# EP1001 FURTHER DIGITAL FABRICATION & PROTOTYPING

Electronics Production

EASYEDA LIBRARY SETUP – STD EDITION

- Download Fab Academy parts library from gitlab repository
  - Eagle library
  - Kicad library

- Start EasyEDA Std Edition

- Click File > Open > Eagle. Click OK, then search for downloaded Eagle library (fab.lbr)

- Select "Extract Libs" and click Import

- Check "Select All" and click Add to Library

- To use the library in Schematic Capture, click Place Symbol. In the Dialog box, select EasyEDA / Symbol / My Libraries > All

- To import Fab Academy Kicad library, follow the steps above but choose kicad-master.zip
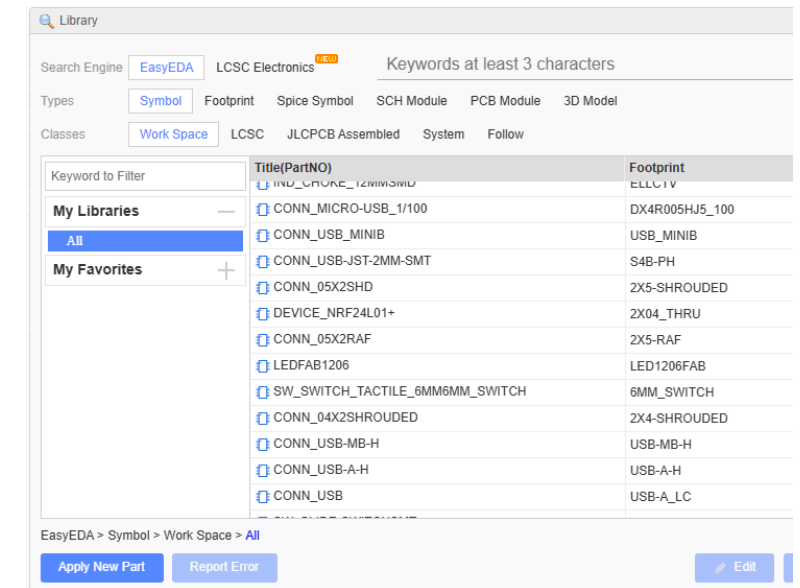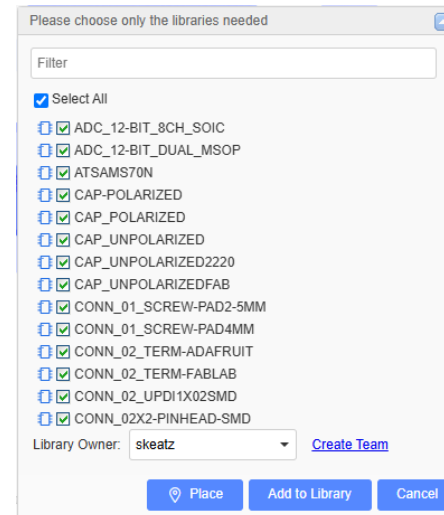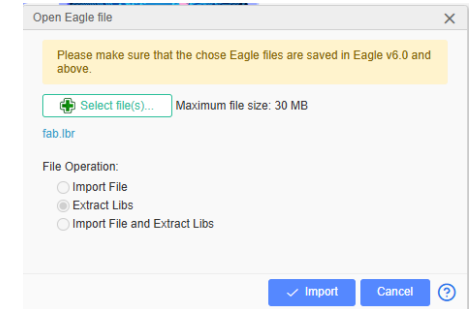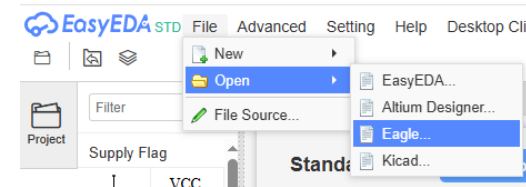
EASYEDA – DRAWING THE SCHEMATIC

- Start EasyEDA Std

- Double-click on schematic file to open

- Place components (Shift-F)

- Connect components

- Label Nets, component Labels & Values
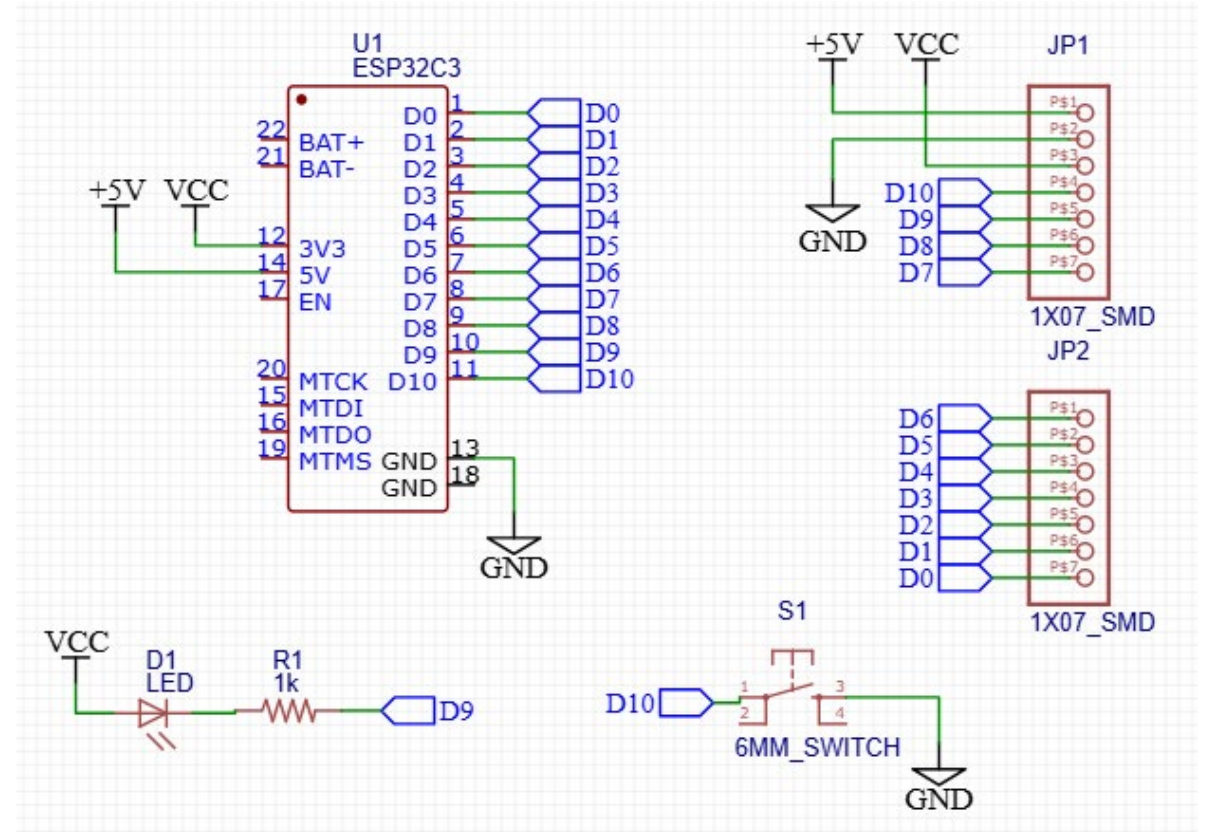
EASYEDA – LAYING OUT PCB

- Click Design > Convert Schematic to PCB to switch to PCB layout view

- Place components on the board

- Set design rules for your board (Design > Design rule: trace width 16 mil, clearance 24 mil)

- Route the ratsnests, adjusting components and traces as required

- Run Design Rule Check (Design > Check DRC)
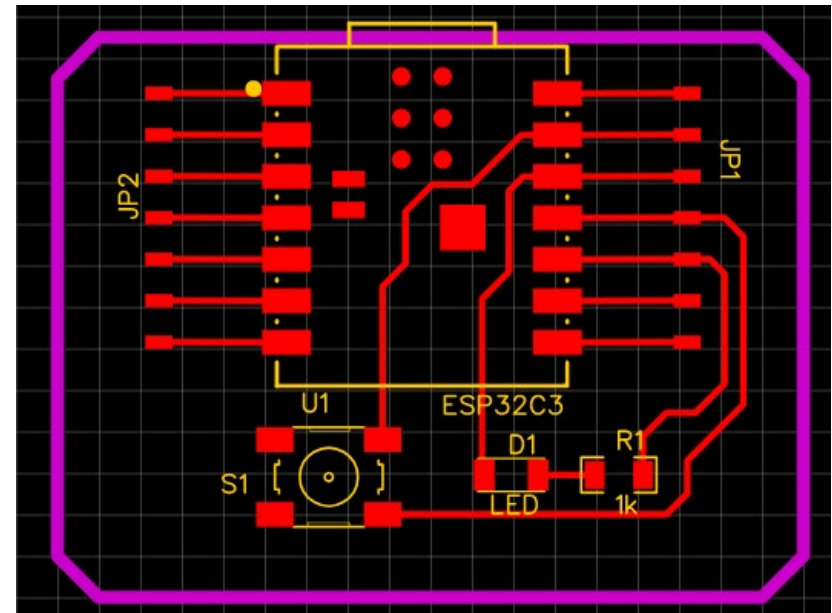
# EASYEDA – SAVING IMAGES FOR CNC

- Disable visibility for all layers, then turn on visibility for TopLayer and BoardOutline layers

- Click File > Export > SVG to export the images for CNC milling

- In the dialog box, select Separated layer, White on Black and layers to be exported

- Click on Export button to export the board layers

- Extract the top and board outline layers for pcb milling using mods
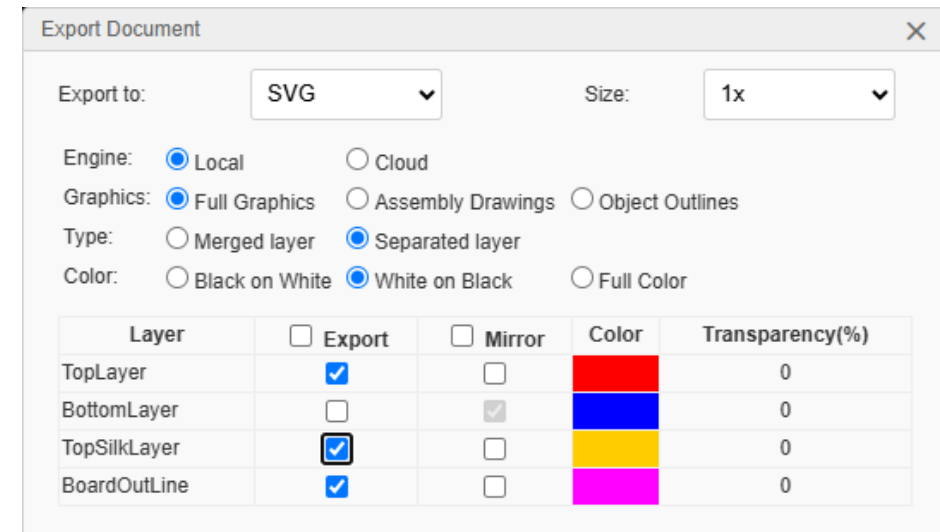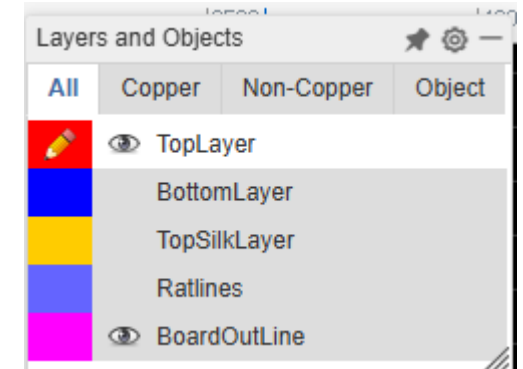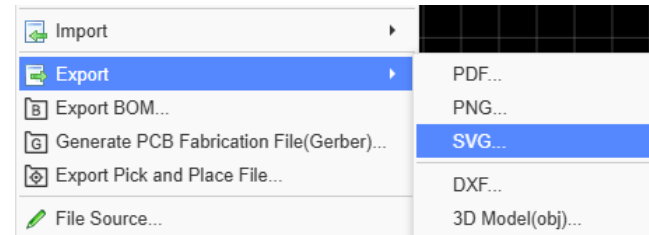
# EASYEDA – SAMPLE OUTPUT IMAGES

- Note 1: the trace and board outline images have to be the same dimensions for mods

- Note 2: you may have to "clean-up" your image(s) before bringing them into mods to generate the gcode

- Note 3: in mods, white areas are retained, black areas are milled

After clean-up
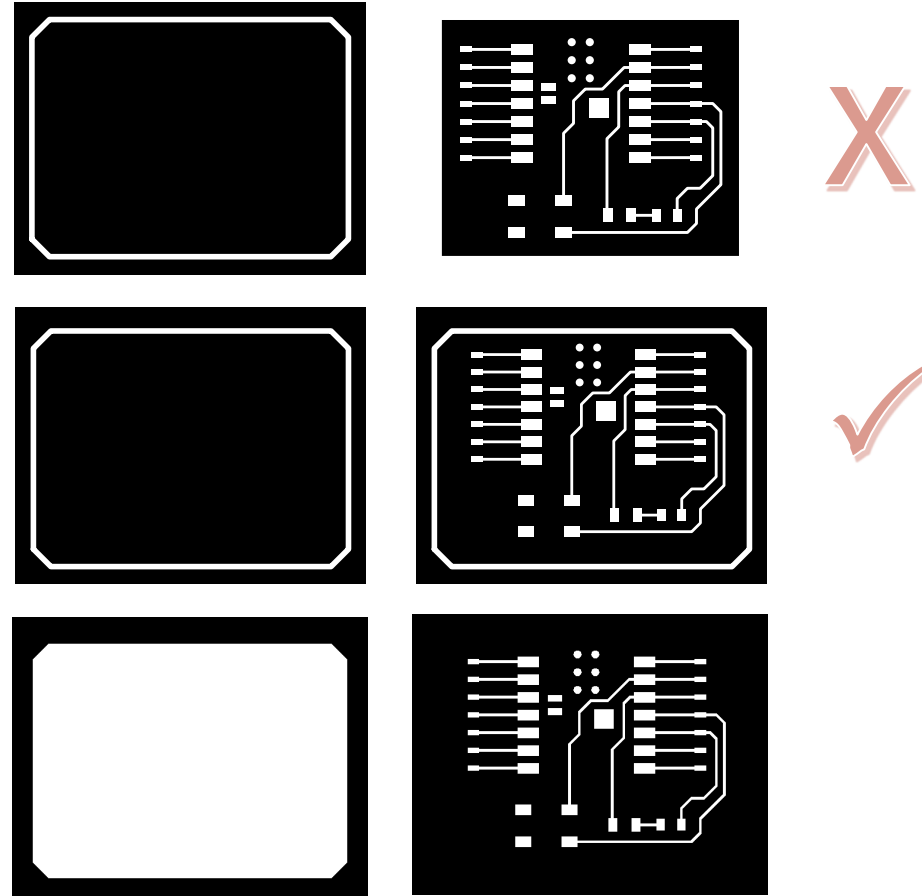
EASYEDA – GERBER OUTPUT

- To generate gerber output, click File > Generate PCB Fabrication File (Gerber)

- Select Generate Gerber at the dialog box. Save the file

- The gerber zip file contains multiple files meant for PCB house

- To fabricate pcbs inhouse using gerber output, you can use FlatCAM
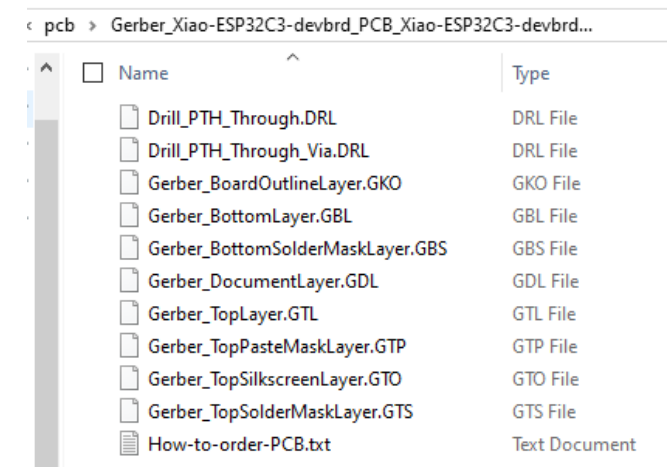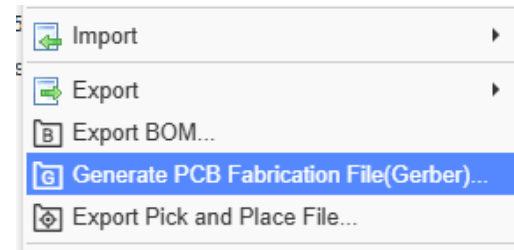
FUSION ELECTRONICS – LIBRARY SETUP

- To add Fab library to Fusion, click Library (tab) > Open Library Manager
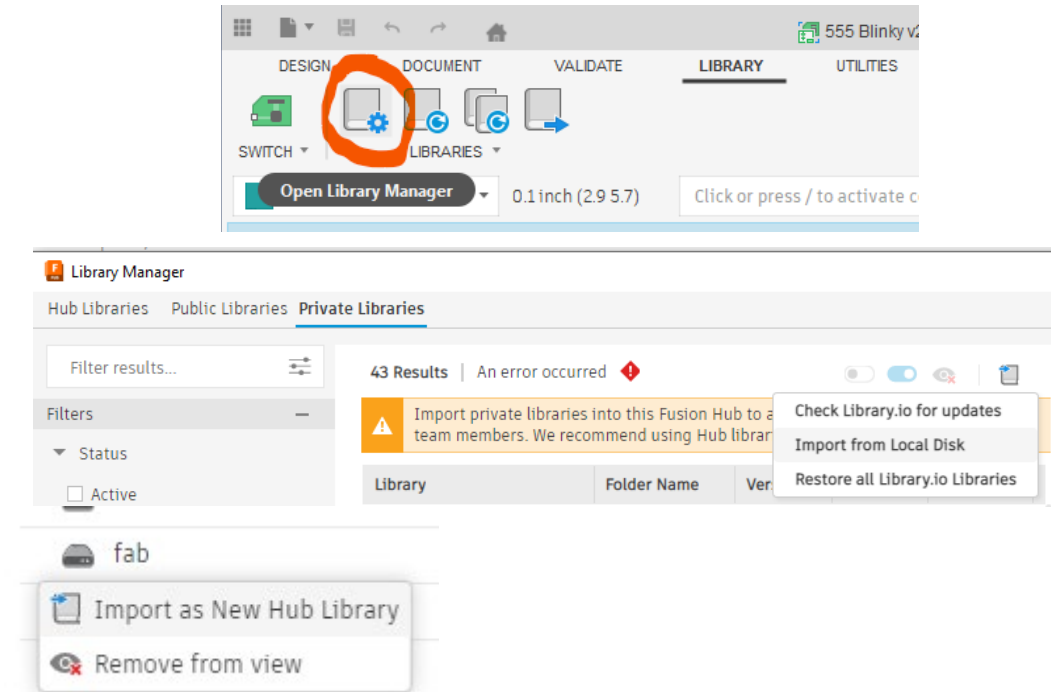
- In the dialog box, click Private Libraries > Sync Libraries > Import from Local Disk

- Locate the libraries folder on your local disk and click Open

- Select the desired library, right-click and select Import as New Hub Library

- In the dialog box, click Import into Fusion Hub. Save the library

# FUSION ELECTRONICS – USING CUSTOM LIBRARIES

- To add a component from the library to your schematic, click Place > Place Components (A)

- In the Place Components tab on the right, use Filter to facilitate your search

- When placing components, click on the Variant drop-down to select the correct footprint for your component

- Once your schematic is complete, click SWITCH to switch to PCB layout module
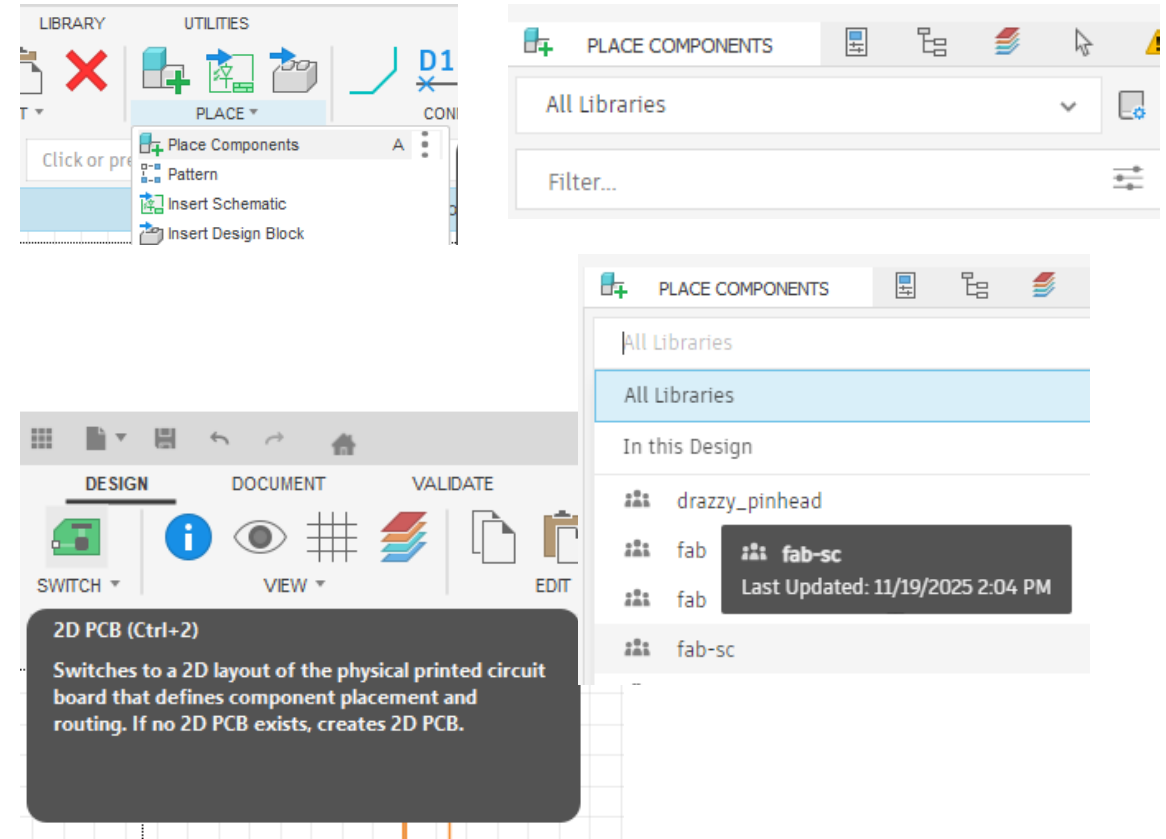
FUSION ELECTRONICS GENERATING IMAGE FOR MILLING PCB

- Once PCB layout is complete, to generate images for milling the PCB:
    - Hide unnecessary layers
    - Make visible only layers required for machining (Top + Pads, BoardOutline/Milling)
    - Turn off Display pad names and Display signal names on pads in Preferences
    - Turn off PCB origins in View settings

- Click Documents tab, then Outputs > Export. Select Export > Image

- In the dialog box, click Monochrome, Resolution = 1000 ~ 2000 dpi and Browse, enter name of your image

- Click OK to export the png image

- Do the same for BoardOutline/Milling layer
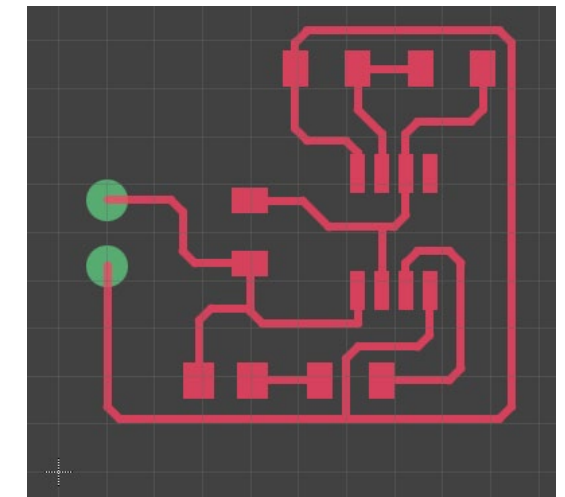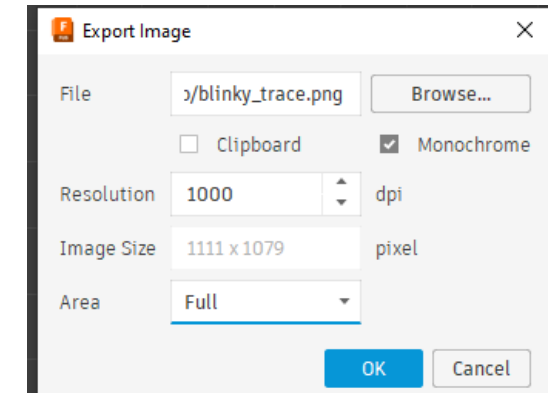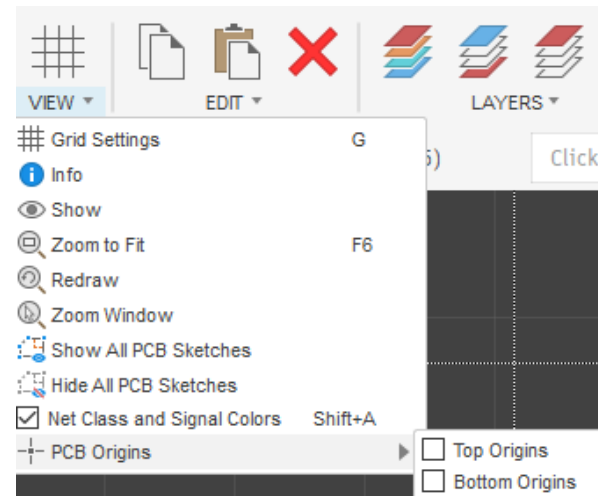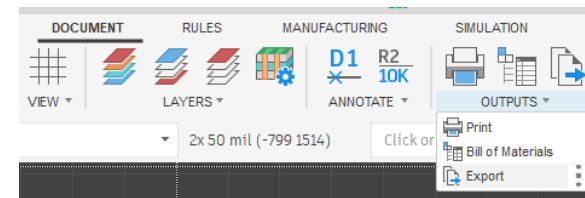
ELECTRONICS PRODUCTION

Fabricating the PCB

WHY MAKE PCBS?

- Customized electronics to meet production requirements

- Customized pcb footprint for product

- Greater reliability

- Ability to prototype and iterate faster

- Better product integration

SKILLS COVERED

- Understanding PCB fabrication workflow

- Producing files for PCB fabrication (from EDA software)

- Operating CNC for PCB milling

- Populating（stuffing）PCB

- Soldering PCB

- Testing PCB
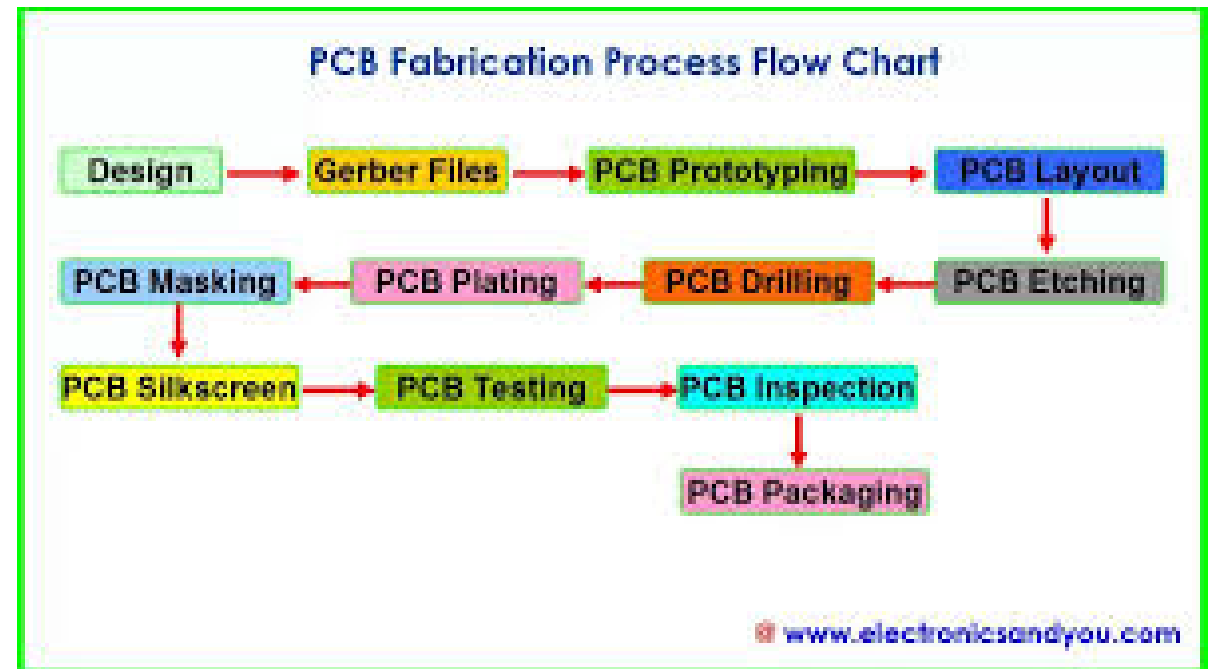
PCB FABRICATION PROCESS

- Etching
    - lithography, tape transfer
    - Ferric/cupric chloride
    - Ammonium/sodium persulfate
    - Citric acid, hydrogen peroxide
    - Safety: ferric chloride SDS
    - Waste disposal
- Flexible PCBs: vinyl cutting
- Laser cutter
- 3D Printing
- Sewing: conductive thread

- Machining
    - Generic CNCs: Stepcraft, Roland SRM-20
    - Dedicated PCB machine: LKPF S104, Wegstr
    - Tools:
        - Flat endmills: 0.010", 1/64", 1/32"
        - Flat endmills: 0.2mm, 0.4mm, 0.8mm
        - Engraving bits: 0.1mm/30 deg
    - Fixturing: clamps, double-sided tape
    - Zeroing
    - Deburring
    - Cleaning

PCB MATERIALS

- Rigid
  - FR1 – phenolic paper
  - FR4 – epoxy glass
- Flexible
  - Kapton, pyralux
  - 3M #1 epoxy film
  - 3M #1126 copper tape
- Typical copper thickness
  - 0.5 oz: 17.5 um
  - 1.0 oz: 35 um
  - 2.0 oz: 70 um

Property	FR-4	FR-2	FR-1
Electrical Insulation	Excellent	Good	Fair
Mechanical Strength	High	Medium	Low
Thermal Stability	High	Medium	Low
Flame Retardancy	Yes	Yes	No
Cost	Moderate	Low	Low
Availability	Widely Available	Limited	Limited
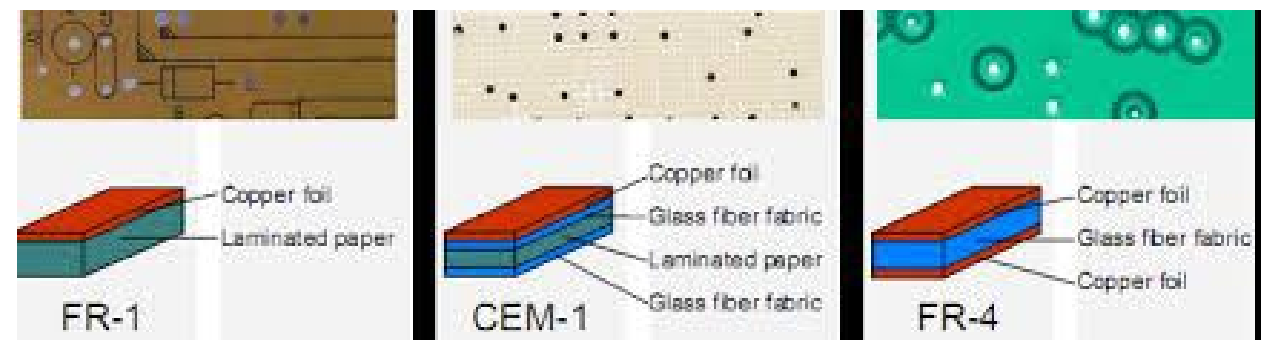
OTHER ALTERNATIVES

Board Houses

- PCBWay, JCLPCB
- Design rules
  - Track width: 15 mils
  - Clearance: 25 mils
- # layers:
  - 1, 2 or multi-layer
- Mechanical:
  - Drill, soldermask, silkscreen
  - Vias
  - Rivets

In-house Protyping

- (Solderless) Breadboards
- Stripboard
- Veroboard
- Perfboard
- Wire-wrapping

CONVERTING IMAGE TO GCODE IN MODS

- Go to mods website:
  - https://skeatz.github.io/mods

- Right-click > program > open server program > G-code > mill 2D PCB png/svg

- Click "select png/svg file" and select your pcb trace file

- Click "mill traces (1/64)" to copy cnc settings for milling traces

- Click "calculate" to generate the gcode

- View the gcode and verify that all the pads and traces in your layout show up

- Repeat for the board outline. Remember to click "mill outline (1/32) to copy the settings for milling the pcb cutout

- Note: board outline is milled to depth of 1.68mm with offset value of 1