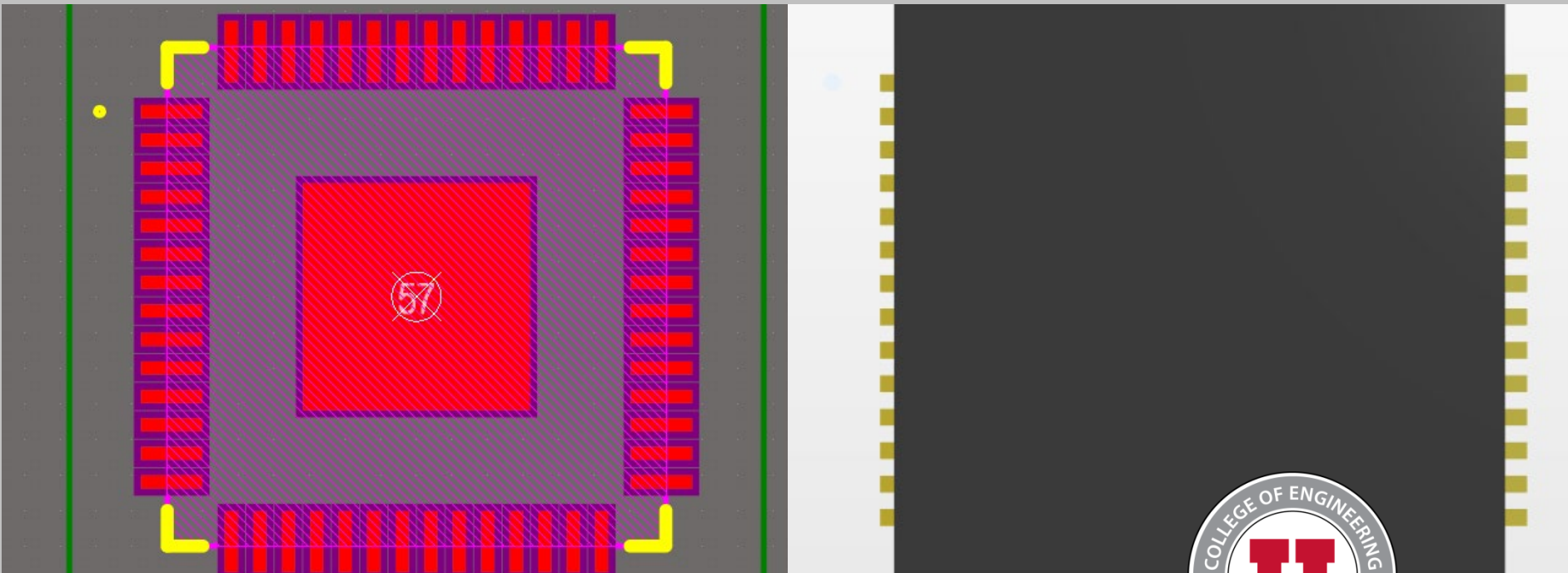# IEEE PCB Design Workshop: (Week 04)
Footprints and PCB Layout

Hosted By: Adrian Sucahyo and IEEE at the University of Utah
Adapted From: IEEE x FSAE Workshop SP25 with Nick Howard and Adrian Sucahyo

Workshop Outline

Tentative Schedule:

- Sept. 3 – Introduction to Schematics
- Sept. 10 – Schematics and Components
- Sept. 17 – Introduction to PCB Layout
- Sept. 24 – Footprints and PCB Layout
- Oct. 1 – Open Work Session
- ** FALL BREAK **
- Oct. 22 – Soldering Week 1
- Oct. 29 – Soldering Week 2
- Nov. 5 – Soldering Week 3
- Nov. 12 – Final Notes and Next Steps

Announcements

- ASUU Budget
  - We have been approved!
  - We will <u>NOT</u> have a fee to get your board manufactured for the soldering portion of the workshop

- Alternative Projects
  - We will be able to get alternative project boards manufactured if submitted by the deadline.
  - Limited to the 10 cm x 10 cm dimensions outlined by JLCPCB.
  - Talk or email me if you have any questions!

- Board Submission Deadline!
  - October 2nd, 11:59 PM

  - Submit Gerber files to get them manufactured with the reference design

Want more experience?

- Consider joining the FSAE tractive team!
  - The Tractive Team is currently looking for students to assist with designing and assembling the electrical system for an electric formula-style race car!
  - No experience required!

U of U FSAE Discord Link

Join the IEEE Discord

- If you haven't already, please join the IEEE Discord server for additional information and updates regarding this workshop
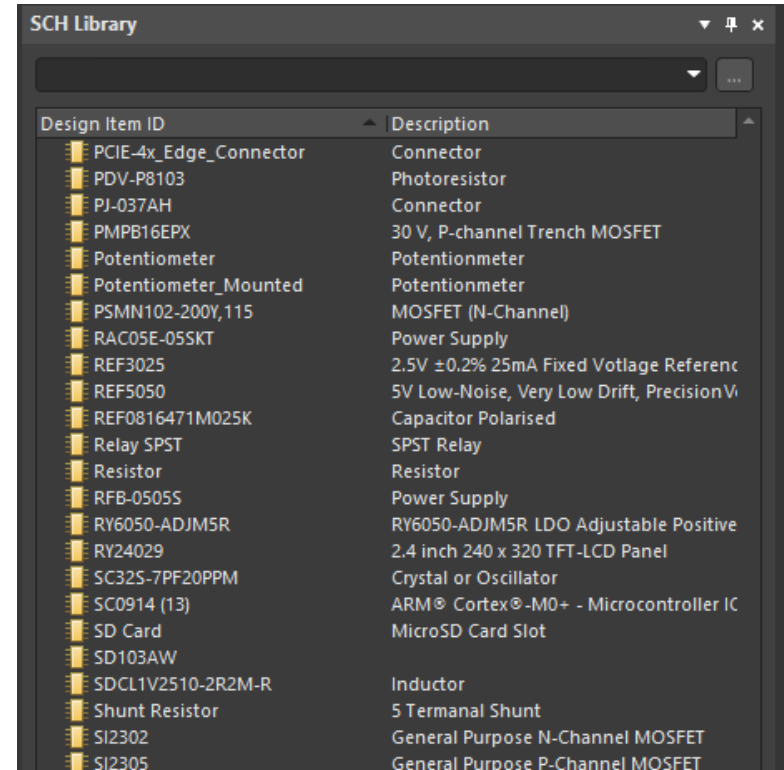
U of U IEEE Discord Link

Component Libraries Refresher

- Libraries are collections of components and footprints
  - All components for a project must be derived from a library
  - Projects may reference multiple different component libraries
  - Relatively consistent across platforms
- Typically, there are symbol and footprint libraries
 - Tightly coupled together

SCH Library

Design Item ID	Description
PCIE-4x_Edge_Connector	Connector
PDV-P8103	Photoresistor
PJ-037AH	Connector
PMPB16EPX	30 V, P-channel Trench MOSFET
Potentiometer	Potentionmeter
Potentiometer_Mounted	Potentionmeter
PSMN102-200Y,115	MOSFET (N-Channel)
RAC05E-05SKT	Power Supply
REF3025	2.5V ±0.2% 25mA Fixed Votlage Referenc
REF5050	5V Low-Noise, Very Low Drift, Precision V
REF0816471M025K	Capacitor Polarised
Relay SPST	SPST Relay
Resistor	Resistor
RFB-0505S	Power Supply
RY6050-ADJM5R	RY6050-ADJM5R LDO Adjustable Positive
RY24029	2.4 inch 240 x 320 TFT-LCD Panel
SC32S-7PF20PPM	Crystal or Oscillator
SC0914 (13)	ARM® Cortex®-M0+ - Microcontroller IC
SD Card	MicroSD Card Slot
SD103AW	
SDCL1V2510-2R2M-R	Inductor
Shunt Resistor	5 Termanal Shunt
SI2302	General Purpose N-Channel MOSFET
SI2305	General Purpose P-Channel MOSFET

Component Library Types (Altium)

- ## .SchLib

  - Contains all component schematic symbols

- ## .PcbLib

  - Contains all footprint information

- ## .LibPkg

  - Also known as an integrated library

  - Packages multiple .SchLib and .PcbLib libraries together for easier management

TD.PcbLib

TD.SchLib

TD_Lib.LibPkg
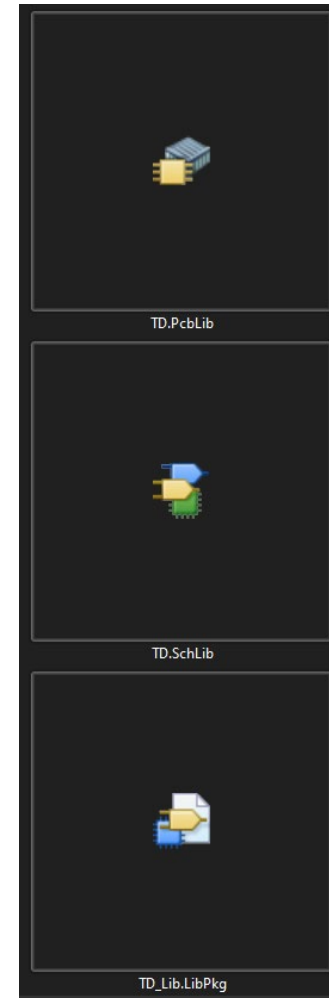
Component Library Types (KiCAD)

- ## .kicad_sym
  - Contains all the symbol information for components in the library

- ## .pretty
  - Contains all the footprint information to be used with components
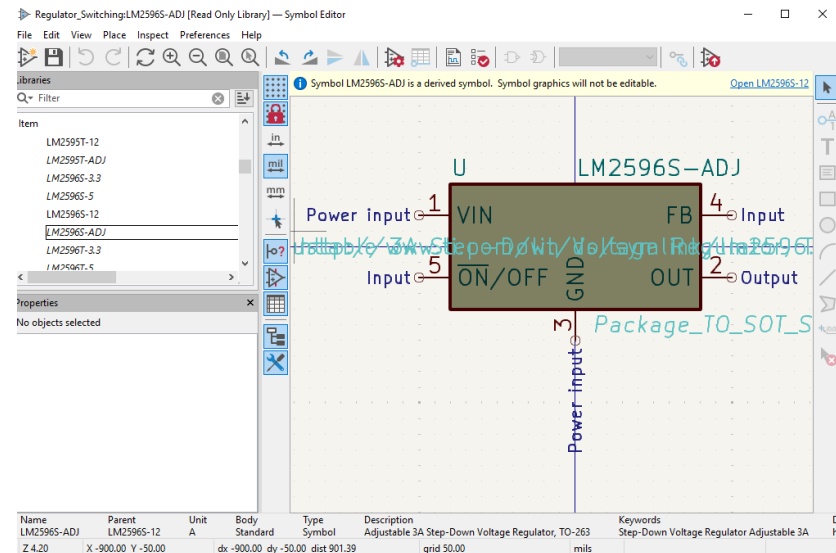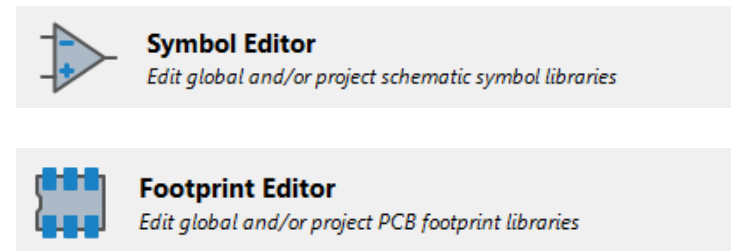
Standard vs. Non-Standard Footprints

- Many components are in "standard" packages
  - Key parameters are standardized across manufacturers
- Certain components are in "non-standard" footprints due to custom features or other restrictions

https://www.indiamart.com/proddetail/smd-electronic-component-2856299588912.html

Resistors and Capacitors

- Many components are in "standard" packages
  - Key parameters are standardized across manufacturers
- Certain components are in "non-standard" footprints due to custom features or other restrictions

Imperial Size Code **Metric Size Code**

Imperial Size Code	Metric Size Code
01005	0402
0201	0603
0402	**1005**
0603	**1608**
0805	2012
1008	2520
1206	3216
1210	3225
1806	4516
1812	4532
2010	5025
2512	6332

.1"x.1"
(100mils)

.5"x.5"
(500mils)

Size
Reference

1x1cm

1x1cm

Size
Reference

Component Sizes (common in **bold**)

DEPARTMENT OF
Electrical & Computer Engineering
COLLEGE OF ENGINEERING | THE UNIVERSITY OF UTAH

Gerber Files

- Gerbers (or Gerber Files) are the files sent to the manufacturer that will be used to make the boards

- Engineers at the fab review files for manufacturability and then send it off to production.

- Gerber files can be generated from EDA software but needs to be reviewed.

https://www.onethesis.com/gerbers-to-footprint/

DEPARTMENT OF
Electrical & Computer Engineering
COLLEGE OF ENGINEERING | THE UNIVERSITY OF UTAH
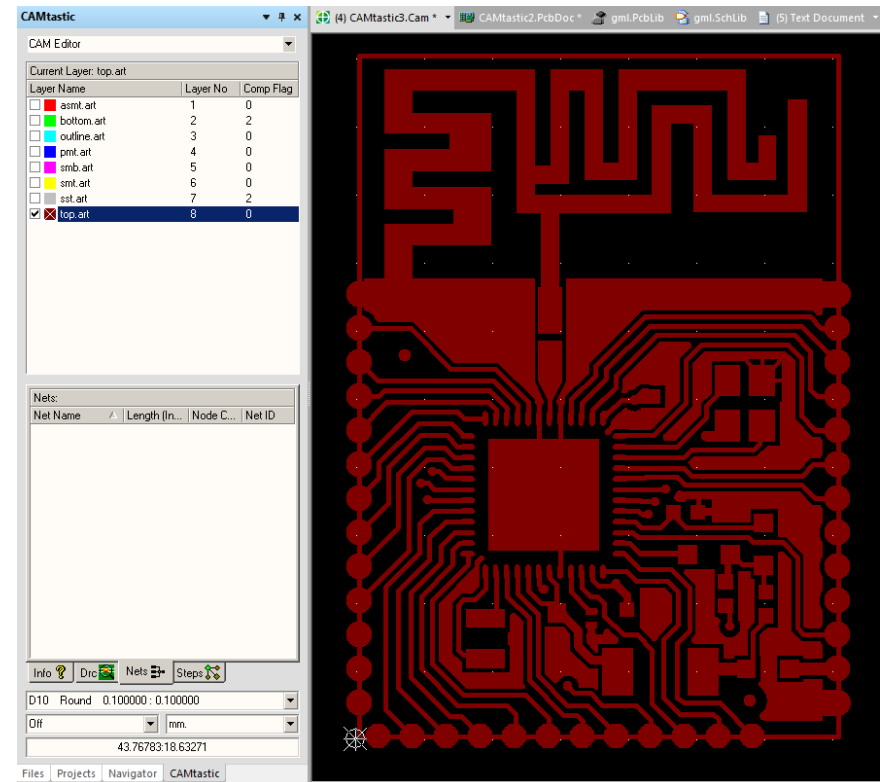
NC Drill Files

- NC Drill Files contain the information for the drill holes of the PCB.
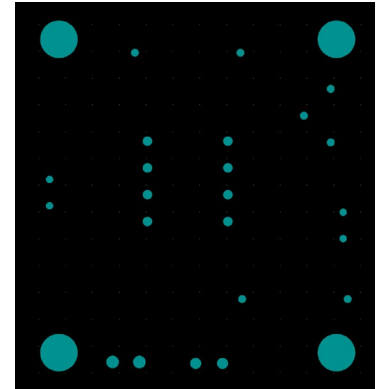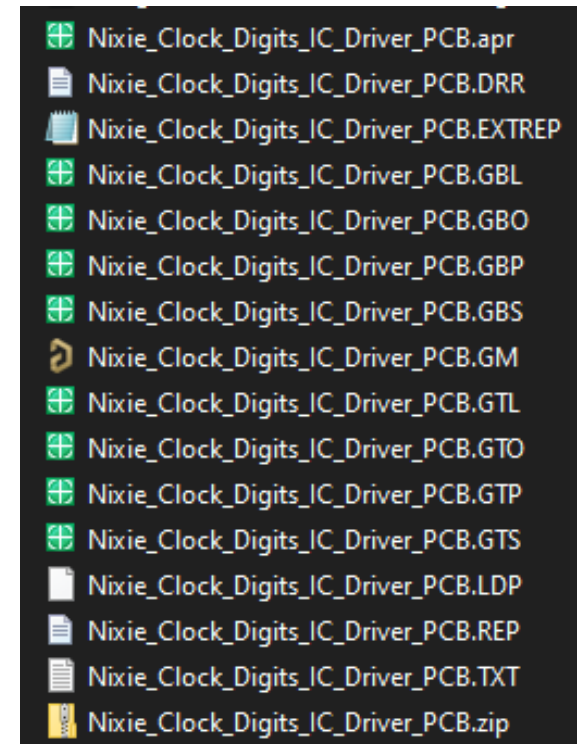 - These are generated separately from the Gerber Files


```
-------------------------------------------------------------------------------------------------------
NCDrill File Report For: Workshop Project.PcbDoc    10/29/2024  1:09:38 AM
-------------------------------------------------------------------------------------------------------

Layer Pair : Top Layer to Bottom Layer
ASCII RoundHoles File : Workshop Project.TXT

Tool      Hole Size            Hole Tolerance          Hole Type     Hole Count  Plated      Tool Travel
-------------------------------------------------------------------------------------------------------
T1        28mil (0.7mm)                                Round         4           PTH         1.29inch (32.82mm)
T2        30mil (0.75mm)                               Round         7           PTH         2.02inch (51.35mm)
T3        35mil (0.9mm)                                Round         8           PTH         0.90inch (22.86mm)
T4        41mil (1.05mm)                               Round         2           PTH         0.10inch (2.54mm)
T5        47mil (1.19mm)                               Round         2           PTH         0.10inch (2.54mm)
T6        140mil (3.556mm)                             Round         4           PTH         3.24inch (82.30mm)
-------------------------------------------------------------------------------------------------------
Totals                                                               27

Total Processing Time (hh:mm:ss) : 00:00:00
```

Gerber File and Drill File Types

- Each file represents a different layer or operation type.
- Common files:
  - .gto, .gbo = top/bottom silkscreen
  - .gts, .gbs = top/bottom solder mask
  - .gtp, .gbp = top/bottom solder paste
  - .gtl, .gbl = top/bottom copper
  - .gm = board outline
  - .drl / .txt = CNC drill

Nixie_Clock_Digits_IC_Driver_PCB.apr
Nixie_Clock_Digits_IC_Driver_PCB.DRR
Nixie_Clock_Digits_IC_Driver_PCB.EXTREP
Nixie_Clock_Digits_IC_Driver_PCB.GBL
Nixie_Clock_Digits_IC_Driver_PCB.GBO
Nixie_Clock_Digits_IC_Driver_PCB.GBP
Nixie_Clock_Digits_IC_Driver_PCB.GBS
Nixie_Clock_Digits_IC_Driver_PCB.GM
Nixie_Clock_Digits_IC_Driver_PCB.GTL
Nixie_Clock_Digits_IC_Driver_PCB.GTO
Nixie_Clock_Digits_IC_Driver_PCB.GTP
Nixie_Clock_Digits_IC_Driver_PCB.GTS
Nixie_Clock_Digits_IC_Driver_PCB.LDP
Nixie_Clock_Digits_IC_Driver_PCB.REP
Nixie_Clock_Digits_IC_Driver_PCB.TXT
Nixie_Clock_Digits_IC_Driver_PCB.zip

Questions?

Download Today's Project Files

Navigate to the workshop GitHub and download today's files

https://github.com/IEEE-U-of-U/IEEE-PCB-Workshop-Fall-2025