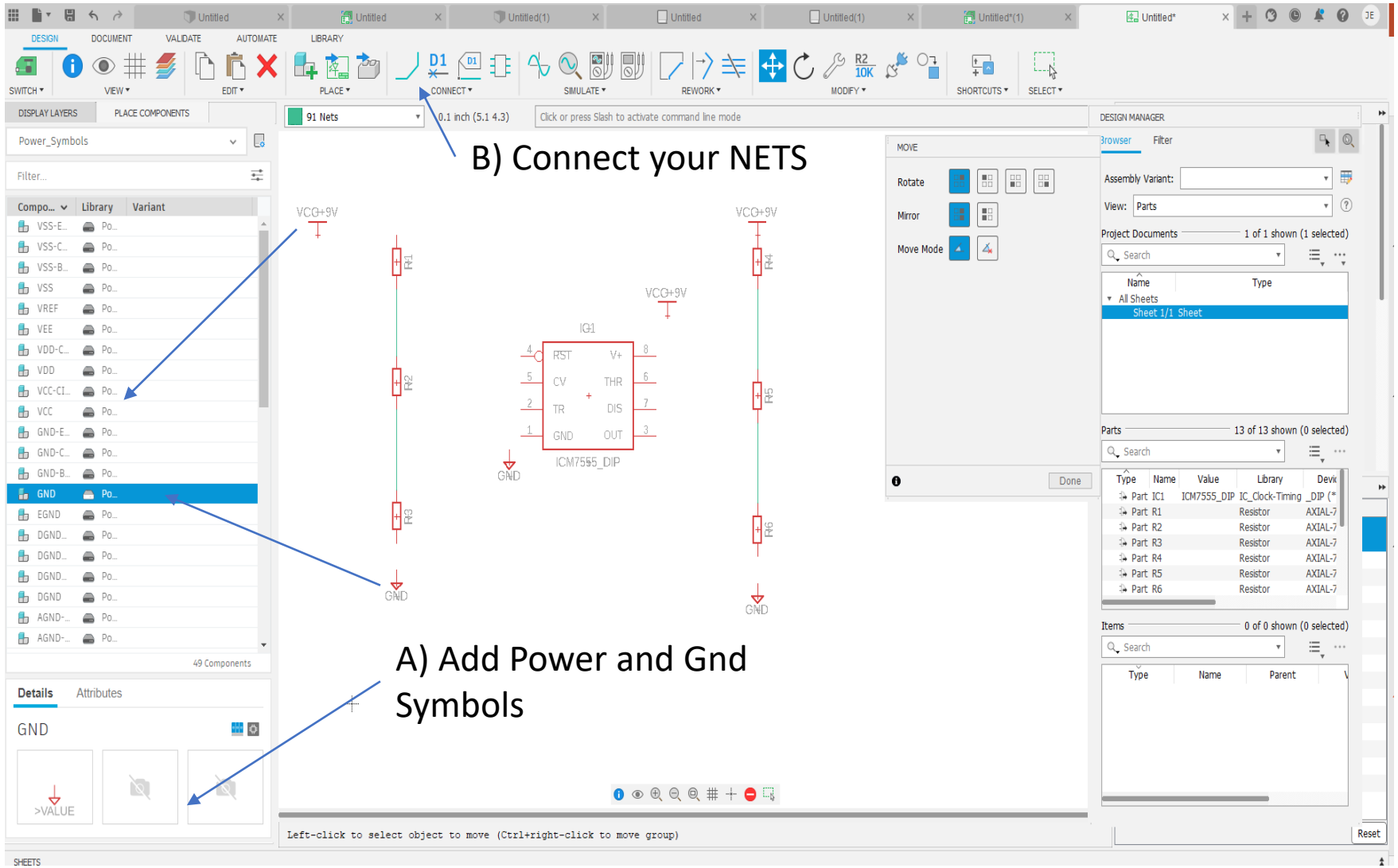

A) Add resistors in your circuit- Type Resistors
IC1
RST
V+
CV
THR
TR
DIS
GND
OUT
ICM7555_DIP


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
DESIGN MANAGER
Browser
Filter
All Libraries
Resistors
A) Type in resistors
B) Choose Style- Value is not important at this time
Compon...
Library
Variant
1-21760...
162376...
162414...
162415...
163001...
2-1623...
7-21762...
74*746
74*747
9-1879...
10 Components
Details
Attributes
ICM7555_DIP
IC1
RST
V+
CV
THR
TR
DIS
GND
OUT
ICM7555_DIP
Assembly Variant:
View:
Parts
Project Documents
Search
Name
All Sheets
Sheet 1/1 Sheet
Parts
Type
Name
Value
Part
IC1
Items
Left-click & drag to define group


A)Choose Resistor
B) Under filter the second selection is through hole
C)Variant is size choose Axial-7.2mm
Resistor
Filter...
Compon...
Library
Variant
R-US
R
PV36
AXIAL-7.2MM-PITCH
AXIAL-11.7MM-PITCH
AXIAL-7.2MM-PITCH
CHIP-0402(1005-M...
CHIP-0603(1608-M...
CHIP-0805(2012-M...
CHIP-1206(3216-M...
CHIP-1210(3225-M...
Details
Attributes
R-US_AXIAL-7.2M
IC1
RST
V+
CV
THR
TR
DIS
GND
OUT
ICM7555_DIP
91 Nets
0.1 inch (-0.1 4.2)
Click or press Slash to activate command line mode
Left-click & drag to define group
SHEETS

Never Delete these- always need three

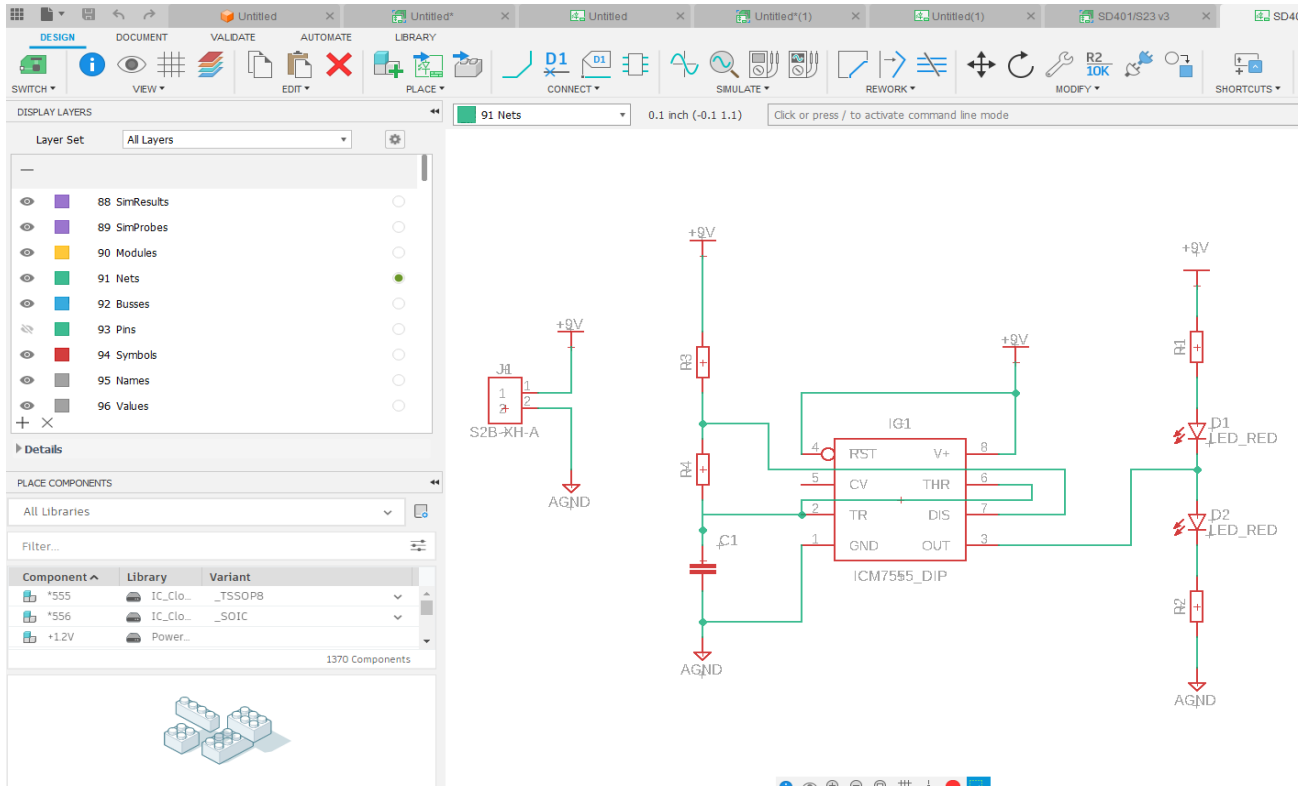



B) Connect your NETS
A) Add Power and Gnd Symbols
VCC+9V
GND
IC1
ICM7555_DIP
RST
CV
TR
GND
V+
THR
DIS
OUT
R1
R2
R3
R4
R5
R6
Power_Symbols
91 Nets
0.1 inch (5.1 4.3)
Click or press Slash to activate command line mode
DESIGN MANAGER
MOVE
Rotate
Mirror
Move Mode
Done
Left-click to select object to move (Ctrl+right-click to move group)
Reset

The first Part of PCB Manufacturing is creating a schematic.
Second part is the PCB itself which requires Gerber Files

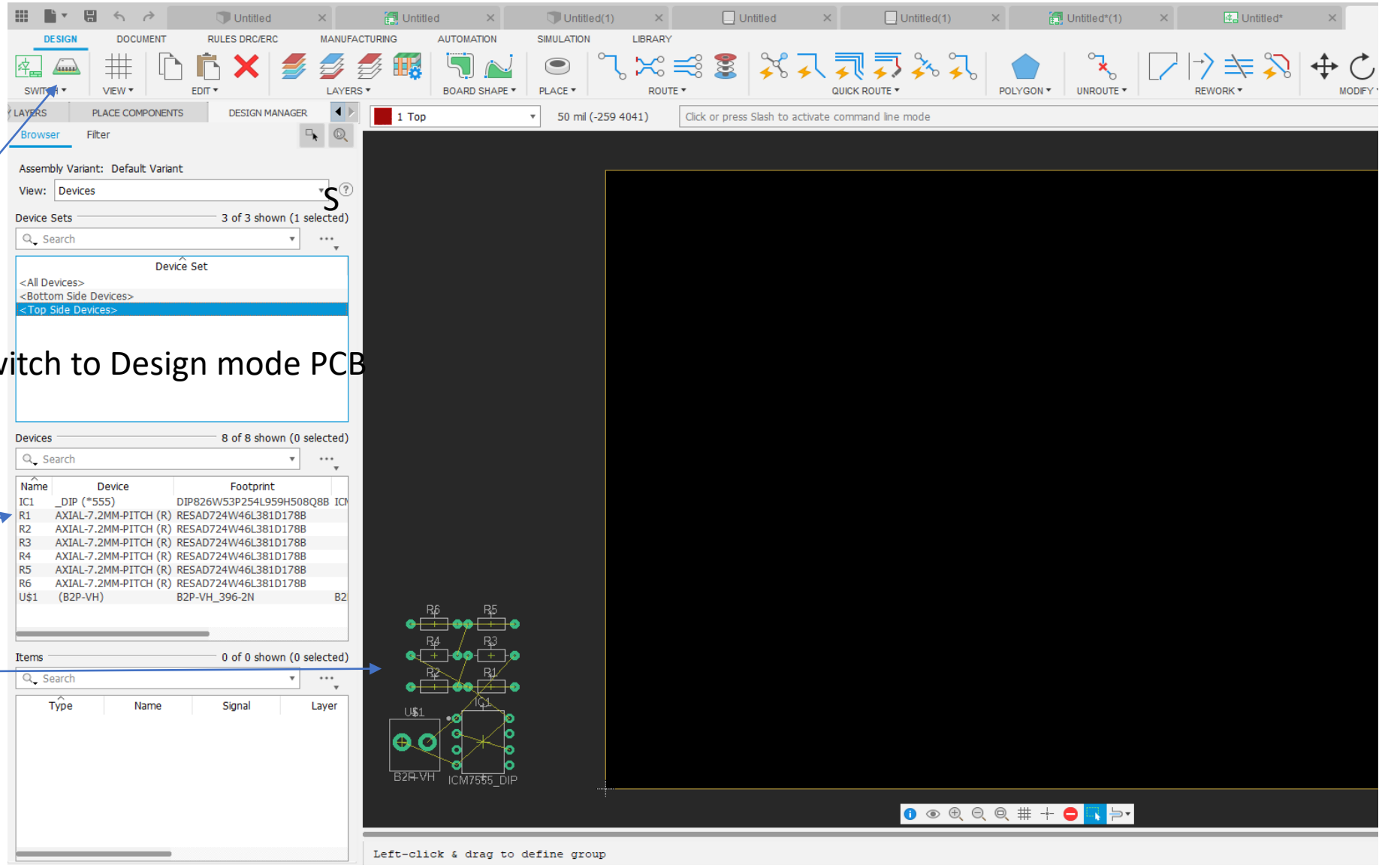
If your curious what is next, on the next slide look at Design >Switch> Schematic to PCB

Once Schematic completed switch to PCB Design



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

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



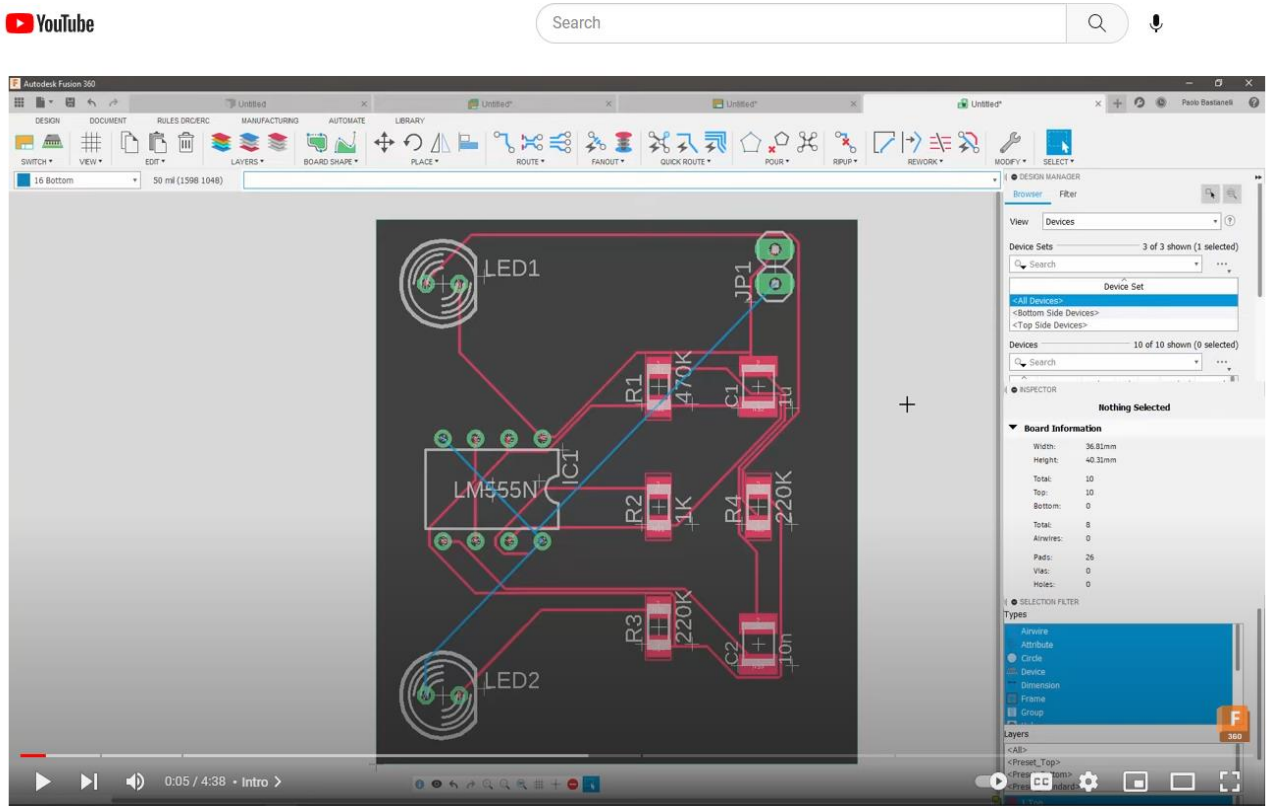
Switch to Design mode PCB

List of components in Design

2D Sim of components in design

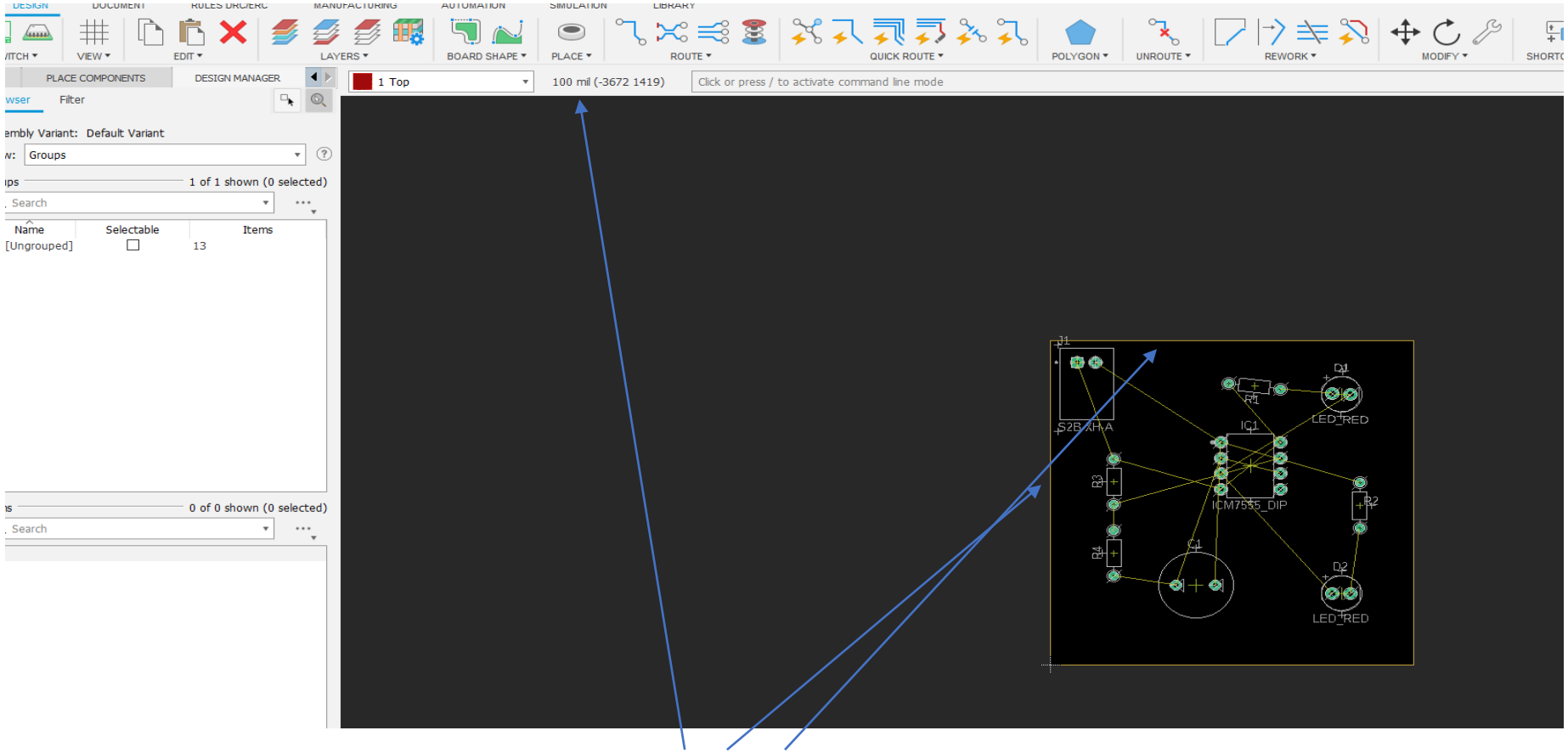
Part 2 Creating Circuit Board Layout

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



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

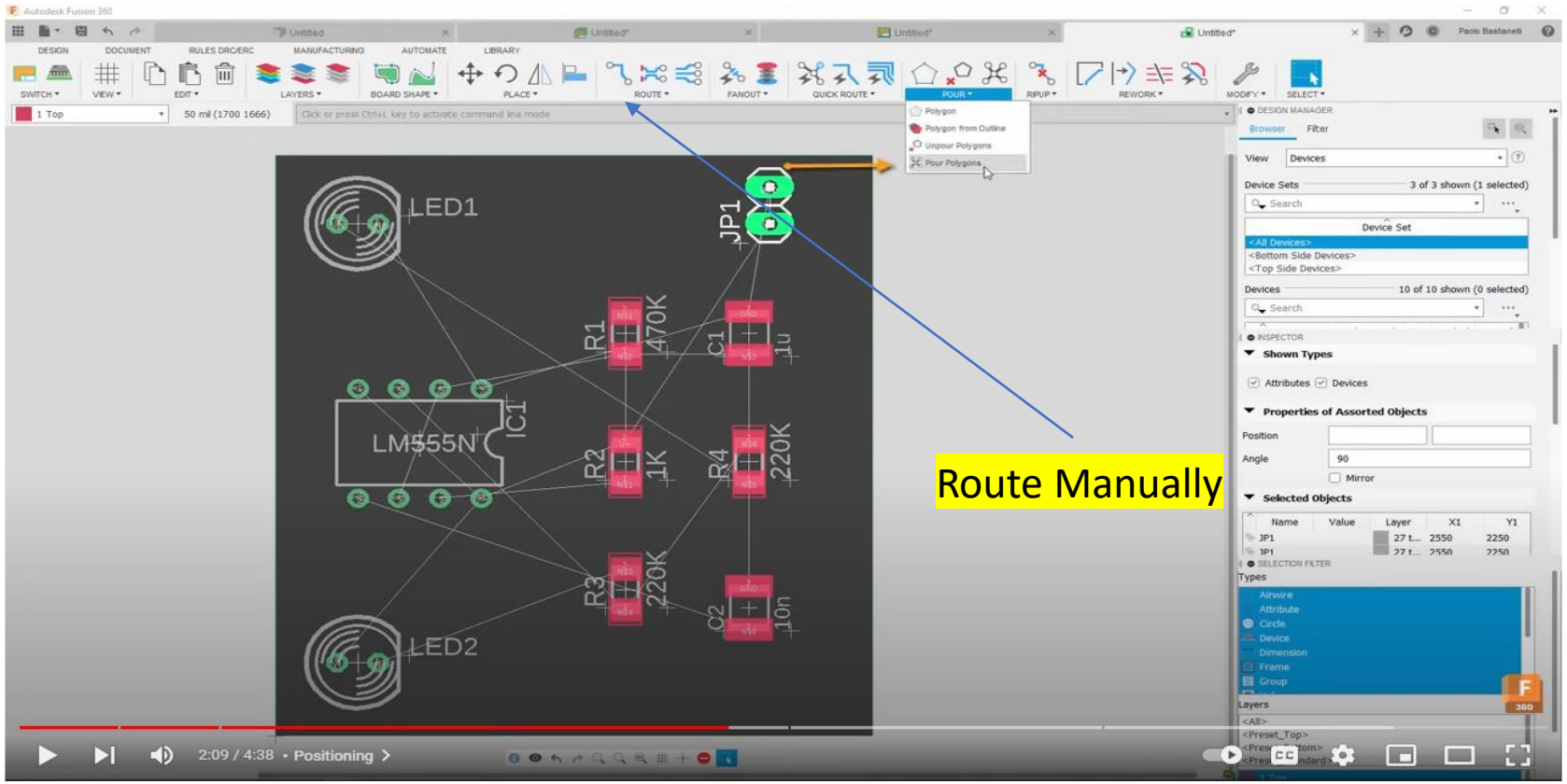
2D sim Components have been dragged into the Black square this is your PCB out line



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

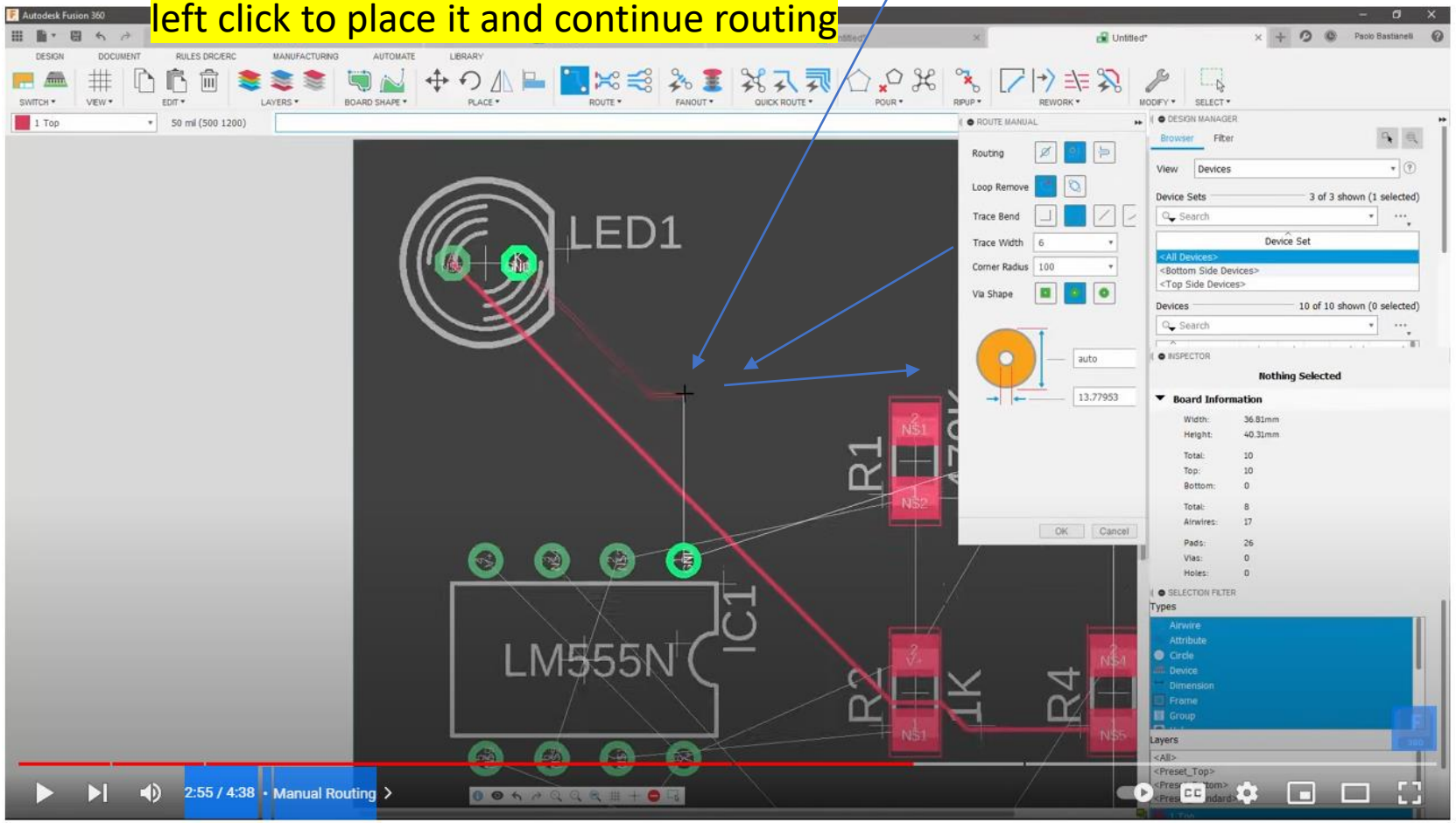
PCB Layout Tutorial Walkthrough – YouTube 0:42/4:38

To route traces manually and then automatically



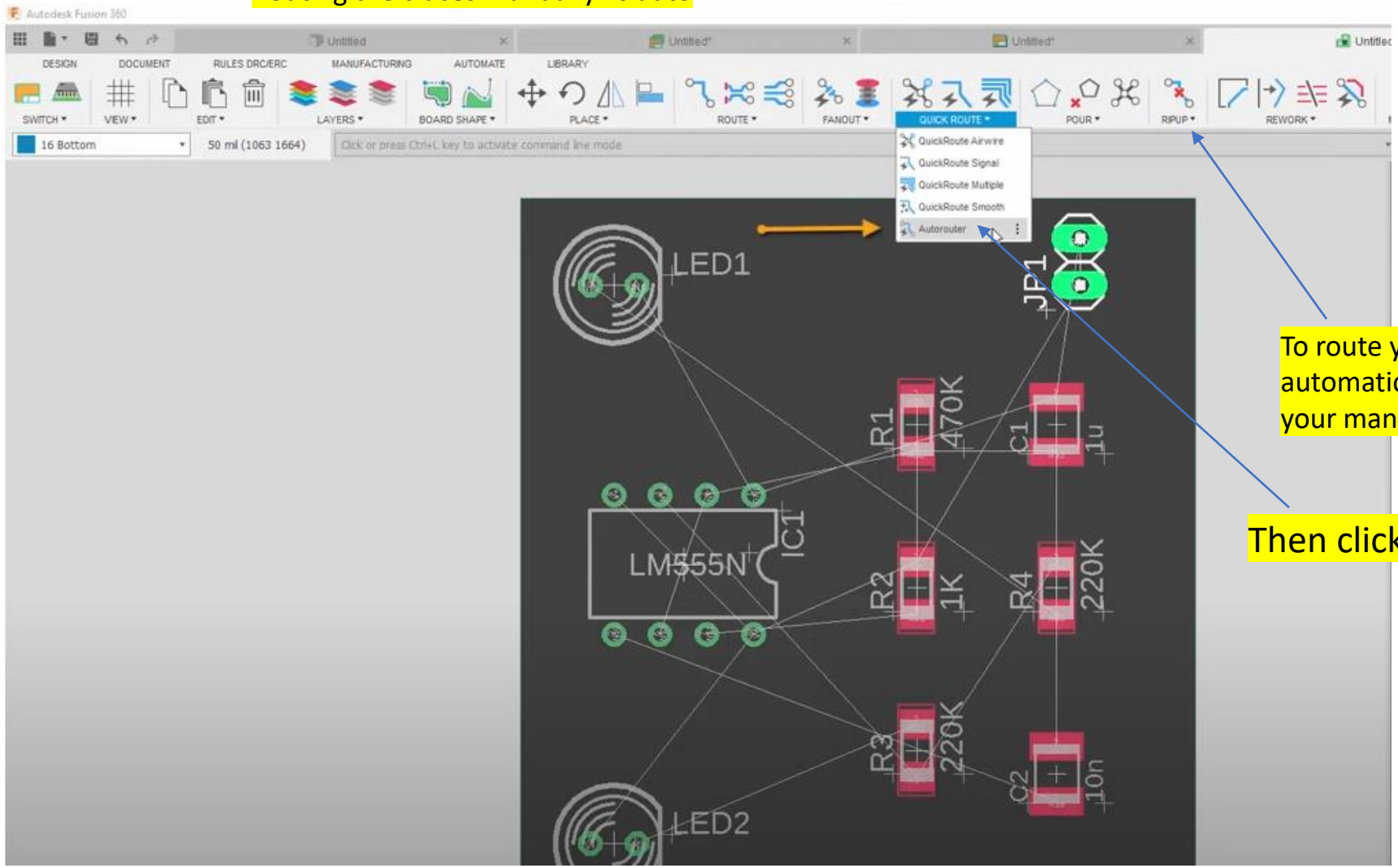
[PCB Layout Tutorial Walkthrough – YouTube] 2:32 mark

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

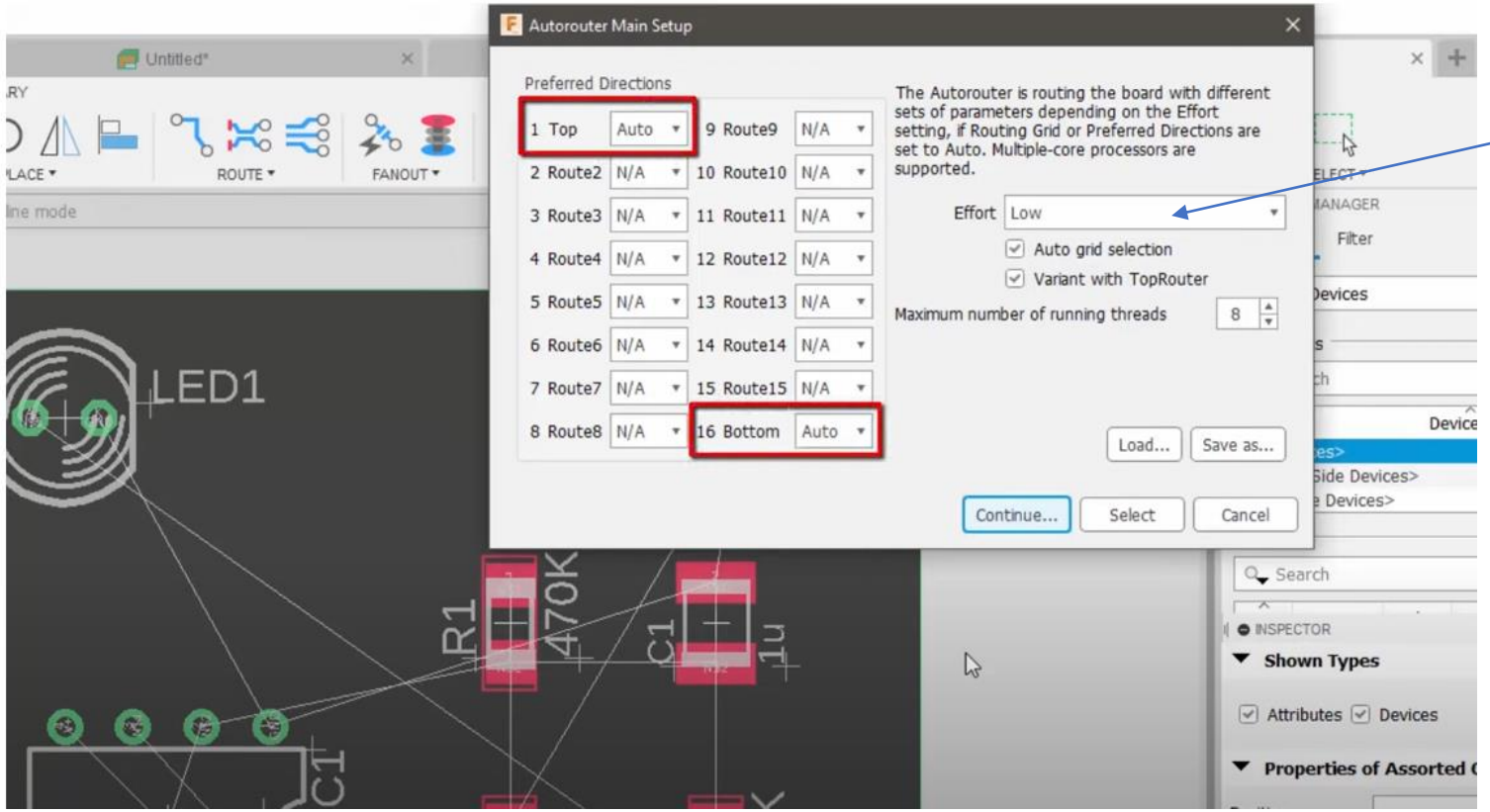


To route your traces automatically you may rip up your manual ones first.

Then click auto router

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

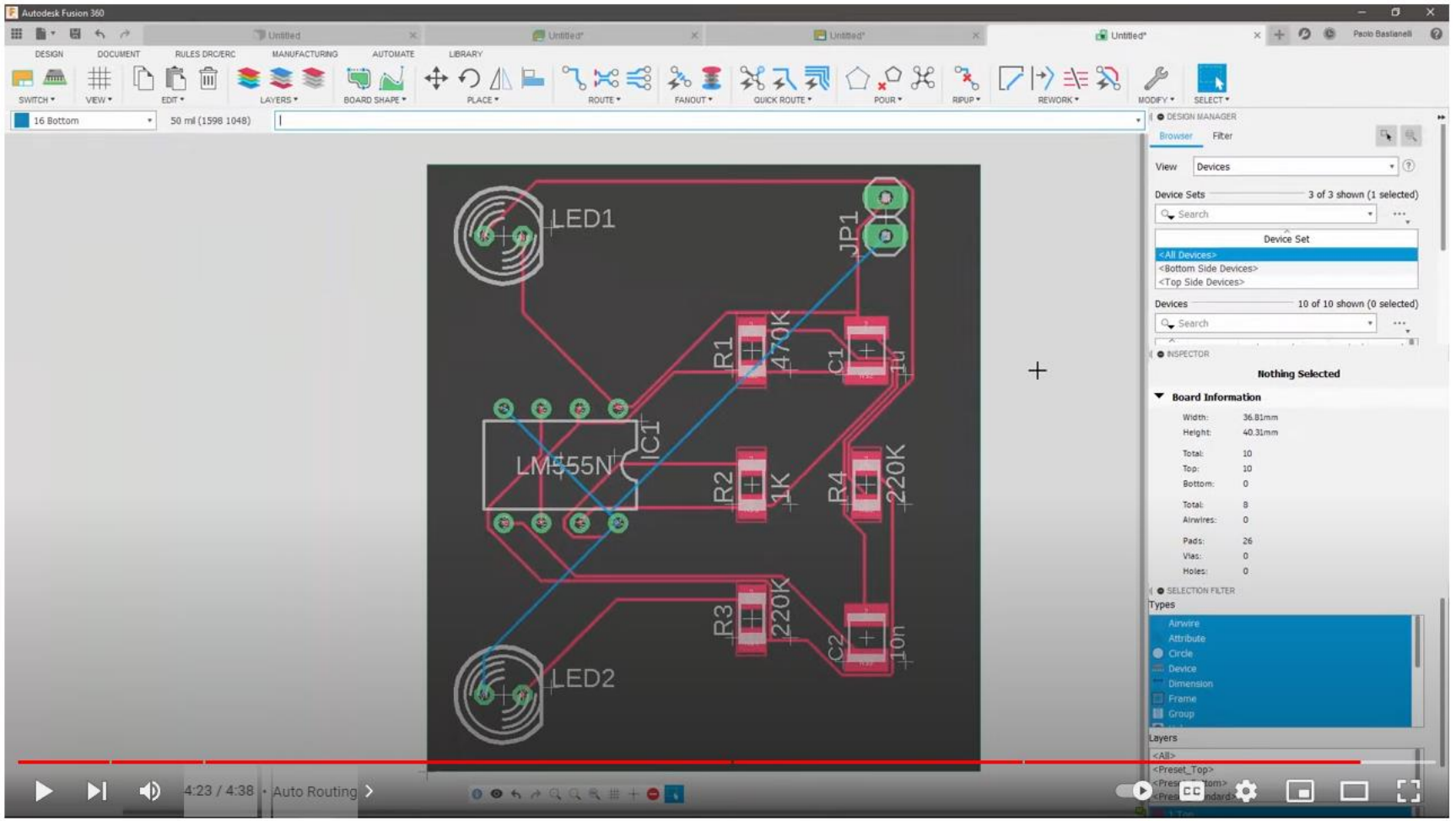
Two layer boards only top and bottom will be selected to auto route



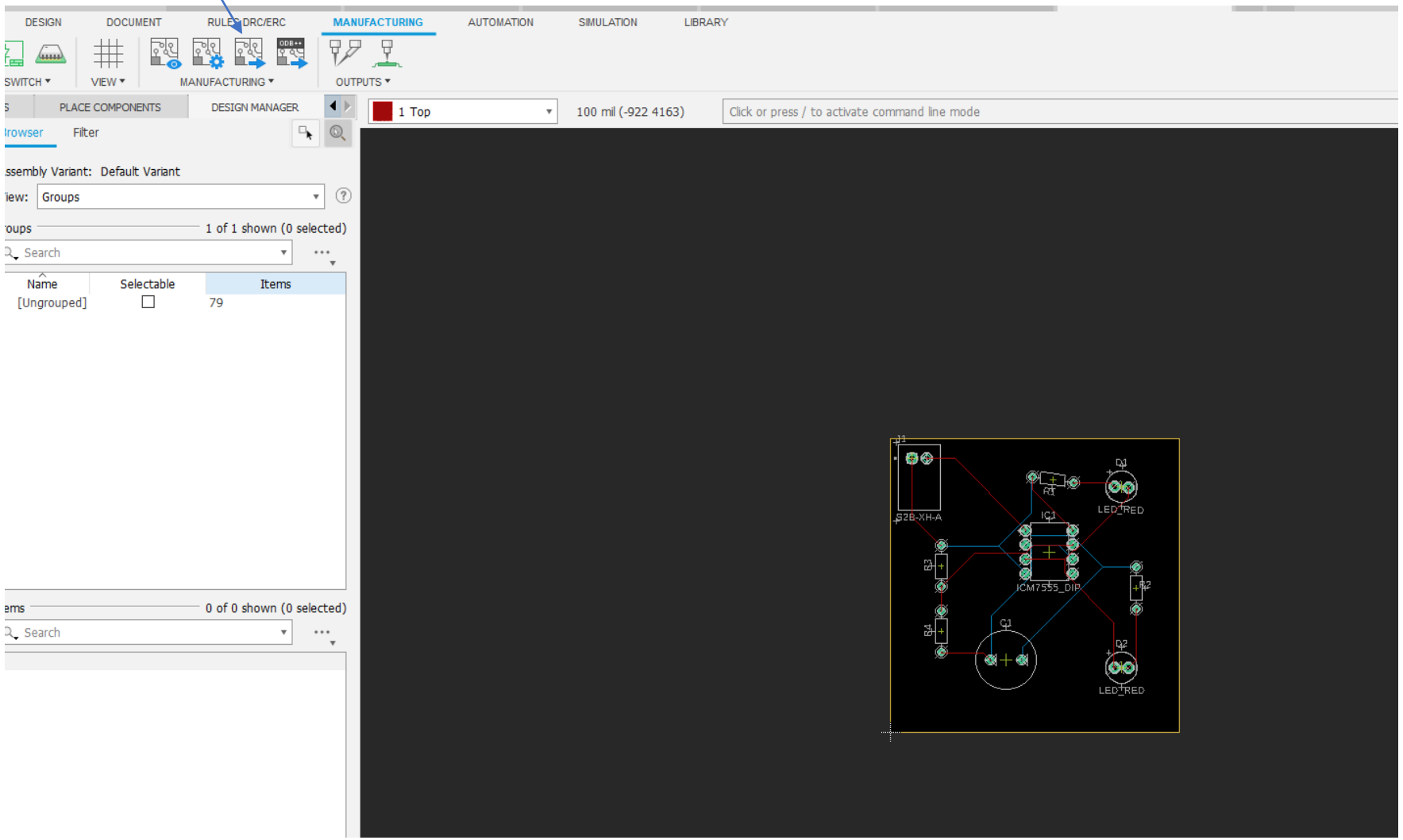
Change the effort to high to get a better selection of solutions

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

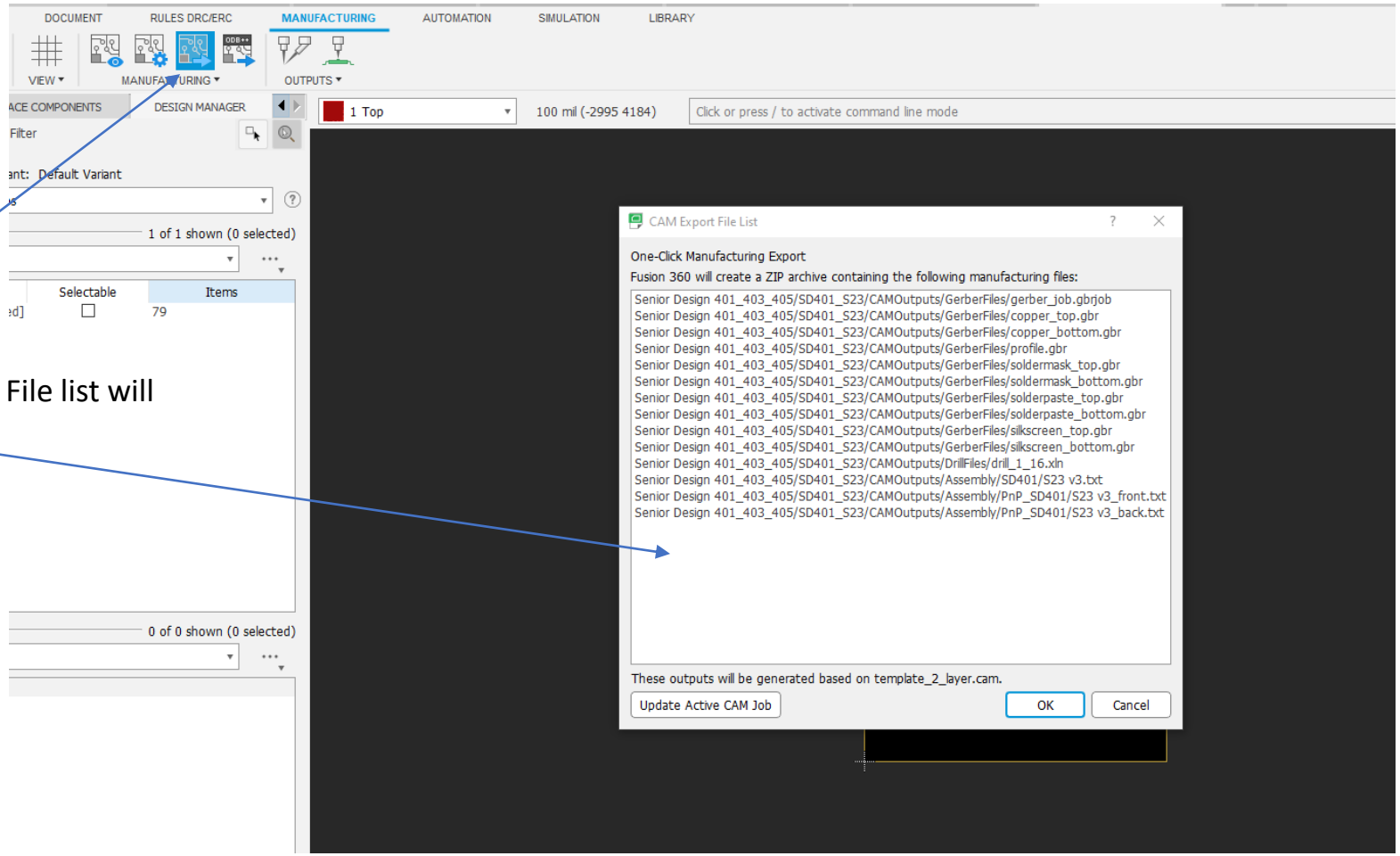
After choosing the solution with the least amount vias, the PCB is complete

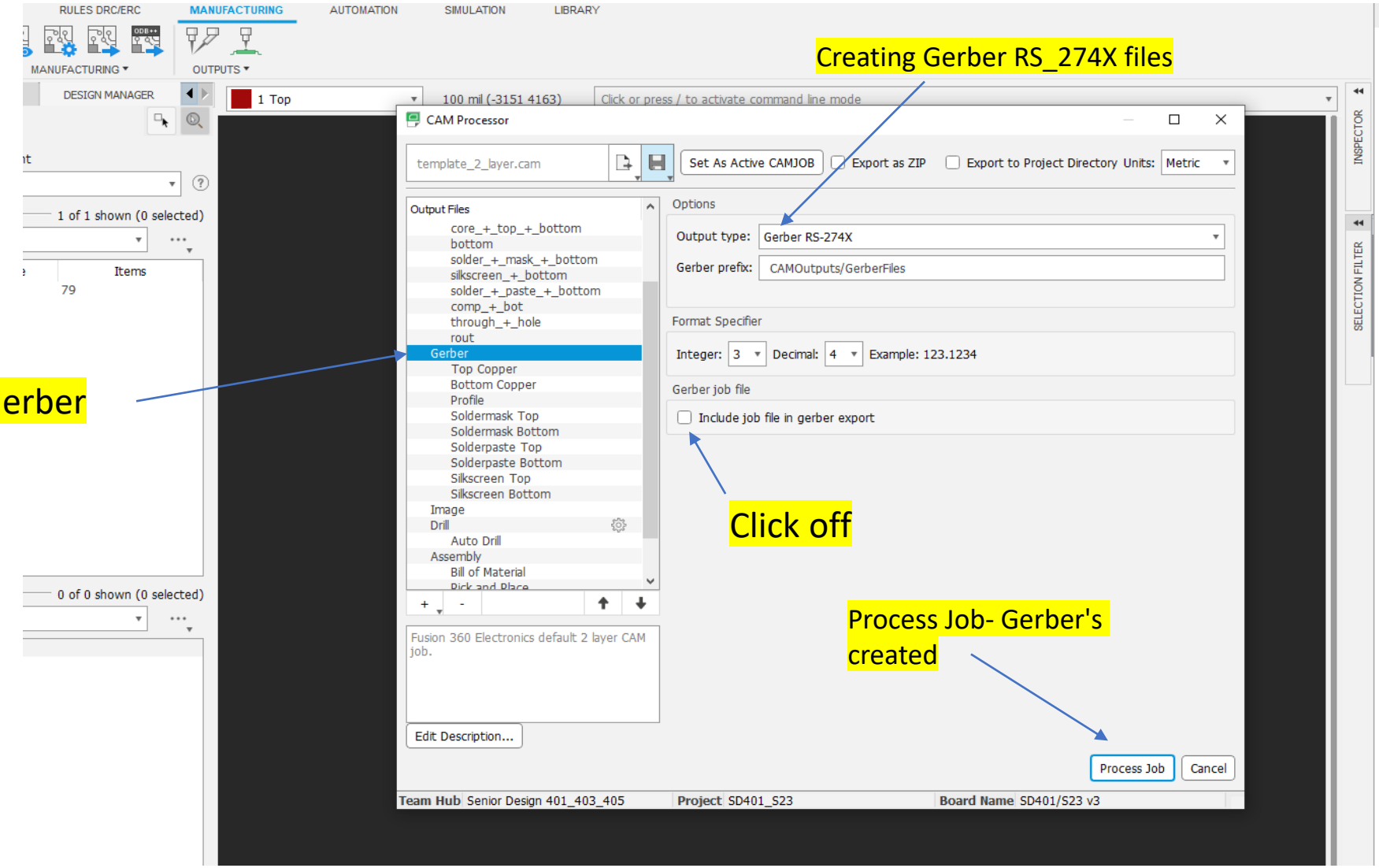


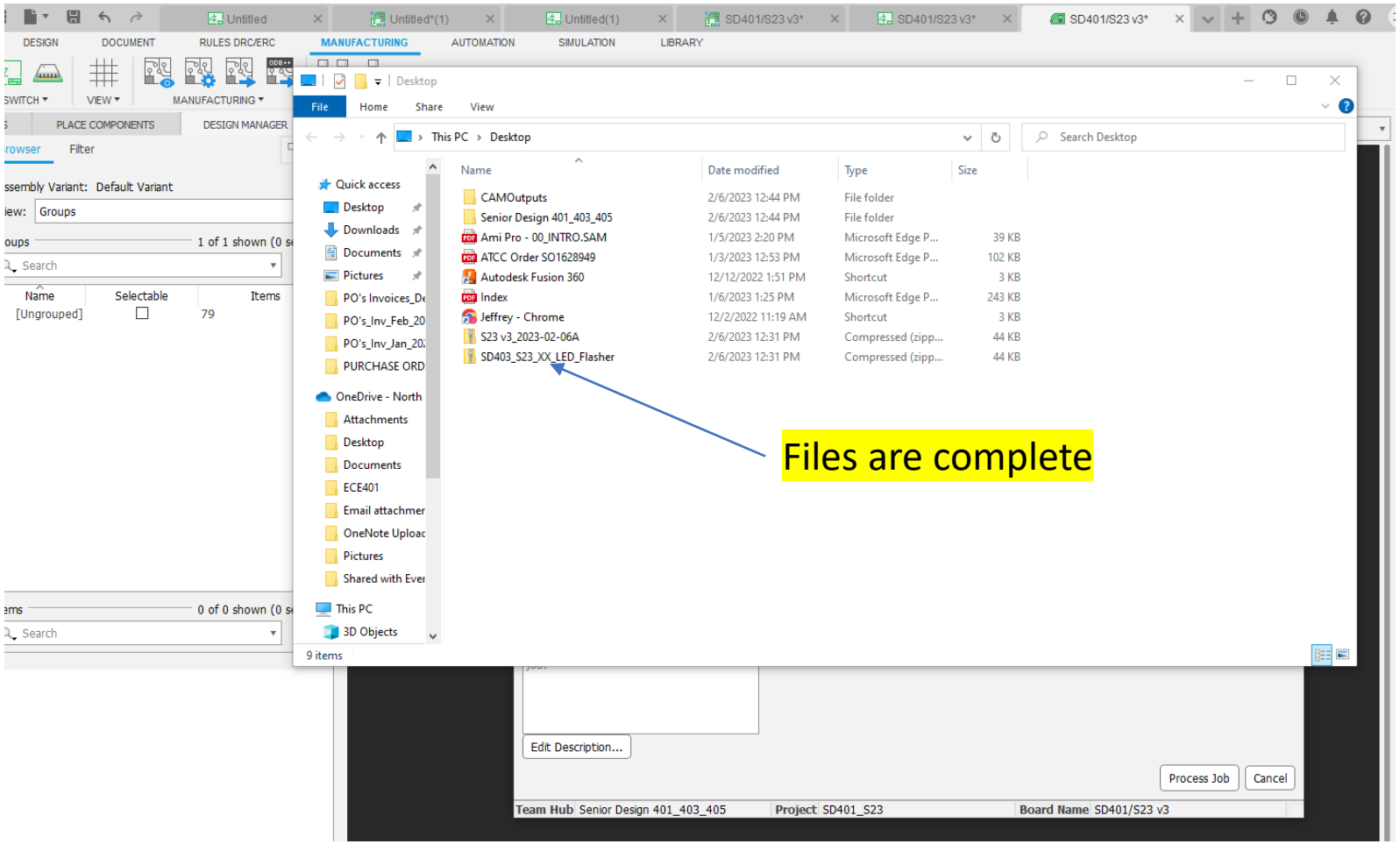
PCB Manufacturing- Creating Gerber Files



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
RULES DRC/ERC
MANUFACTURING
AUTOMATION
SIMULATION
LIBRARY
OUTPUTS
DESIGN MANAGER
1 Top
100 mil (-3151 4163)
Click or press / to activate command line mode
CAM Processor
template_2_layer.cam
Set As Active CAMJOB
Export as ZIP
Export to Project Directory
Units:
Metric
Output Files
core_+_top_+_bottom
bottom
solder_+_mask_+_bottom
silkscreen_+_bottom
solder_+_paste_+_bottom
comp_+_bot
through_+_hole
rout
Gerber
Top Copper
Bottom Copper
Profile
Soldermask Top
Soldermask Bottom
Solderpaste Top
Solderpaste Bottom
Silkscreen Top
Silkscreen Bottom
Image
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
Fusion 360 Electronics default 2 layer CAM job.
Edit Description...
Process Job
Cancel
Team Hub
Senior Design 401_403_405
Project
SD401_S23
Board Name
SD401/S23 v3
1 of 1 shown (0 selected)
Items
79
0 of 0 shown (0 selected)
INSPECTOR
SELECTION FILTER



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

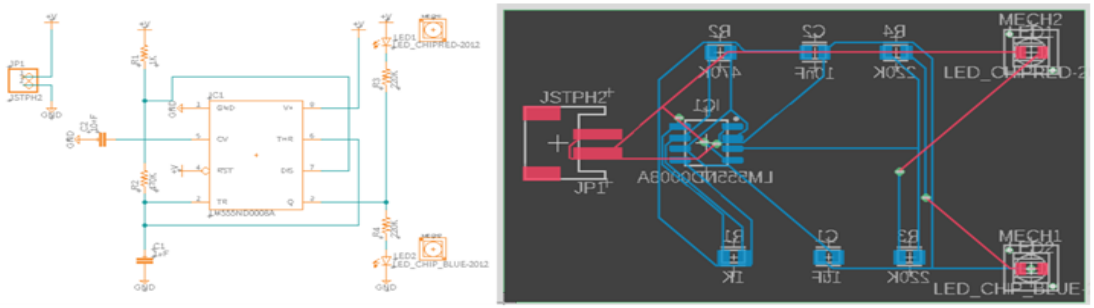
Electronics / Tutorials / Tutorial: Create a PCB layout

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 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

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

Creating Gerber files