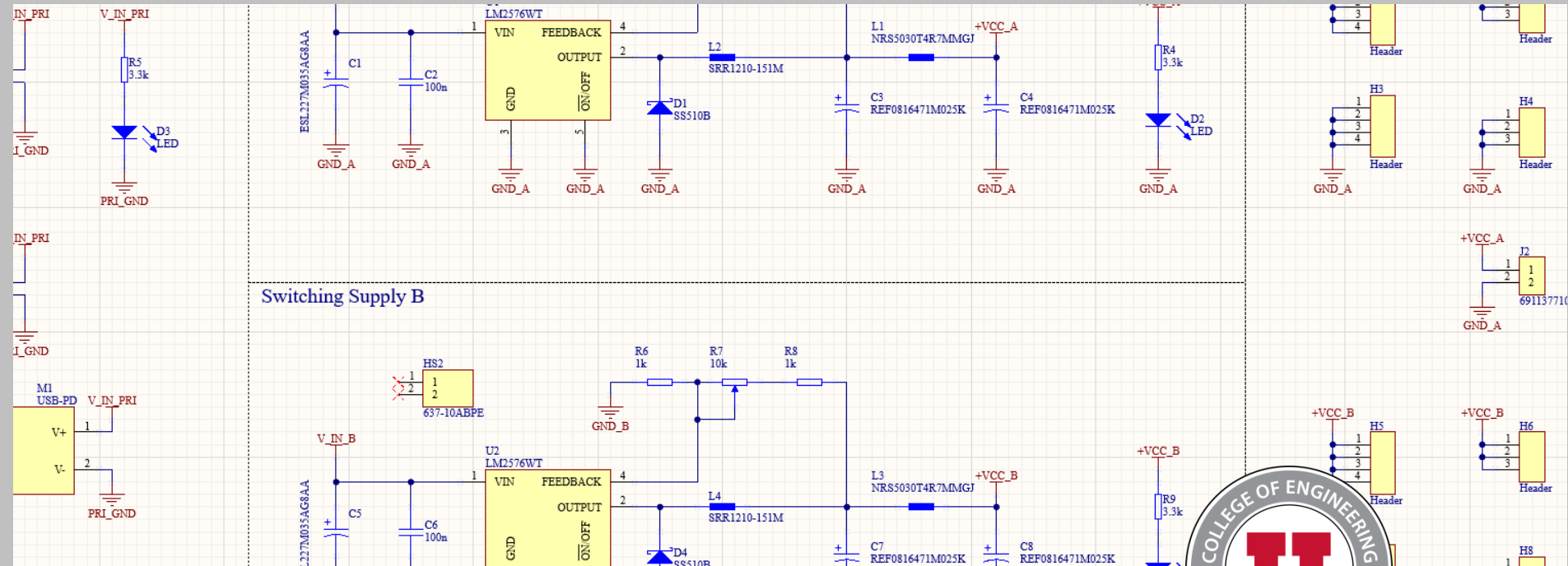
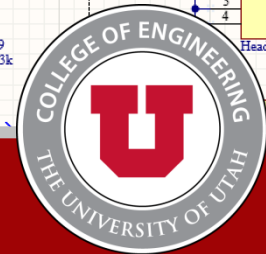


IEEE PCB Design Workshop: (Week 02) Schematics and Components



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 – Layout Continued
- 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 Requests
 - We are currently still waiting for ASUU to process our budget requests.
 - We will keep you updated through email when we hear back.
- 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!

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

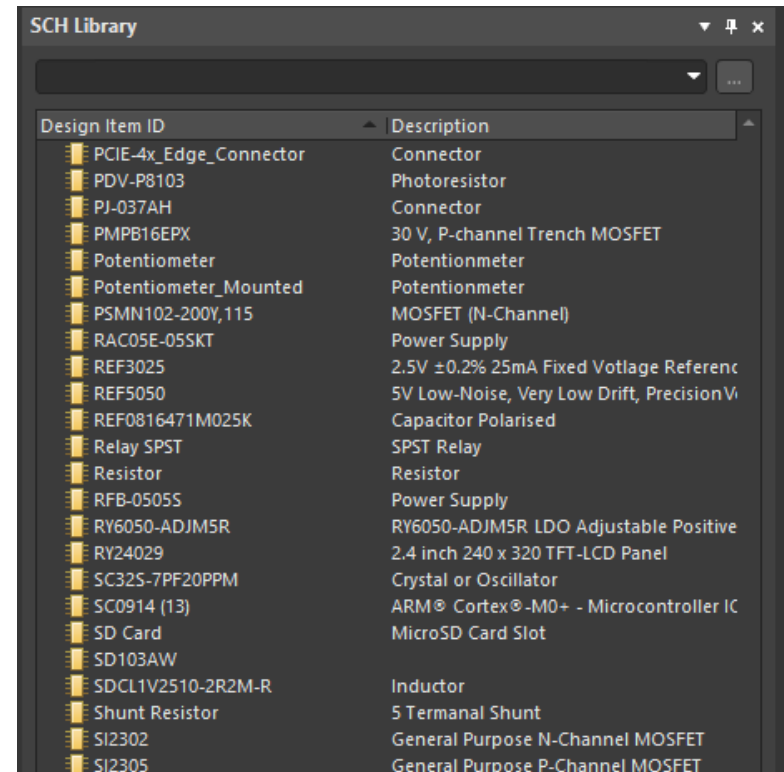


U of U IEEE Discord Link



Component Libraries

- 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

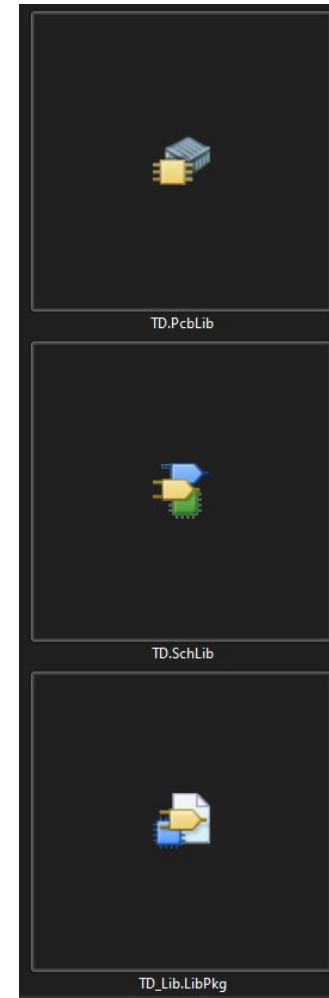


The screenshot shows a software window titled "SCH Library". It contains a table with two columns: "Design Item ID" and "Description". The table lists various electronic components and their specifications.

Design Item ID	Description
PCIE-4x_Edge_Connector	Connector
PDV-P8103	Photoresistor
PJ-037AH	Connector
PMPB16EPX	30 V, P-channel Trench MOSFET
Potentiometer	Potionmeter
Potentiometer_Mounted	Potionmeter
PSMN102-200Y,115	MOSFET (N-Channel)
RAC05E-05SKT	Power Supply
REF3025	2.5V \pm 0.2% 25mA Fixed Voltage Reference
REF5050	5V Low-Noise, Very Low Drift, Precision Voltage Reference
REF0816471M025K	Capacitor Polarised
Relay SPST	SPST Relay
Resistor	Resistor
RFB-05055	Power Supply
RY6050-ADJM5R	RY6050-ADJM5R LDO Adjustable Positive Regulator
RY24029	2.4 inch 240 x 320 TFT-LCD Panel
SC325-7PF20PPM	Crystal or Oscillator
SC0914 (13)	ARM [®] Cortex [®] -M0+ - Microcontroller IC
SD Card	MicroSD Card Slot
SD103AW	
SDCL1V2510-2R2M-R	Inductor
Shunt Resistor	5 Terminal Shunt
SI2302	General Purpose N-Channel MOSFET
SI2305	General Purpose P-Channel MOSFET

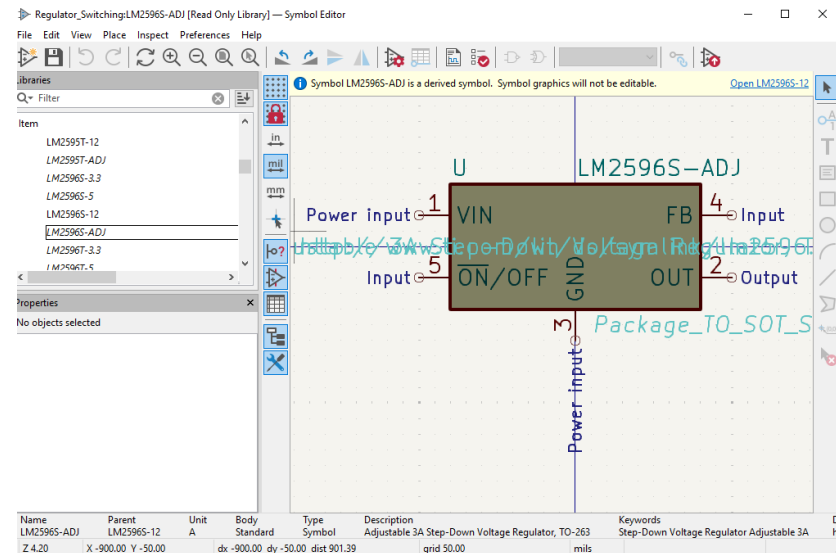
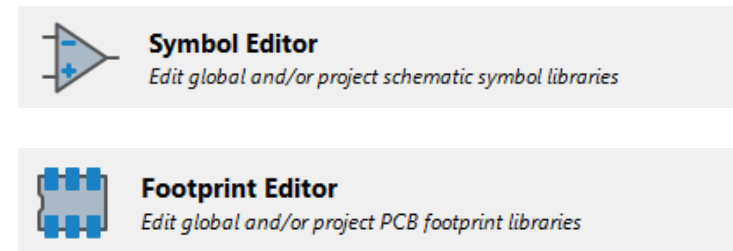
Component Library Types (Altium)

- .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



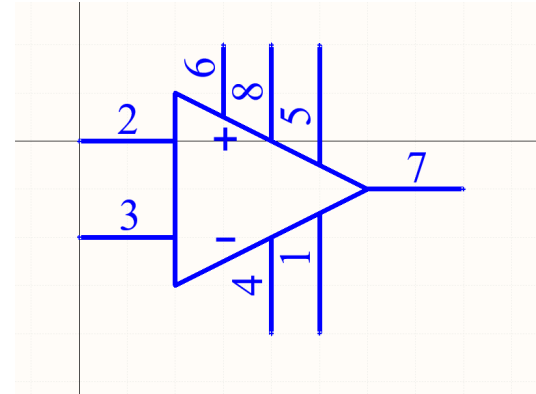
Component Library Types (KiCAD)

- .kicad_sym
 - Contains all the symbol information for components in the library
- .pretty
 - Contains all the footprint information to be used with components



Making Component Symbols

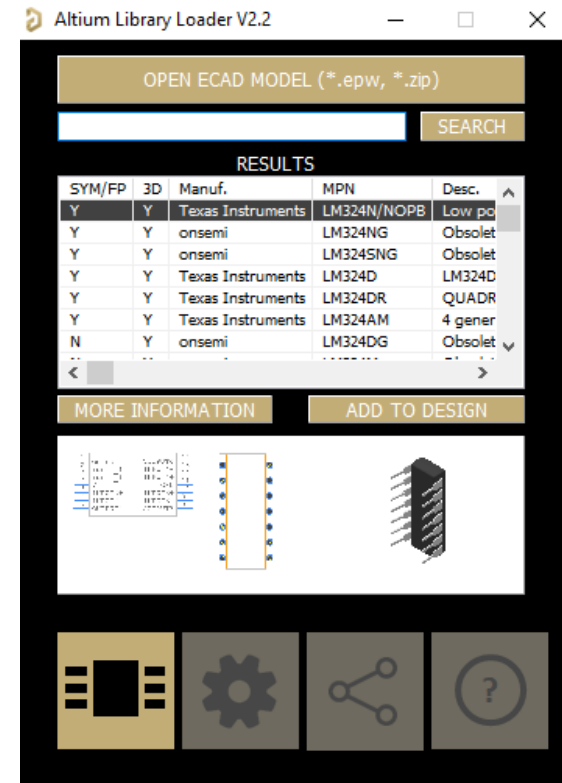
- Making symbols is a time-consuming process
- All pins and designators must be correct, or they will result in errors during layout
- Requires the ability to read a datasheet



5	OUT0	VDD	16
6	OUT1		
7	OUT2	SDI	2
8	OUT3	CLK	3
9	OUT4	LE/DM1	4
10	OUT5	SDO	14
11	OUT6	OE/DM2	13
12	OUT7		
1	GND	R-EXT	15

SamacSys (Component Manager)

- SamacSys is a third-party component manager
 - Integrates with Altium with an extension
 - Creates schematic symbols and footprints for you
 - Can be used with KiCAD as well



Questions?

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>