


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_deeplink&cjevent=909e6a5fa63211ed8367bd120a1c0e0b&affname=100357191_10282382

[Download Fusion 360 for Free | Free Trial | Autodesk](https://www.autodesk.com/products/fusion-360/free-trial)

<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=lqwHkB9lsUo>

Fusion360 PCB Design Tutorial

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

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=lqwHkB9lsUo>

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

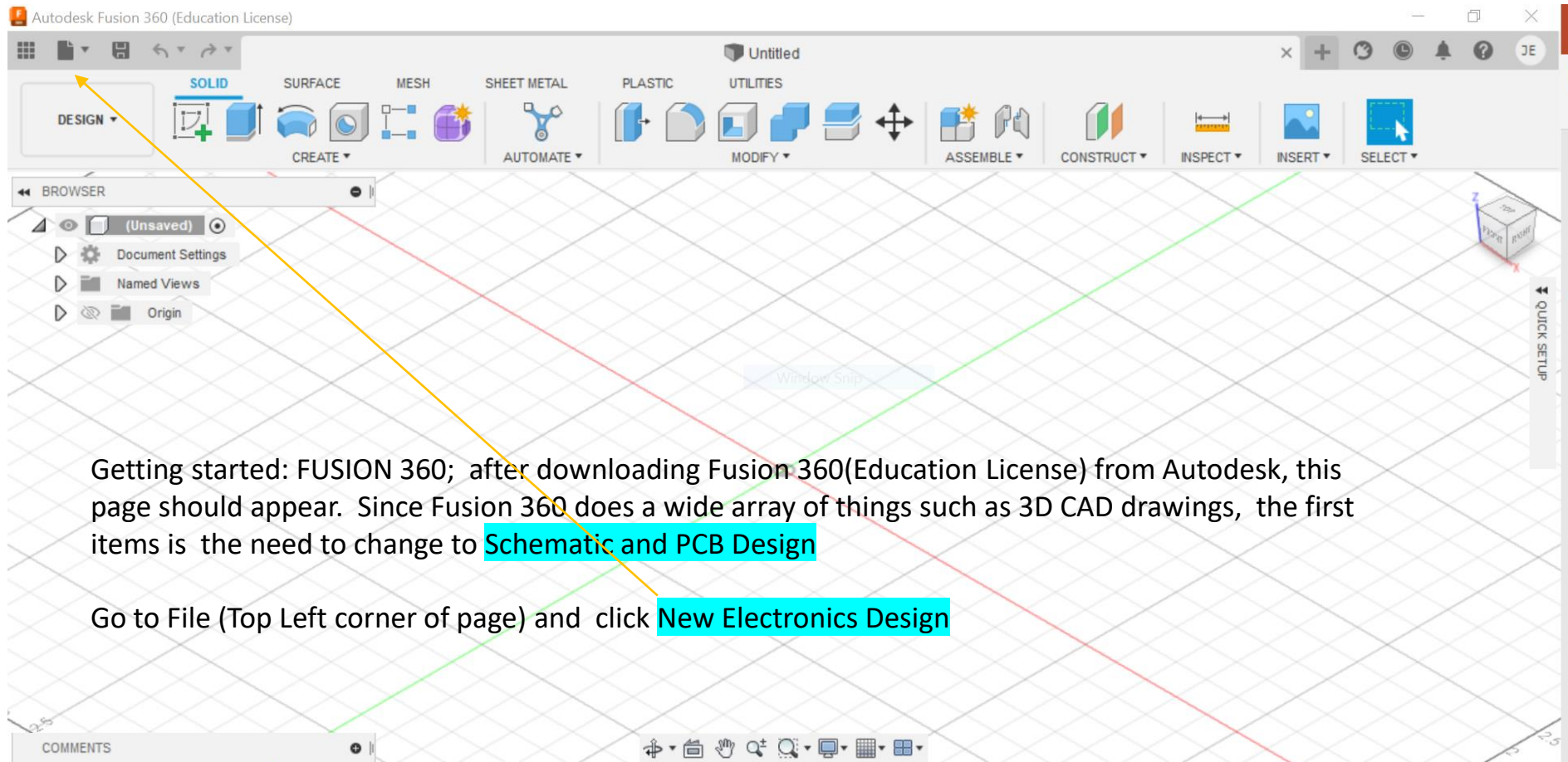
1) Schematic- After Breadboarding is finished and with the benefit of Circuit Lab schematic Create a schematic diagram

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- labeled with your SD project #

https://www.youtube.com/watch?v=jgUZeBiusw&list=PLmA_xUT-8UIL80Xm8Gxz98YNum3I9Glnr by George Garcia



This page opens up to start a schematic

The screenshot shows the Autodesk Fusion 360 (Education License) interface. The top bar includes the application name and window tabs for 'Untitled'. Below the top bar is the 'COMMON' toolbar with icons for creating new files, opening recent files, and saving. A 'CREATE' dropdown menu is visible, with '(Unsaved)' selected. The main workspace is currently empty, with a 'Rectangular Snip' tool highlighted. On the right side, a 'QUICK SETUP' dialog box is open, showing settings for 'Default Units' (set to 'in'), 'CAD Experience' (set to 'New to CAD'), and navigation controls for 'Pan', 'Zoom', and 'Orbit'. A 'START HERE' button is prominently displayed at the bottom of the dialog. Three blue arrows point from text annotations to specific elements: one from 'Work in millimeters or inches?' to the 'Default Units' dropdown, one from 'Make sure to use a mouse' to the mouse icons in the navigation section, and one from 'Take the time to watch the tutorials' to the 'START HERE' button. A fourth blue arrow points from 'To create a schematic' to the 'CREATE' dropdown menu. The bottom of the interface shows a 'Description' field and an 'Edit' button.

Work in millimeters or inches?

Make sure to use a mouse

Take the time to watch the tutorials

To create a schematic

QUICK SETUP

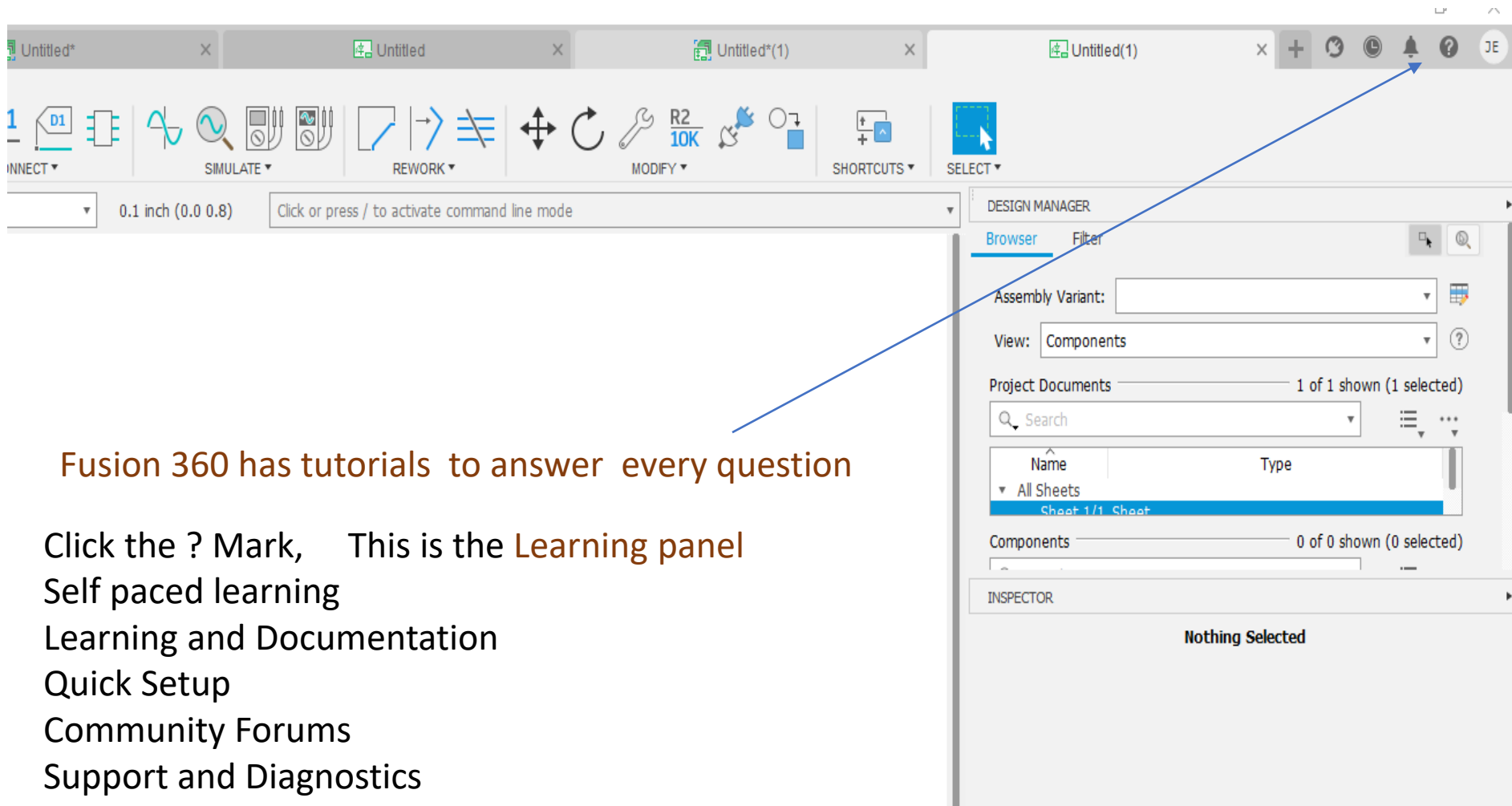
1 SETUP
Default Units: in
New documents will use in as their default unit.

2 NAVIGATE & VIEW
Maneuver your model using familiar controls.
CAD Experience: New to CAD

Pan: Mouse
Zoom: Mouse
Orbit: SHIFT + Mouse

3 LEARN BY DOING
Start mastering Fusion 360 with guided tutorials.
START HERE

Close



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-8UUL80Xm8Gxz98YNum3I9Glnr
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"/>

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.

...

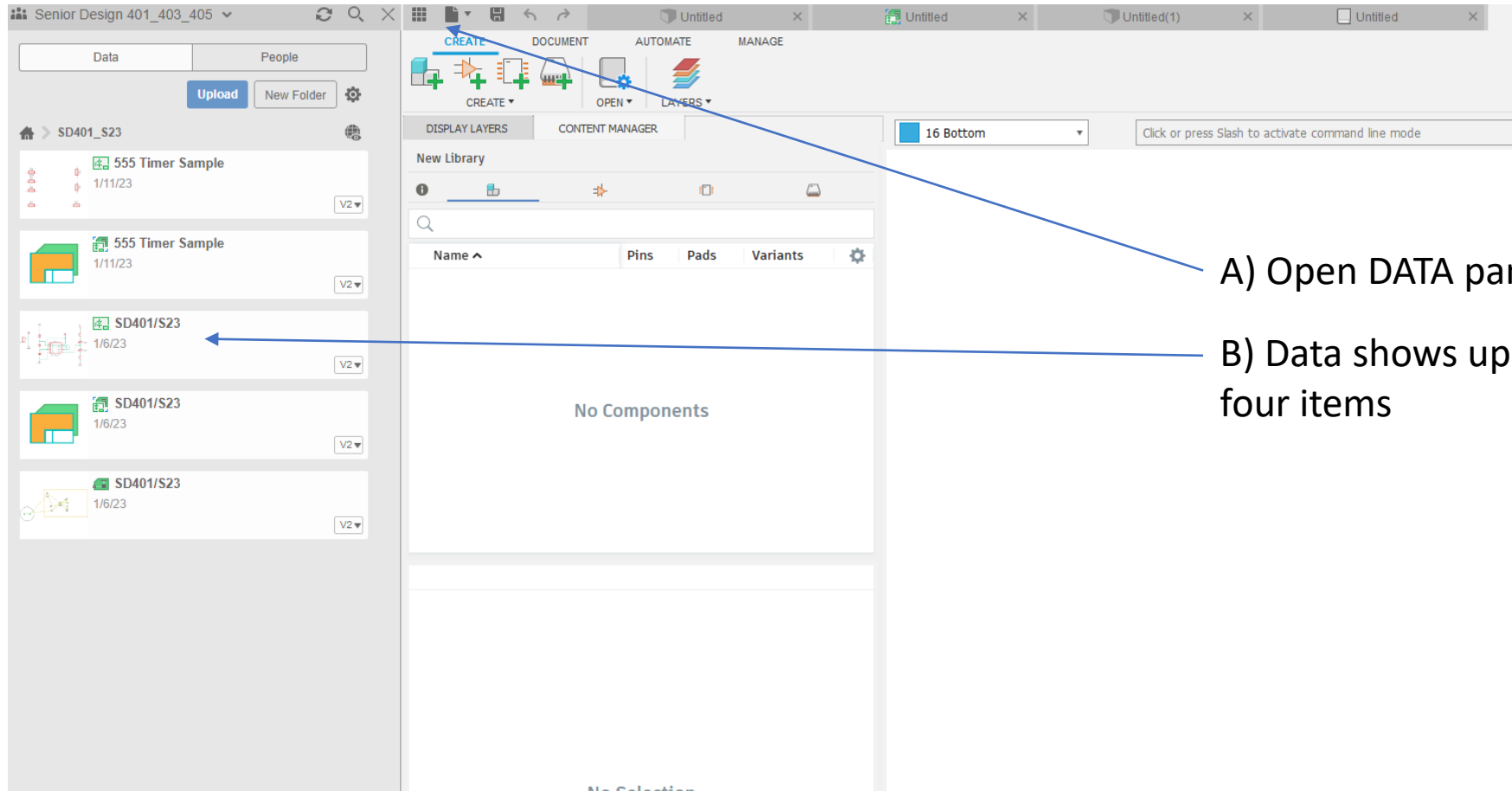
To create a new, custom-defined library:

Go to the Manufacture workspace.

Select Manage -> Tool Library.

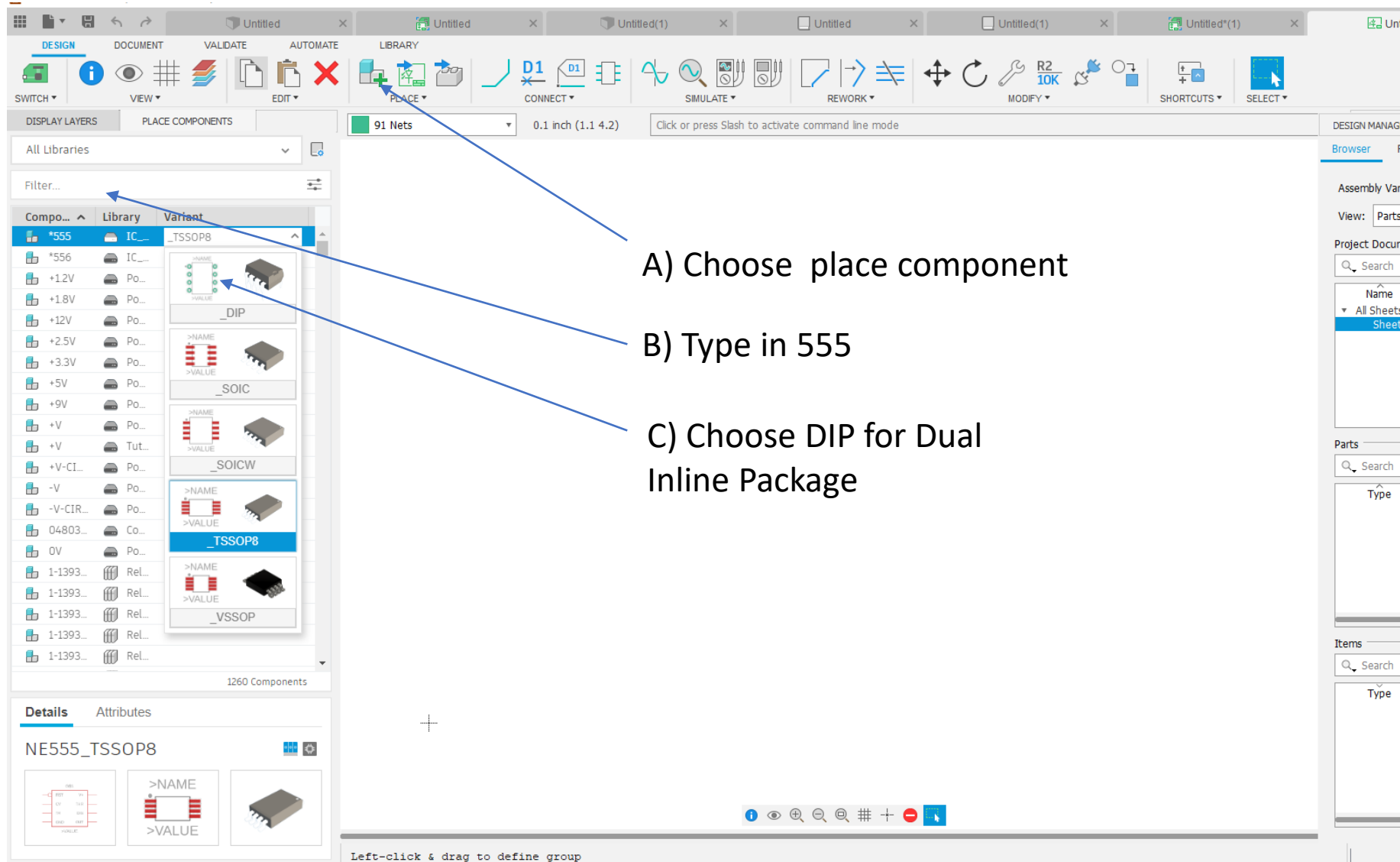
Right-Click on the Local folder.

Select New Tool Library.



A) Open DATA panel

B) Data shows up showing four items



The screenshot shows a PCB design software interface with a component library on the left and a schematic diagram in the center. The library lists various components, with the *555 IC in a DIP package selected. The schematic shows a 555 timer (ICM7555_DIP) with pins 1, 2, 4, 5, 6, 7, 8, and 3 labeled. Annotations A and B provide instructions on how to interact with the library and the schematic.

A) Will change to DIP package

B) Select drag and Drop on screen

Type	Name	Value
Part	IC1	ICM7

The screenshot shows a PCB design software interface with a component library on the left and a circuit diagram in the center. The library lists various components, with **ICM7555_DIP** selected. The circuit diagram shows an 8-pin component labeled **ICM7555_DIP** with pins 1 through 8 connected to various terminals. A blue arrow points from the text "A) Add resistors in your circuit- Type Resistors" to the **ICM7555_DIP** component in the library.

Component Library Table:

Compo...	Library	Variant
*555	IC...	_DIP
*556	IC...	_SOIC
+12V	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...	

Circuit Diagram Pinout:

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

Component Details:

ICM7555_DIP

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

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

ICM7555_DIP

Left-click & drag to define group

Never Delete these- always need three

The screenshot shows the Altium Designer software interface. The top toolbar includes buttons for DESIGN, DOCUMENT, VALIDATE, AUTOMATE, LIBRARY, PLACE, CONNECT, SIMULATE, REWORK, MODIFY, SHORTCUTS, and SELECT. The left sidebar shows a component library with a search filter and a list of components. The main workspace displays a circuit board layout with an IC1 component (ICM7555_DIP) and six resistors (R1-R6). Annotations with blue arrows point to specific features: 'A) Select and insert Resistors wherever required' points to the component library; 'B) Double right click to rotate components' points to resistor R3; 'Move Command' points to the 'MOVE' button in the toolbar; and 'Change Value Command' points to the 'R2 10K' button in the toolbar. The right sidebar shows the DESIGN MANAGER with a browser and parts list.

A) Select and insert Resistors wherever required

B) Double right click to rotate components

Move Command

Change Value Command

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

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	

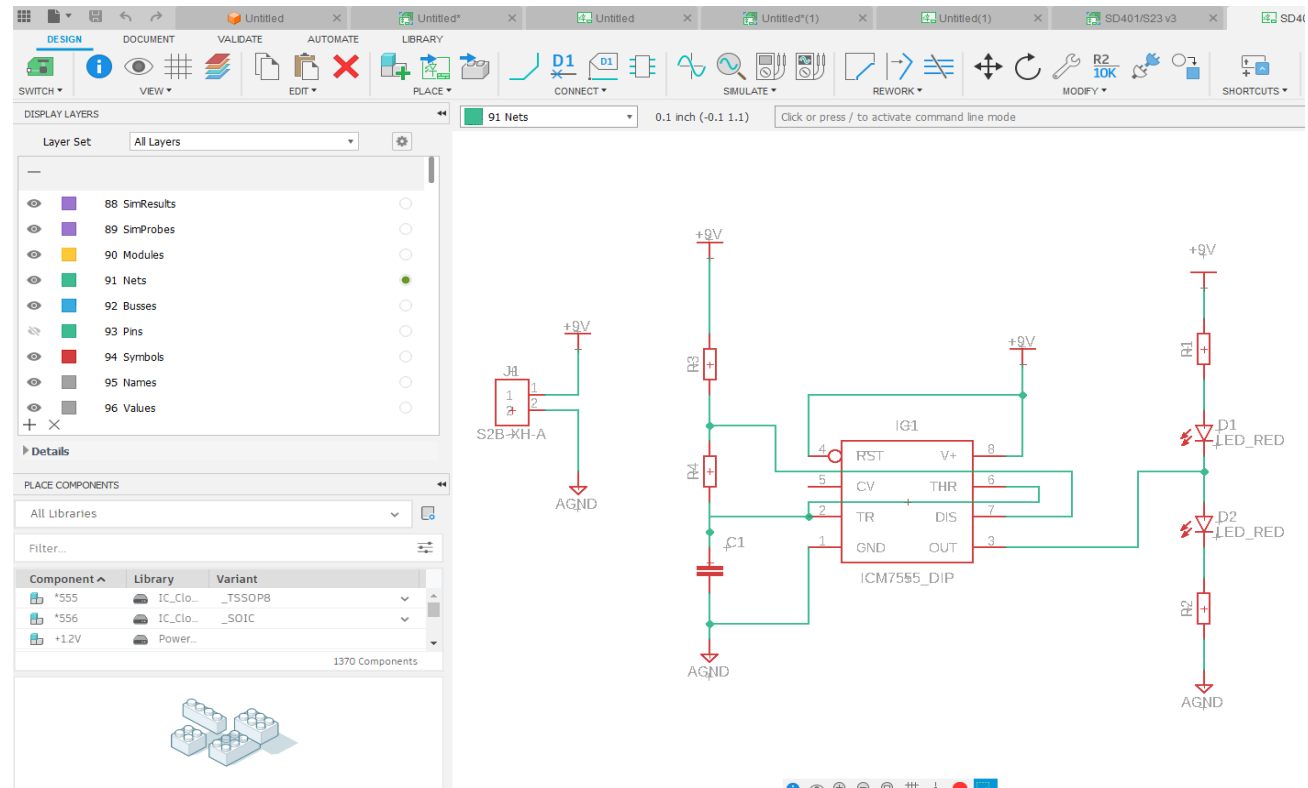
Items 0 of 0 shown (0 selected)

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

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

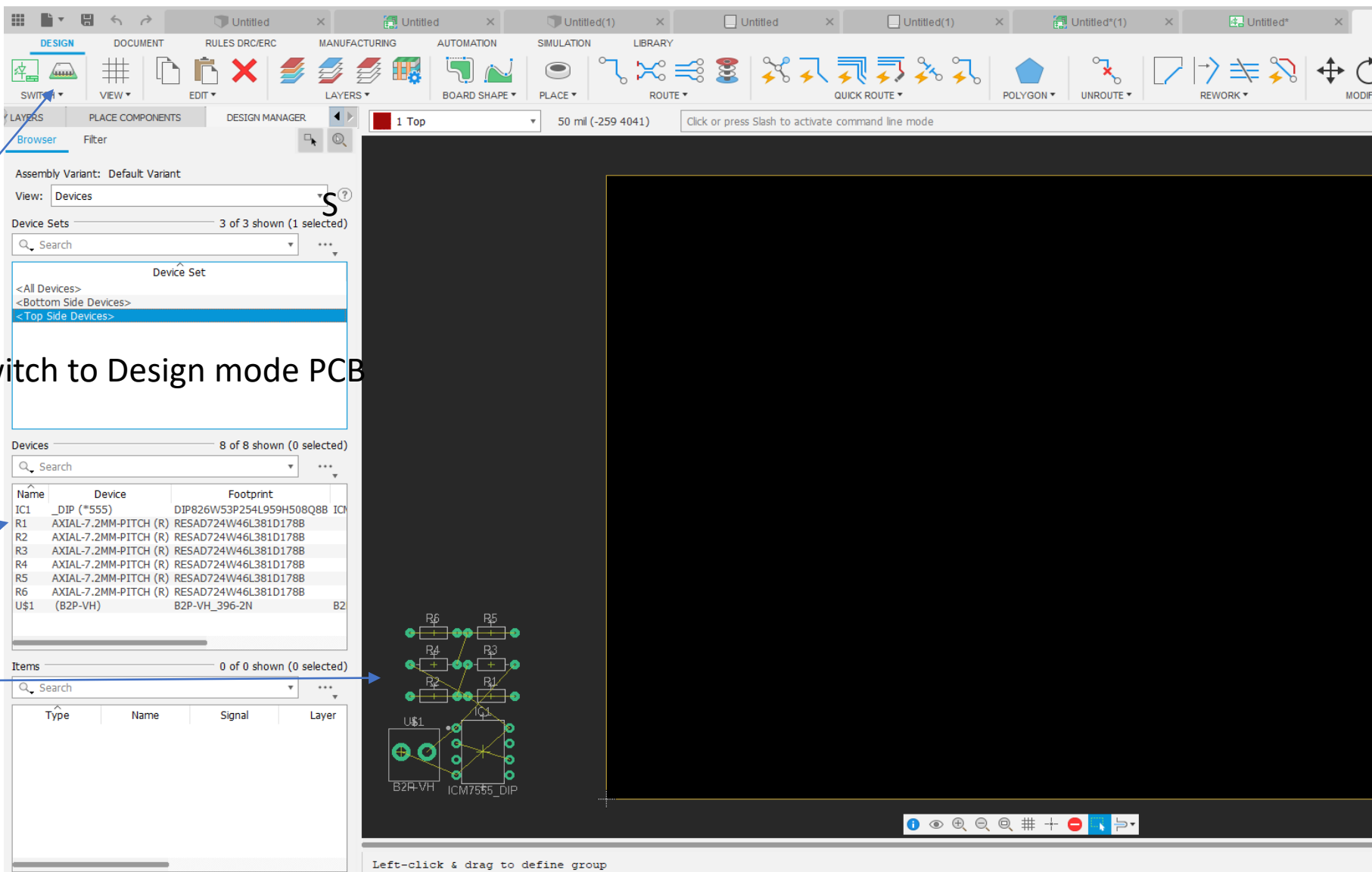
If your curious what is next, on the next slide look at Design >Switch> Schematic to PCB

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



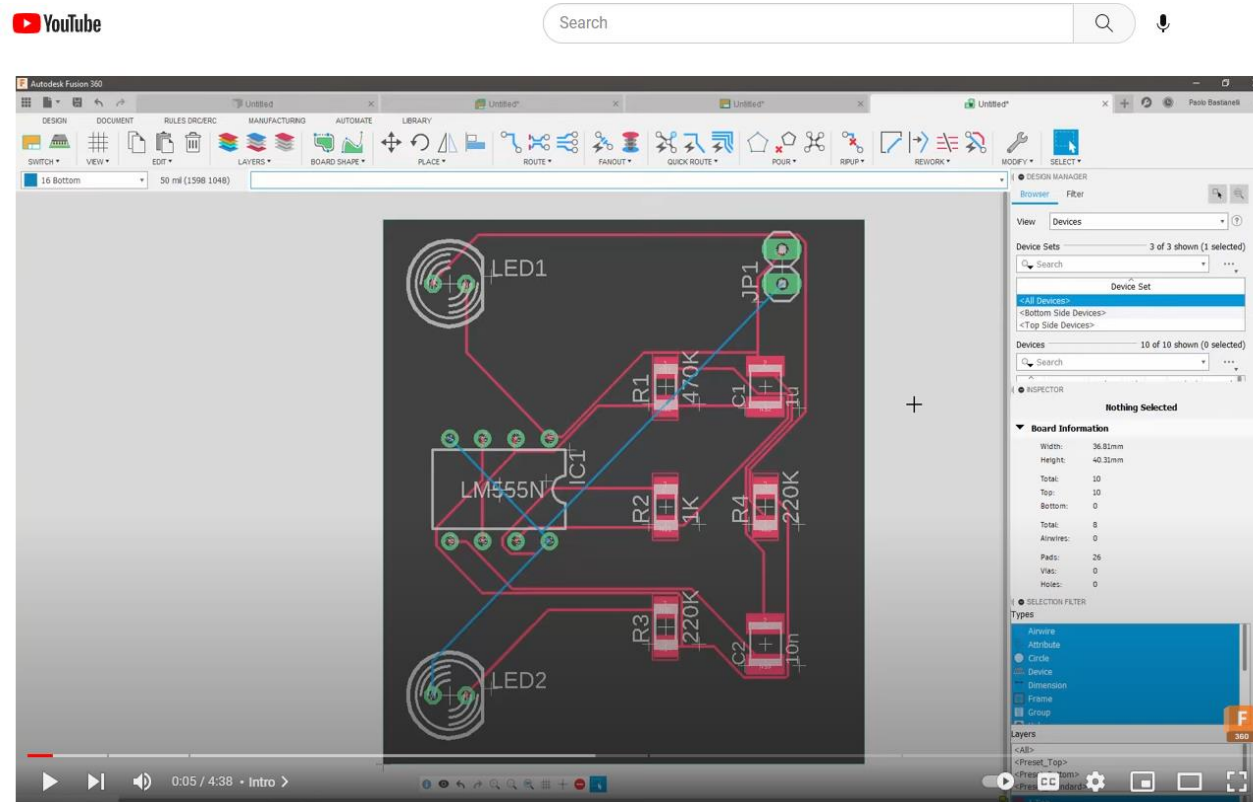
Switch to Design mode PCB

List of components in Design

2D Sim of components in design

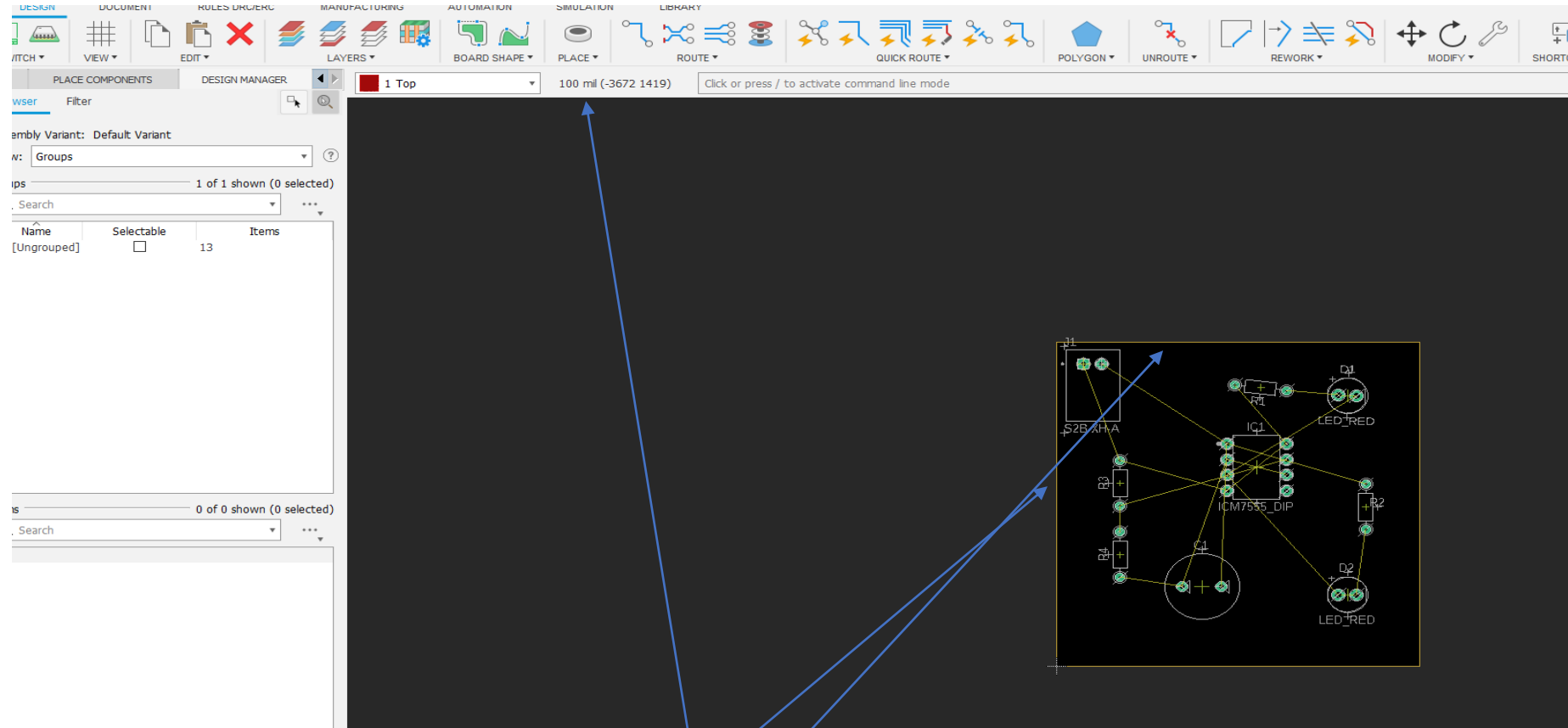
Part 2 Creating Circuit Board Layout

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



[PCB Layout Tutorial Walkthrough – YouTube](https://www.youtube.com/watch?v=VZZBEocoYDA) 0:04/4:38

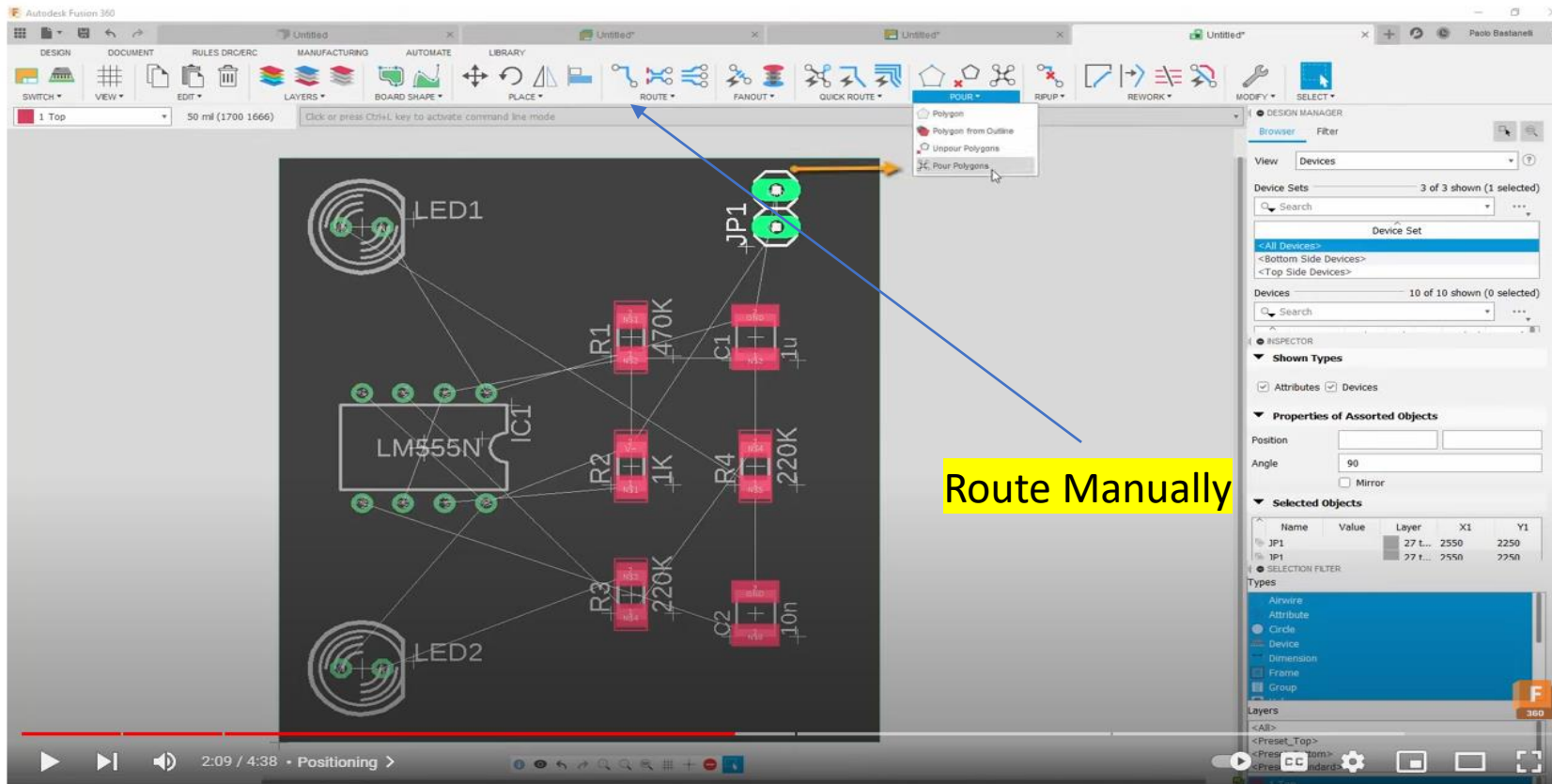
2D sim Components have been dragged into the Black square this is your PCB out line



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

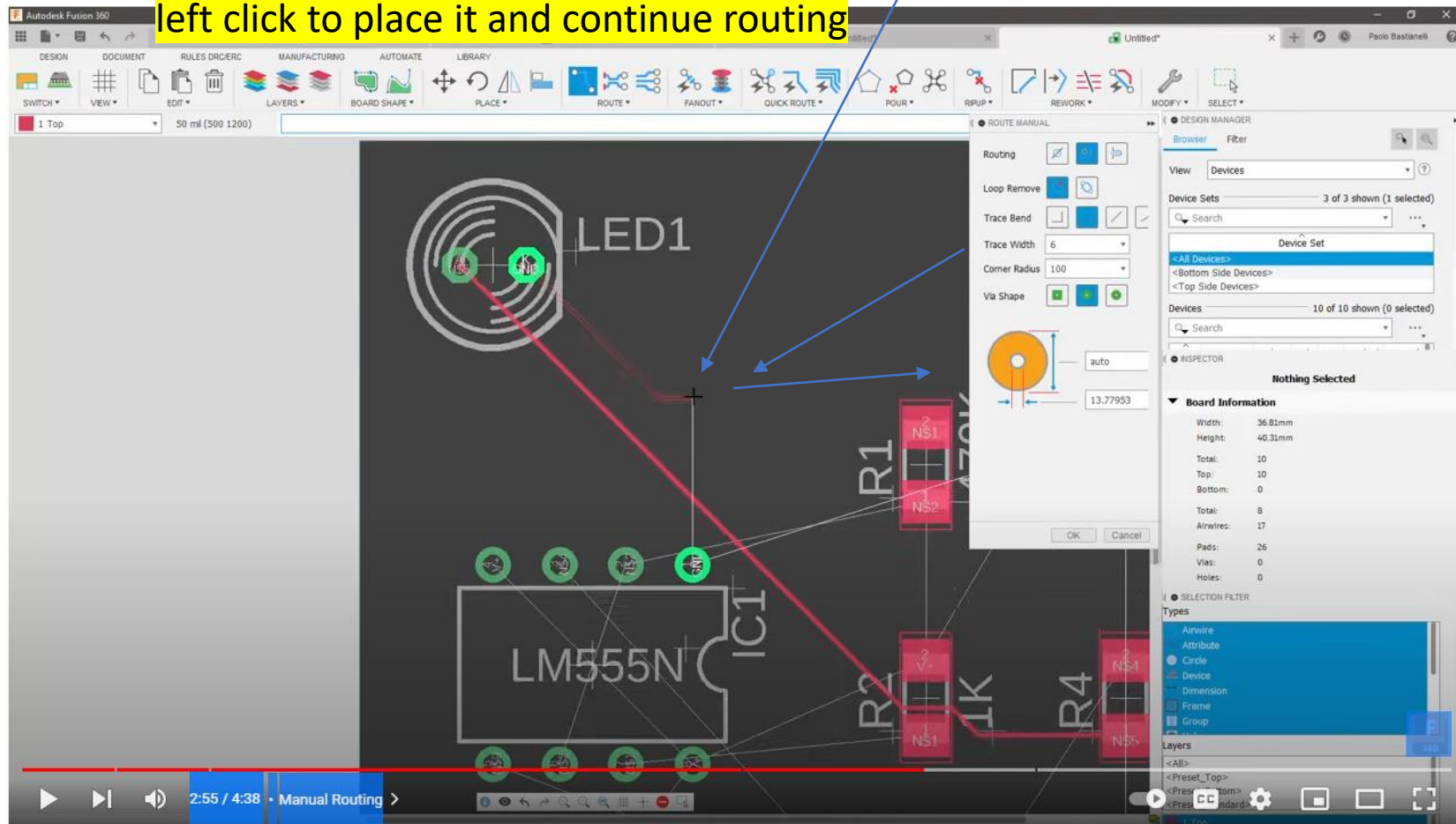
[PCB Layout Tutorial Walkthrough – YouTube](#) 0:42/4:38

To route traces manually and then automatically



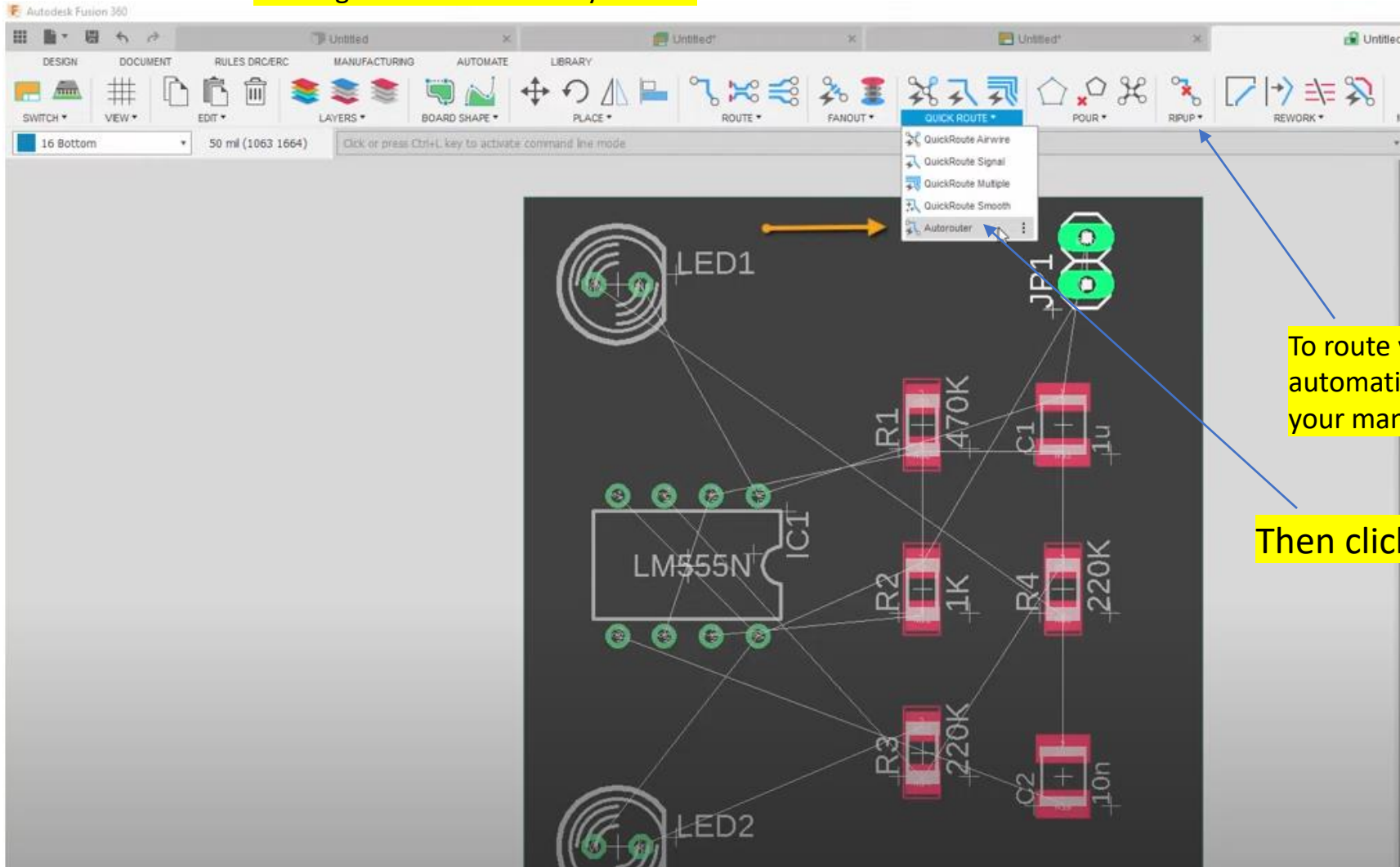
[PCB Layout Tutorial Walkthrough – YouTube](#) 2:32 mark

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



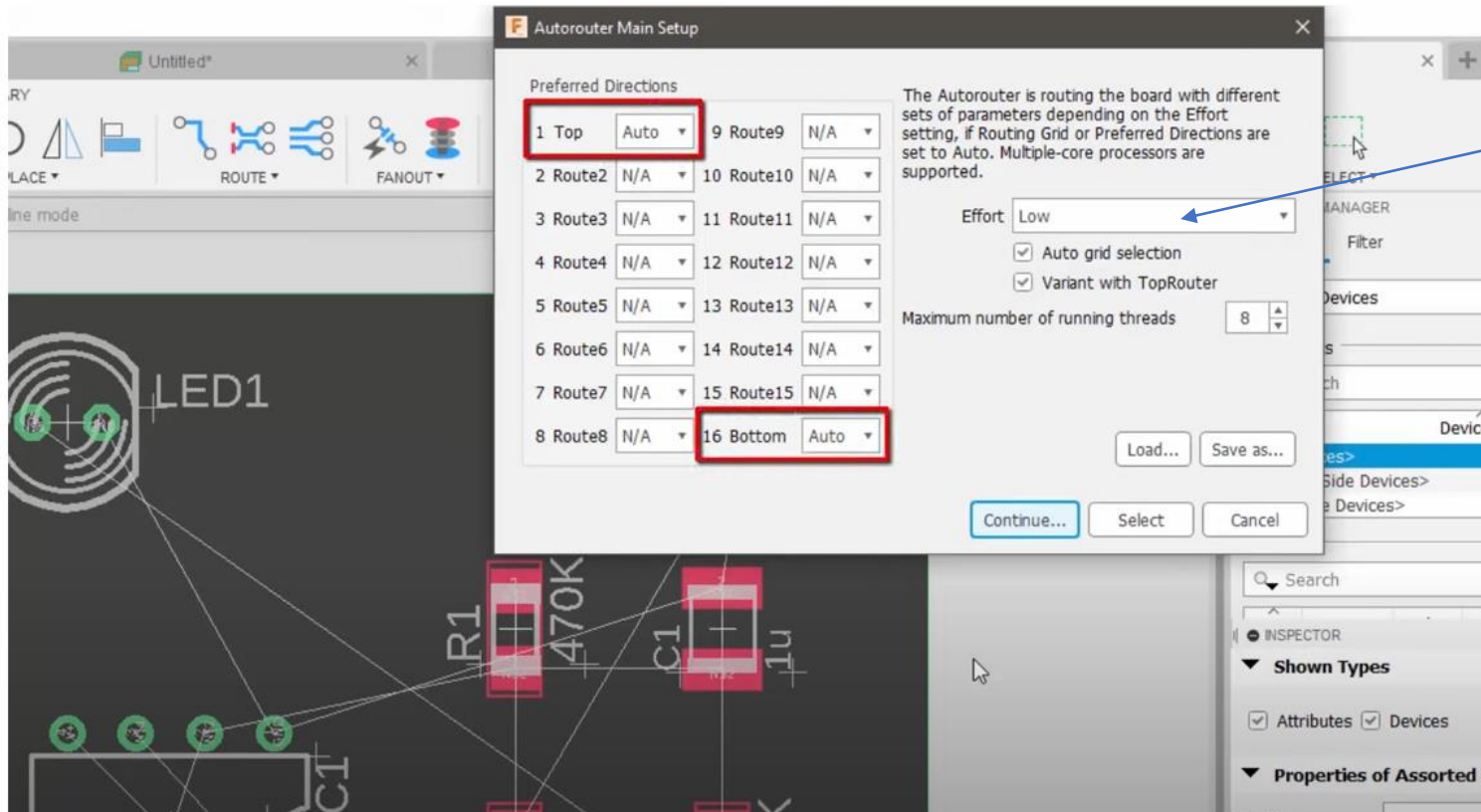
<https://www.youtube.com/watch?v=VZZBEocoYDA> 2:55/4:38

Routing the traces manually vs auto



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

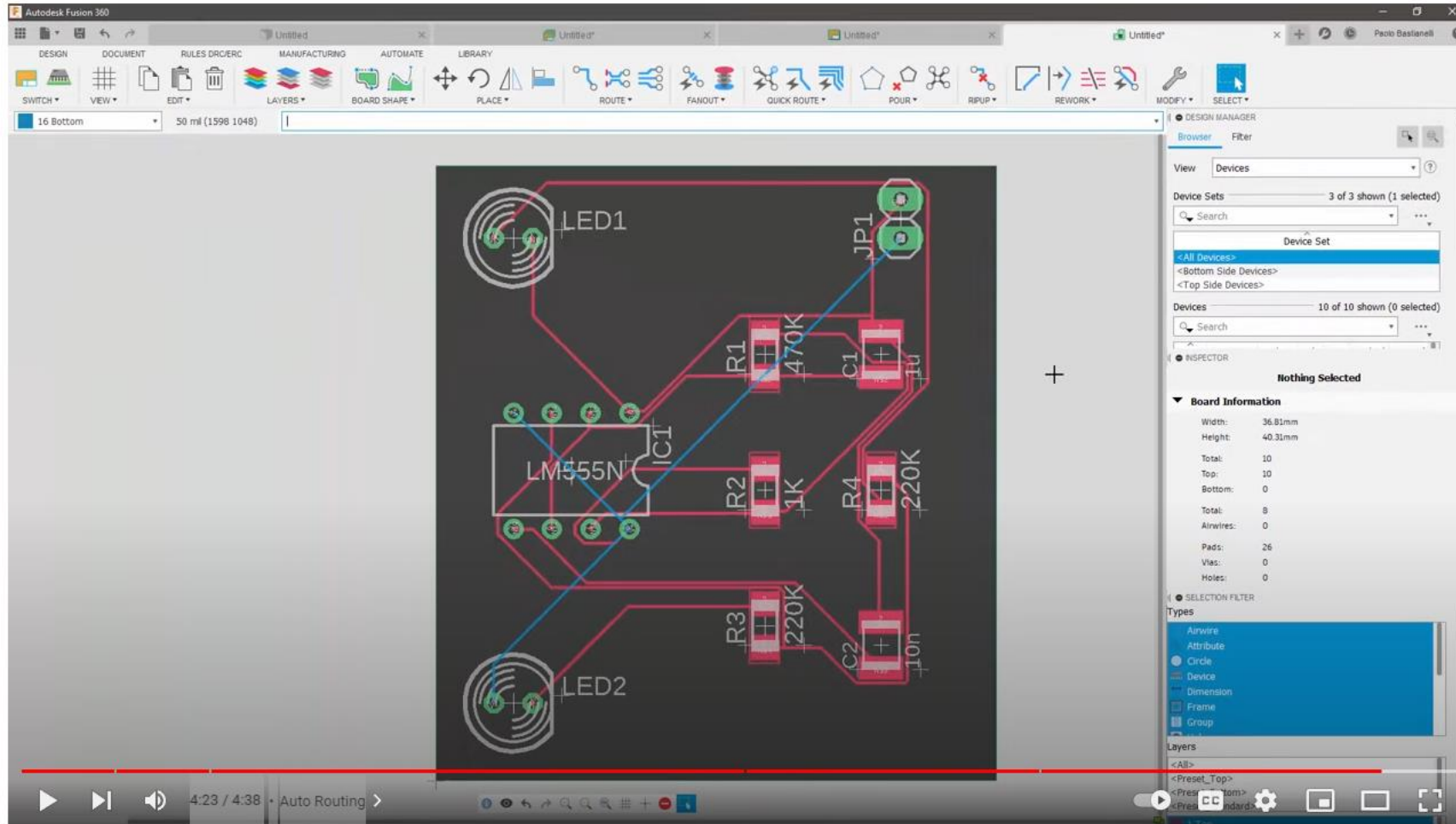
Two layer boards only top and bottom will be selected to auto route



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 of vias, the PCB is complete



PCB Manufacturing- Creating Gerber Files

The screenshot displays the Altium Designer interface with the **MANUFACTURING** tab selected. The top menu bar includes DESIGN, DOCUMENT, RULES/DRC/ERC, MANUFACTURING, AUTOMATION, SIMULATION, and LIBRARY. The MANUFACTURING toolbar contains icons for switching, viewing, manufacturing, and outputs. The left sidebar shows the **DESIGN MANAGER** panel with the following details:

- Assembly Variant: Default Variant
- View: Groups
- Groups: 1 of 1 shown (0 selected)
- Search: [Empty]
- Table:

Name	Selectable	Items
[Ungrouped]	<input type="checkbox"/>	79

The main workspace shows a PCB layout with the following components and connections:

- IC1: ICM7555_DIP
- IC2: S2B-XH-A
- LED1: LED_TRED
- LED2: LED_TRED
- LED3: LED_TRED
- LED4: LED_TRED
- Resistors: R1, R2, R3, R4
- Capacitor: C1

The layout is shown on a dark background with a yellow border around the components.

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

The screenshot displays the Fusion 360 software interface with the 'MANUFACTURING' tab selected. The 'CAM Export File List' dialog box is open, showing a list of manufacturing files to be included in a ZIP archive. The list includes Gerber files for copper, soldermask, solderpaste, and silkscreen, as well as drill files and assembly files. The dialog also includes a note about the template used for generation and buttons for 'Update Active CAM Job', 'OK', and 'Cancel'. A blue arrow points from the text on the left to the 'OK' button in the dialog.

DOCUMENT RULES DRC/ERC MANUFACTURING AUTOMATION SIMULATION LIBRARY

VIEW MANUFACTURING OUTPUTS

FACE COMPONENTS DESIGN MANAGER 1 Top 100 mil (-2995 4184) Click or press / to activate command line mode

Filter

Variant: Default Variant

1 of 1 shown (0 selected)

Selectable	Items
<input type="checkbox"/>	79

0 of 0 shown (0 selected)

CAM Export File List

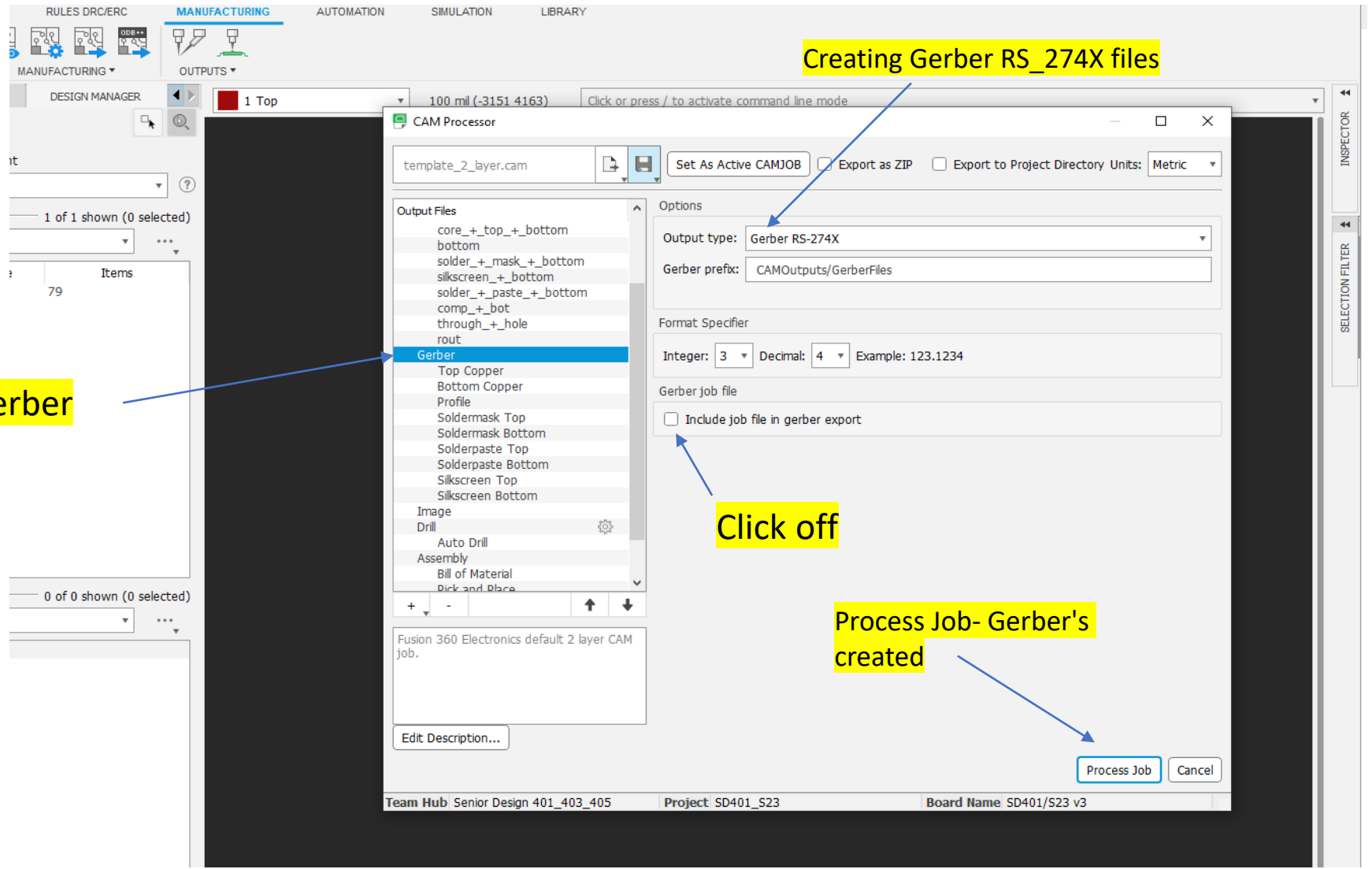
One-Click Manufacturing Export

Fusion 360 will create a ZIP archive containing the following manufacturing files:

- Senior Design 401_403_405/SD401_S23/CAMOutputs/GerberFiles/gerber_job.gbrjob
- Senior Design 401_403_405/SD401_S23/CAMOutputs/GerberFiles/copper_top.gbr
- Senior Design 401_403_405/SD401_S23/CAMOutputs/GerberFiles/copper_bottom.gbr
- Senior Design 401_403_405/SD401_S23/CAMOutputs/GerberFiles/profile.gbr
- Senior Design 401_403_405/SD401_S23/CAMOutputs/GerberFiles/soldermask_top.gbr
- Senior Design 401_403_405/SD401_S23/CAMOutputs/GerberFiles/soldermask_bottom.gbr
- Senior Design 401_403_405/SD401_S23/CAMOutputs/GerberFiles/solderpaste_top.gbr
- Senior Design 401_403_405/SD401_S23/CAMOutputs/GerberFiles/solderpaste_bottom.gbr
- Senior Design 401_403_405/SD401_S23/CAMOutputs/GerberFiles/silkscreen_top.gbr
- Senior Design 401_403_405/SD401_S23/CAMOutputs/GerberFiles/silkscreen_bottom.gbr
- Senior Design 401_403_405/SD401_S23/CAMOutputs/DrillFiles/drill_1_16.xln
- Senior Design 401_403_405/SD401_S23/CAMOutputs/Assembly/SD401/S23 v3.txt
- Senior Design 401_403_405/SD401_S23/CAMOutputs/Assembly/PnP_SD401/S23 v3_front.txt
- Senior Design 401_403_405/SD401_S23/CAMOutputs/Assembly/PnP_SD401/S23 v3_back.txt

These outputs will be generated based on template_2_layer.cam.

Update Active CAM Job OK Cancel

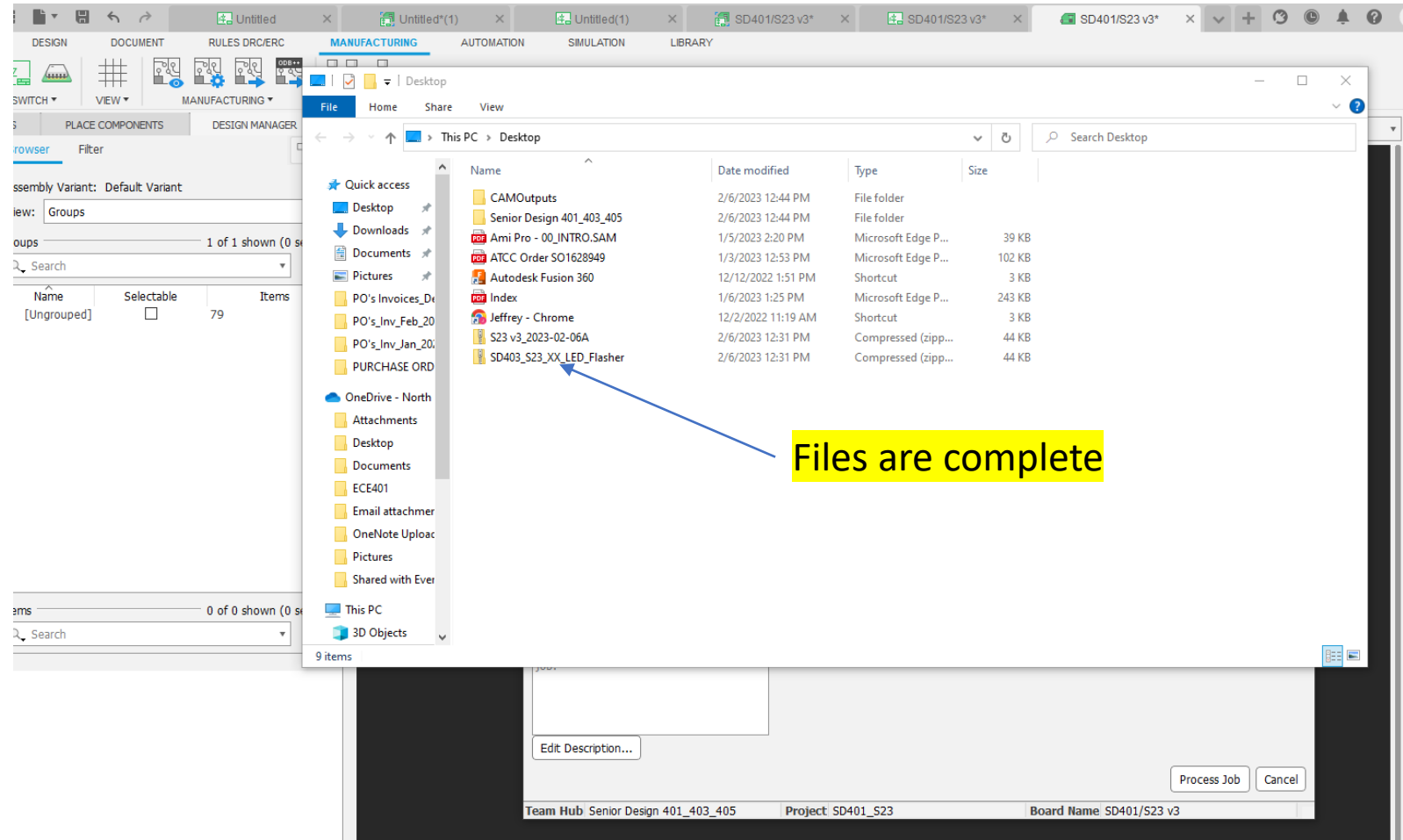


Creating Gerber RS_274X files

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

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

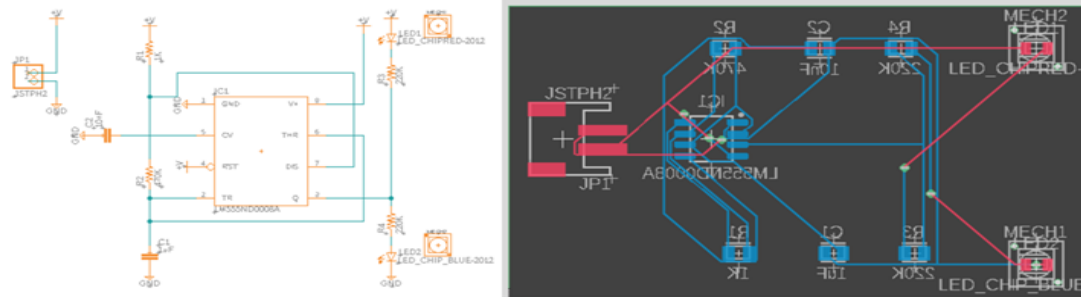
Electronics / Tutorials / Tutorial: Create a PCB layout

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 engineers, 100 different 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

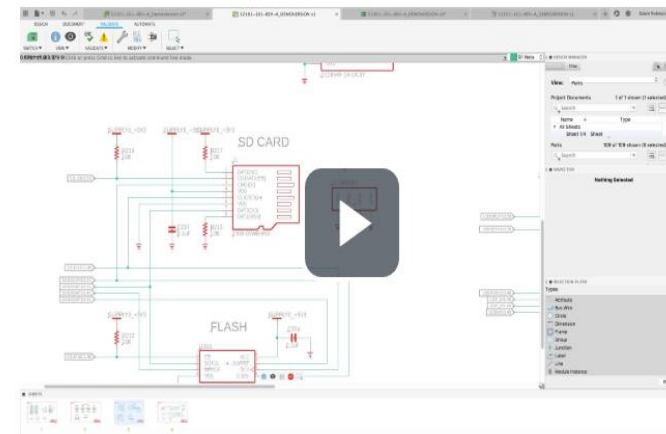
Electronics / [Computer-aided manufacturing \(CAM\) support](#)

Computer-aided manufacturing (CAM) support

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

Video: Overview of producing manufacturing data for a printed circuit board using the CAM processor in Fusion 360

Length: 0:47



Pages in this section

- [Prepare manufacturing data](#)
- [About CAM Preview](#)