KÄRNTEN
University of
Applied Sciences

PCB DESIGN BASICS

METIN OKTAY YILMAZ
SYSTEMS DESIGN/EMBEDDED SYSTEMS

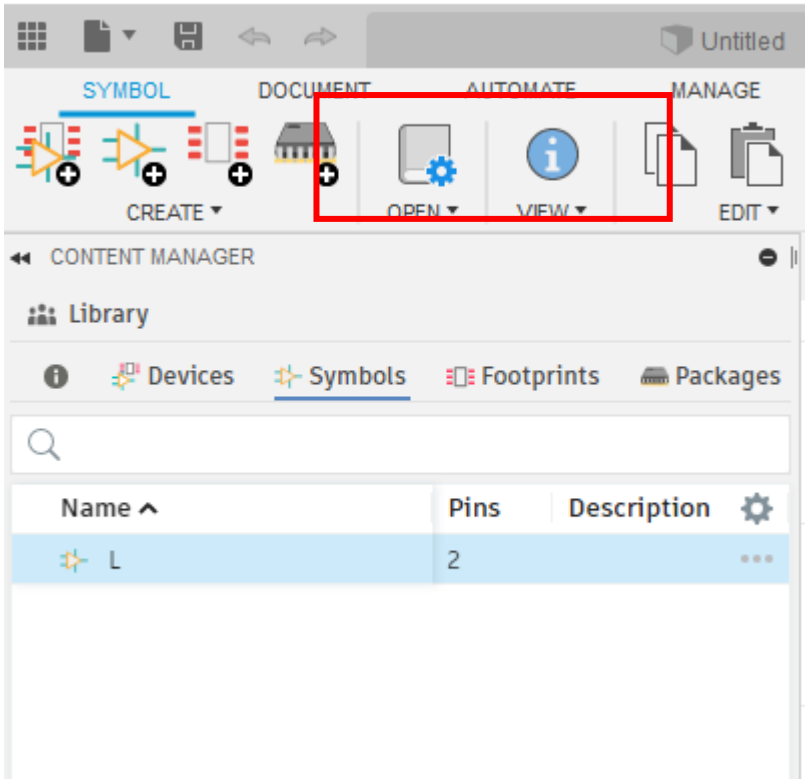
CAD TOOL CHOOSEN

- **Autodesk Fusion360 (EDA tool previously Cadsoft Eagle; now comes embedded in Fusion360)**

Fusion360 tutorial: https://www.autodesk.com/products/fusion-360/blog/january-27-2020-product-update-whats-new/

GENERAL SETUP

- **First**: Symbol creation
- **Then**: Footprint
- **Then:** Component creation, pin connection and writing device description
- **Lastly**: 3D Package (using built-in automatic generator)



Fusion 360 library design GUI

MUST KNOW LAYERS IN FUSION360

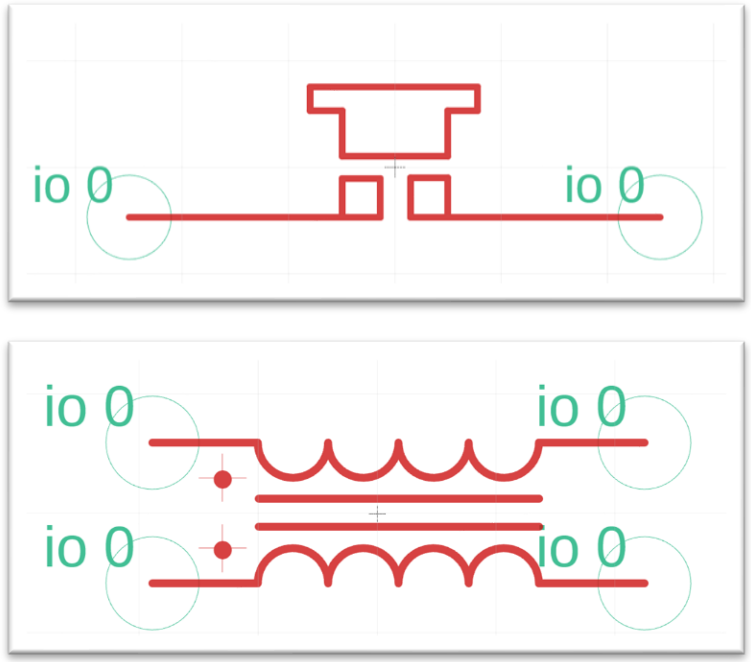
- Layer 1: Top copper surface
- Layer 2-15: Routes
- Layer 16: Bottom copper surface
- Layer 17: Through hole pads
- Layer 18: Vias
- Layer 19: Unrouted connections
- Layer 20: Dimensions of the board
- Layer 21-22: Silkscreen top and bottom called tPlace-bPlace
- Layer 23-24: Origins of the components -> NOT PRINTED ONTO THE BOARD
- Layer 25-26: Names of the components -> PRINTED TO THE BOARD
- Layer 27-28: Values of the components e.g. resistance, capacitance
- Layer 29-30: Solder mask exclusion areas
- Layer 31-32: Solder paste application areas

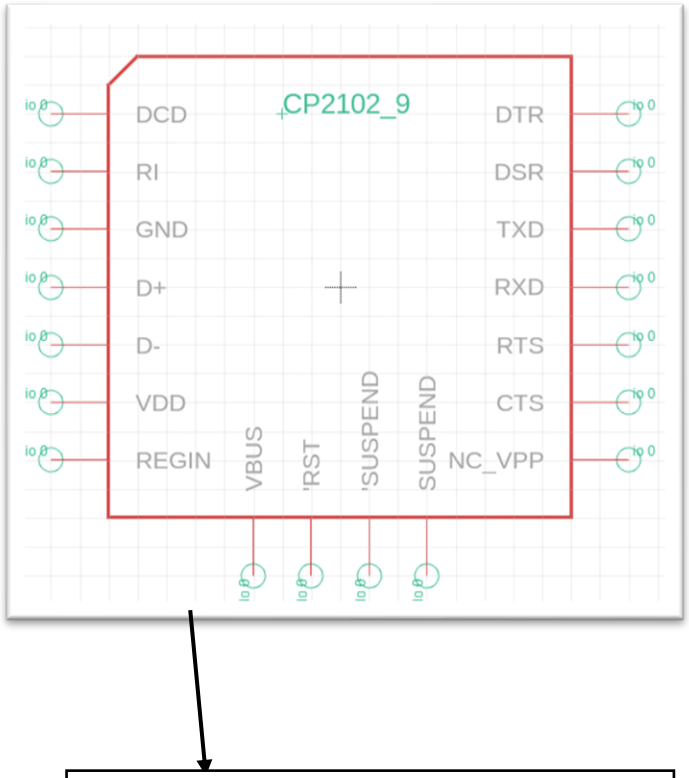
Source:
https://www.autodesk.com/products
/fusion-360/blog
/every-layer-explained-autodesk-eagle/

MY LIBRARY CONSISTS OF:

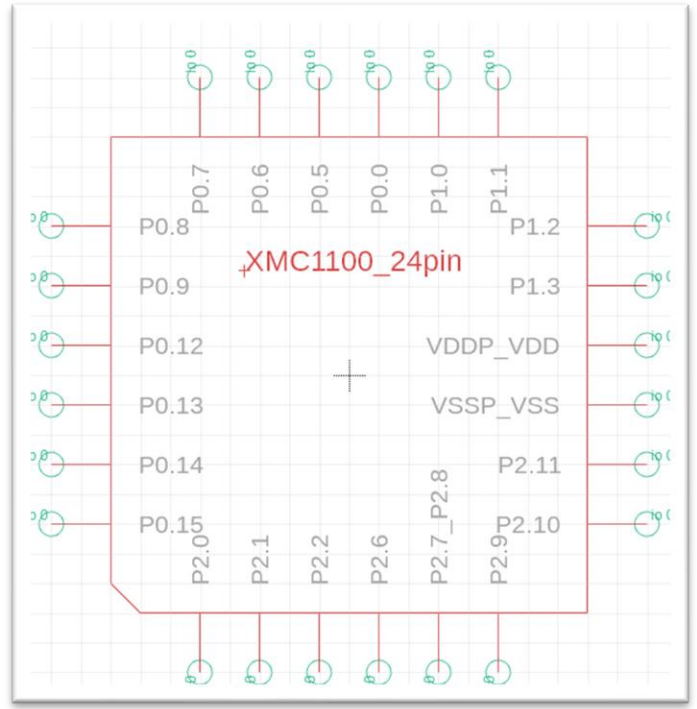
- XMC1100 MCU
- CP2012 USB TO SERIAL INTERFACE
- TPS74A24 LDO REGULATOR
- PUSH BUTTON / LED
- USB MINI B PORT
- WE_CNSV & WE_TSV PROTECTION AND FILTERING COMPONENTS
- RESISTORS / CAPACITORS

THE SYMBOL





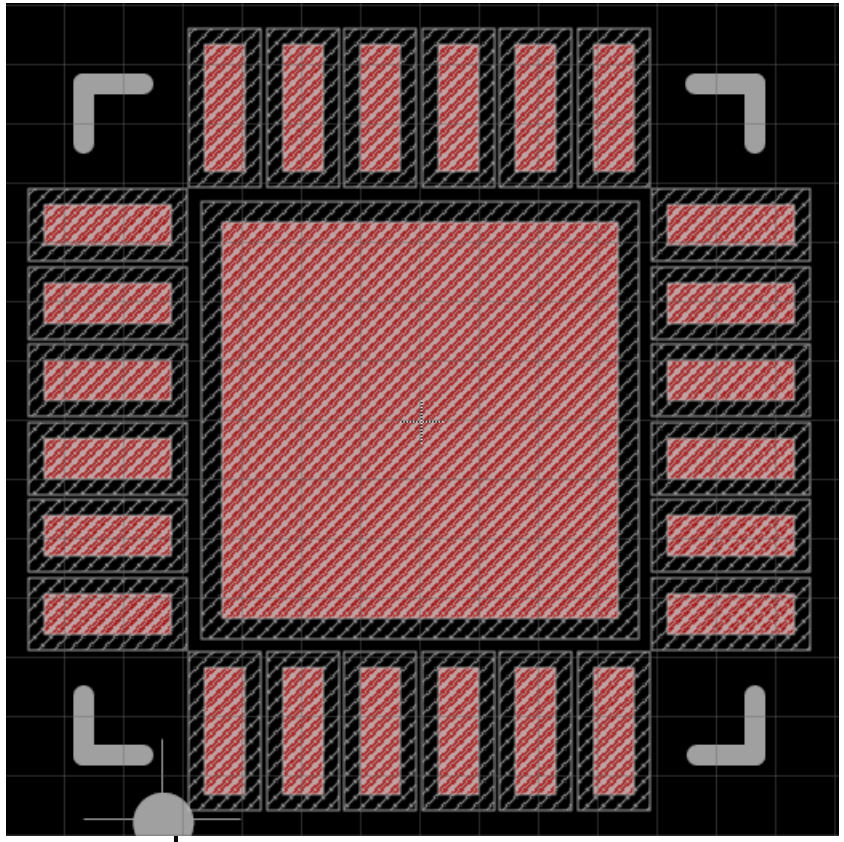
10 NC PINS NOT INCLUDED IN SYMBOL



https://www.youtube.com/watch?v=-LWLXkMUcbk&ab_channel=AutodeskFusion360

THE FOOTPRINT

- Metric
- Center of Gravity: All components follow datasheet guidelines (most of them has 2 lines of symmetry)
- Silkscreen Outline Line Width: 0.381mm (15mil)
- Silkscreen to Land (Pad) Clearance: 0.127mm(5mil)

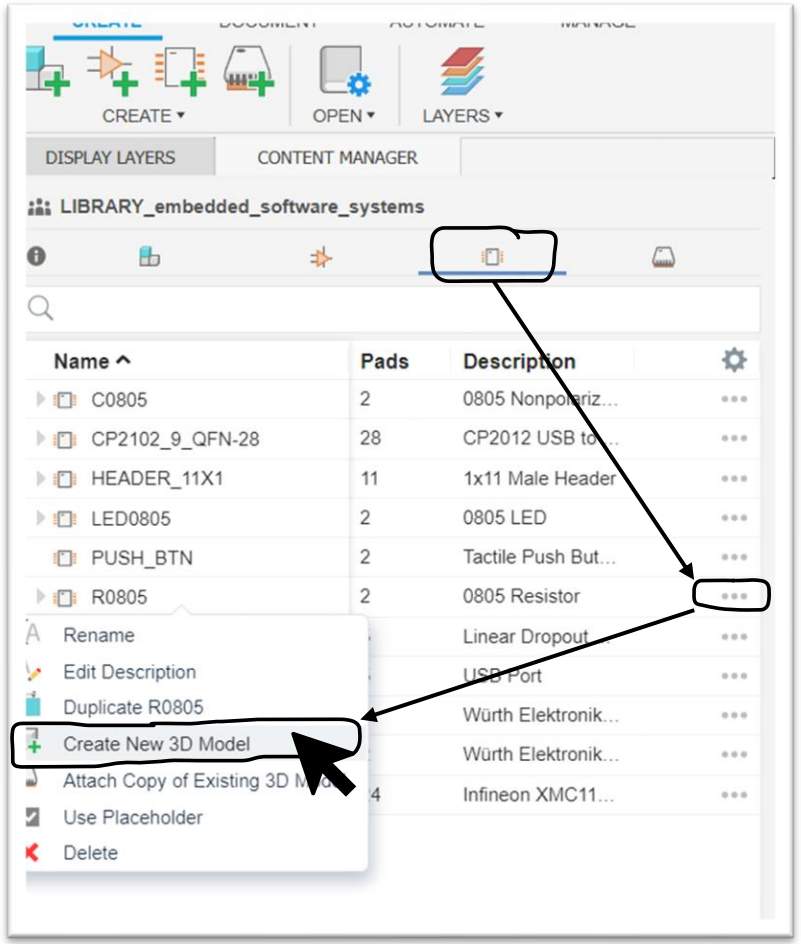


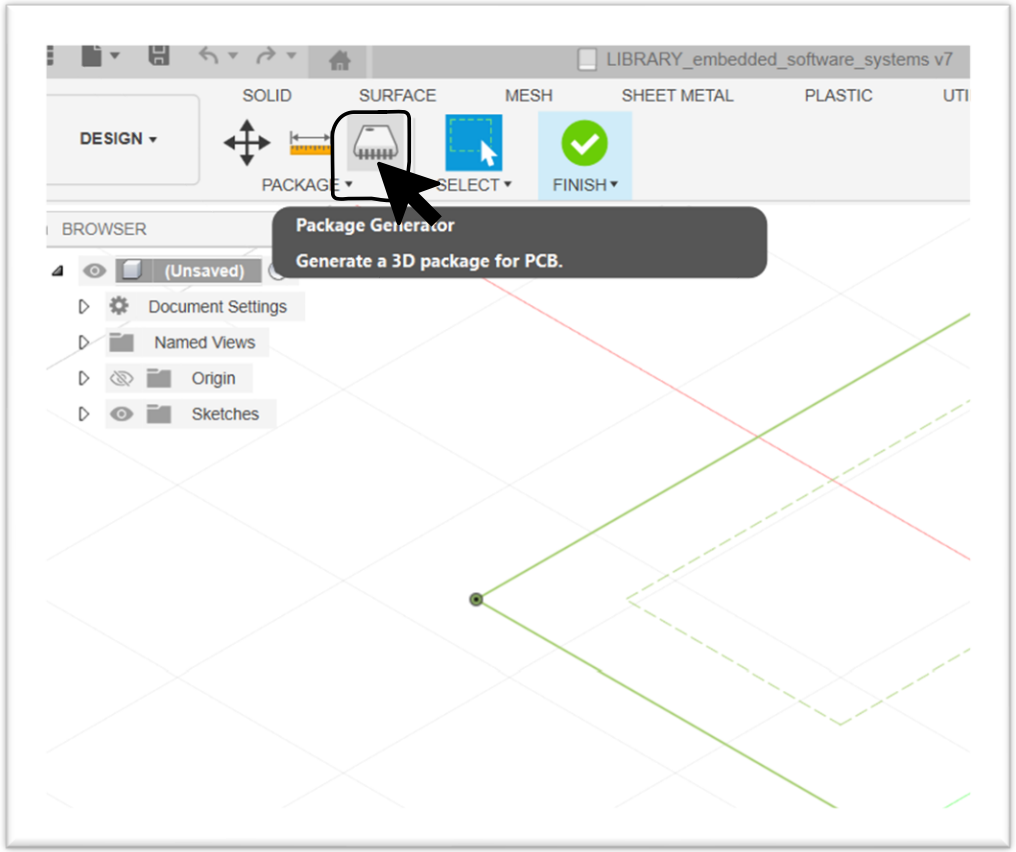
XMC1100 VQFN24 footprint

Pin No.1 Marker

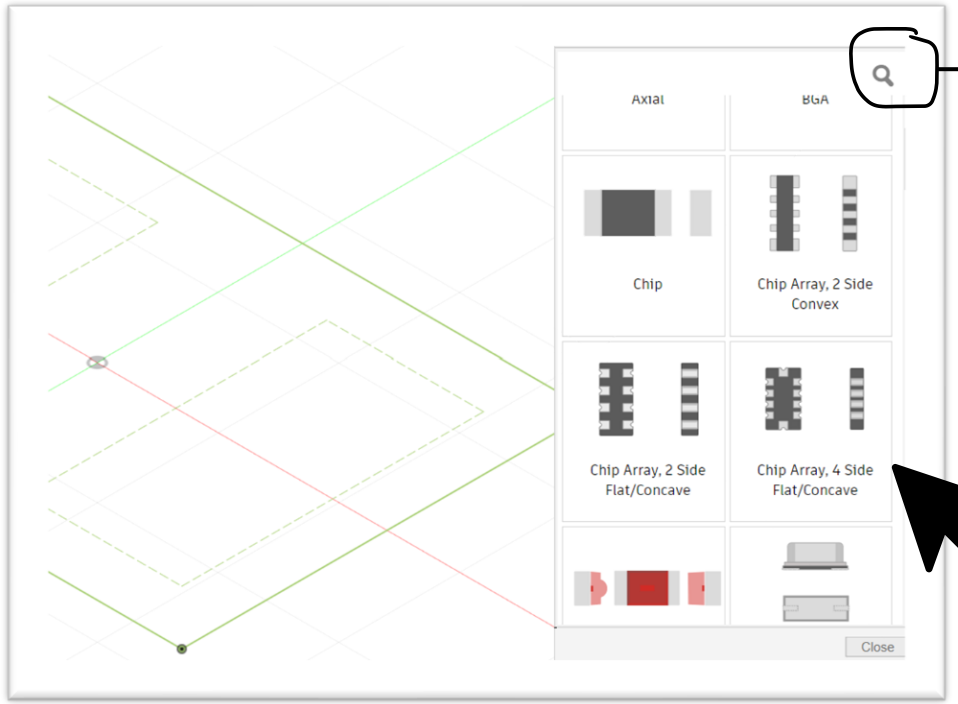
The footprint designers must read:
https://www.pcblibraries.com/downloads/temp/PCB_Design_Optimization_Starts_in_the_CAD_Library_.pdf

3D PACKAGE CREATION





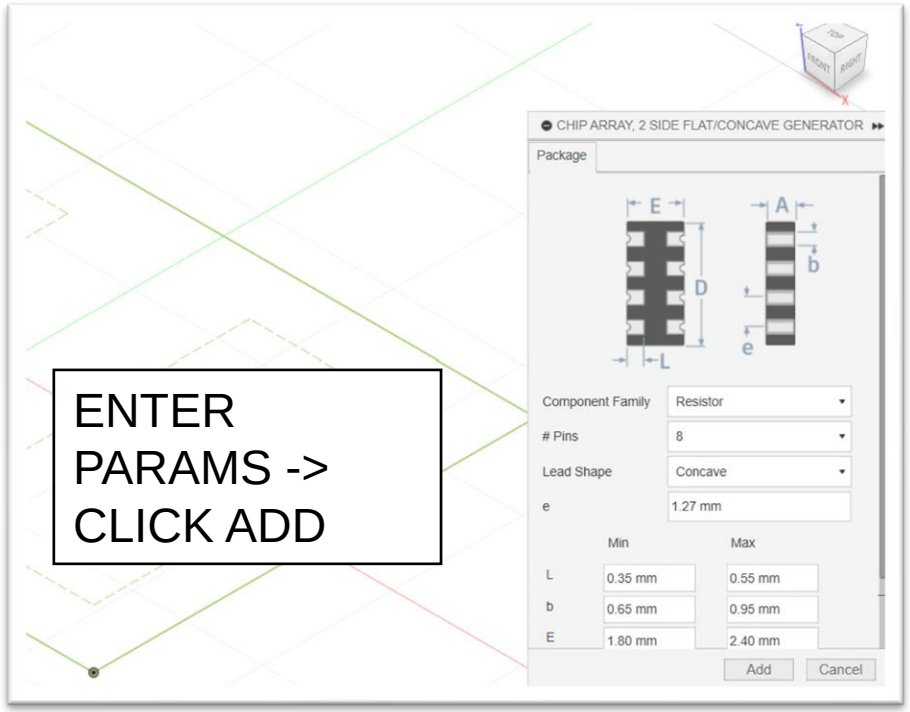
CONTINUED...



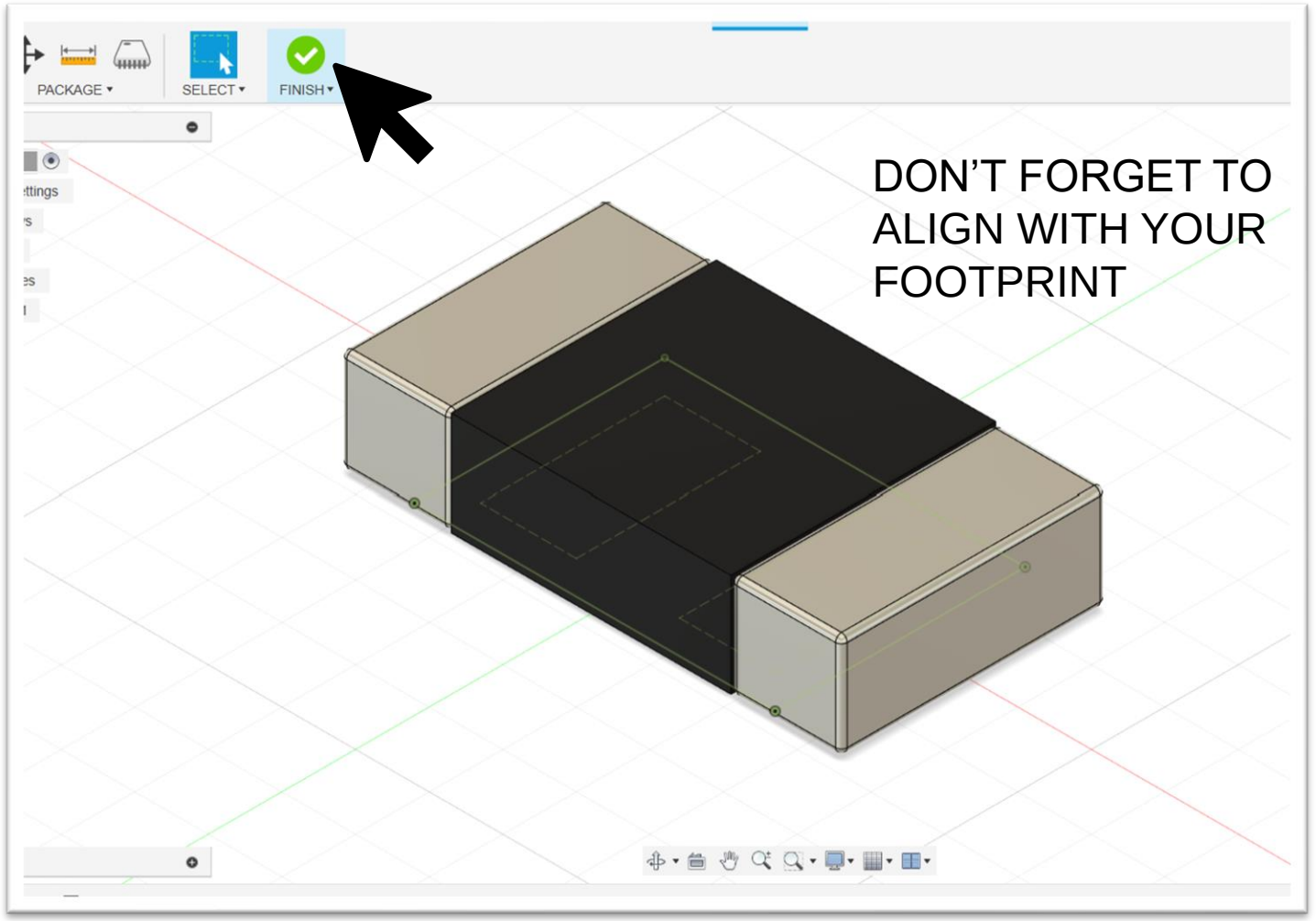
SEARCH PACKAGE

SELECT

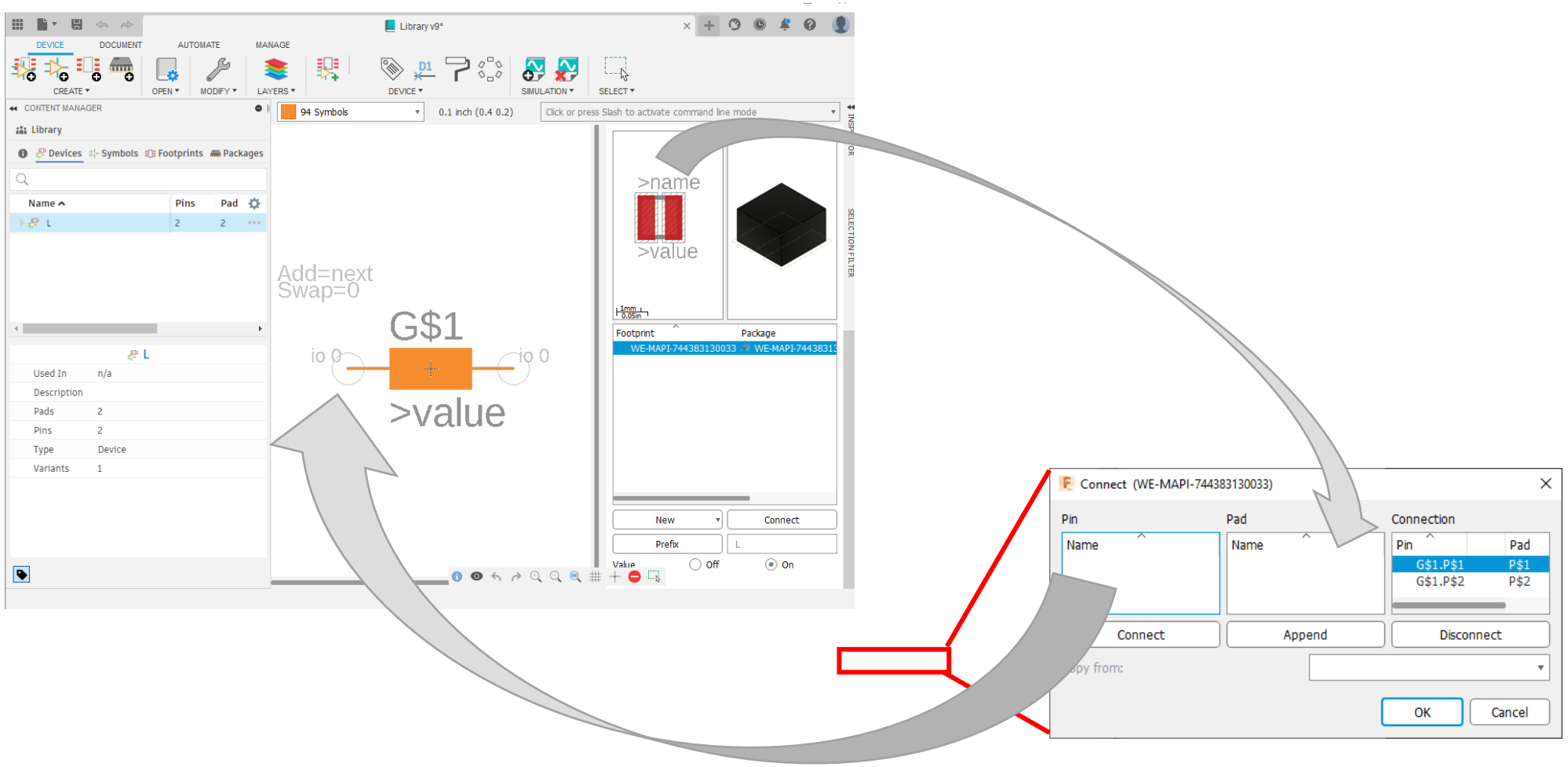
ENTER PARAMS -> CLICK ADD



VOILA


PACKAGE
SELECT
FINISH
DON’T FORGET TO ALIGN WITH YOUR FOOTPRINT

PIN-PAD CONNECTION



THE FINAL DEVICE / COMPONENT

MAKE SURE SYMBOL AND FOOTPRINT NAMES MATCH!
(DOES NOT HAVE TO BUT IT SAVES TIME AND ENERGY)

1. CLICK CREATE COMPONENT -> TYPE A NAME. THEN: