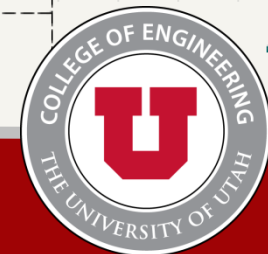
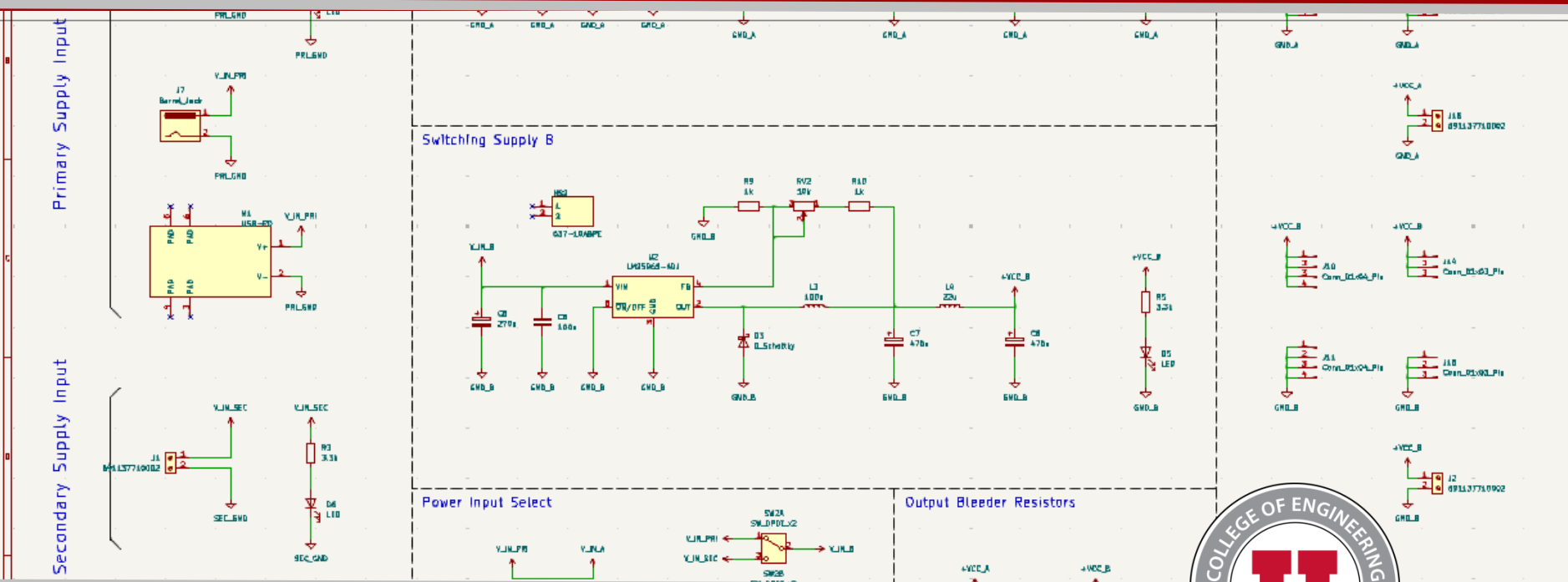


IEEE KiCAD Crash Course: (Week 02) Symbols and Footprints



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

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

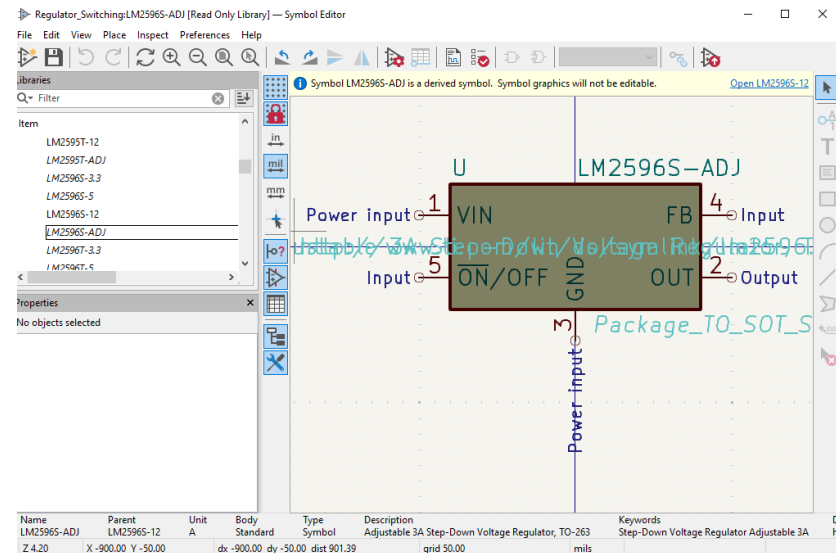
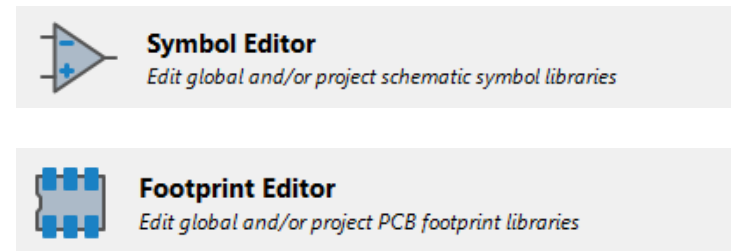


U of U IEEE Discord Link



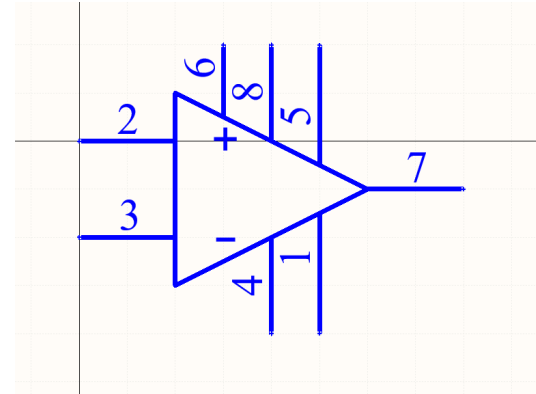
Component Library Types (KiCAD)

- .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

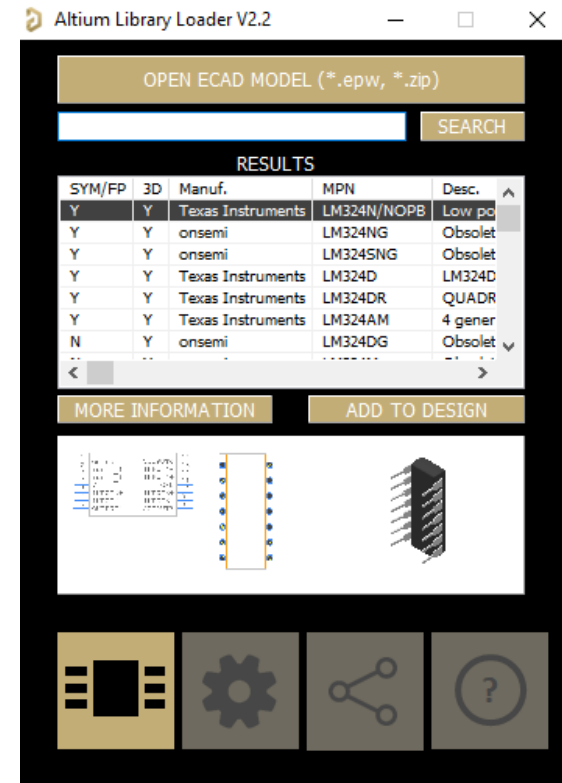
- Making symbols is a time-consuming process
- All pins and designators must be correct, or they will result in errors during layout
- Requires the ability to read a datasheet



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
 - Creates schematic symbols and footprints for you
 - Collaboration with Digikey or Mouser to help designers use their components
 - Free to use!



Component Footprints

- Footprints are the arrangement of pads that are used to attach components to the PCB
- Footprints must match the component to ensure
- Each component has a unique package, however there are standard sizes.
 - Ex: 0805, 0603, 0402
 - Ex: DIP, TSSOP, QFN

Solder Mask and Solder Paste Recommendations

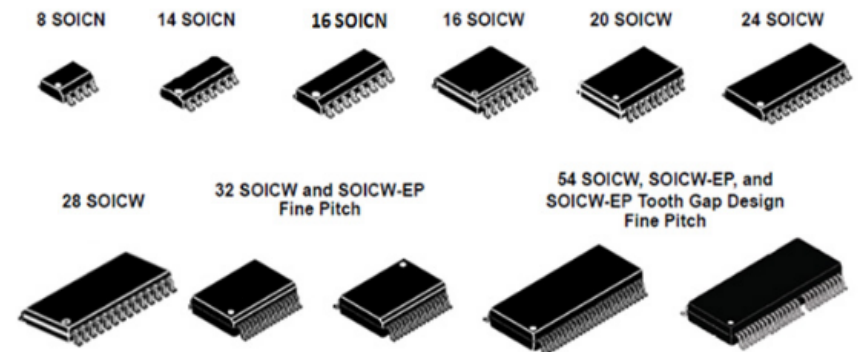
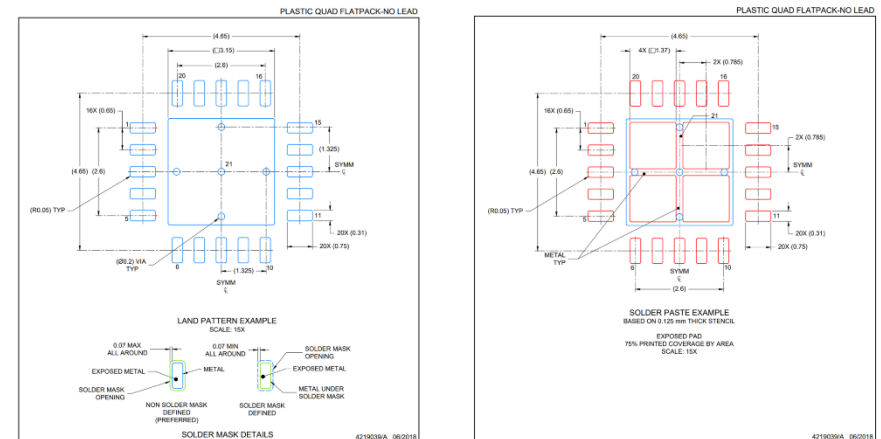
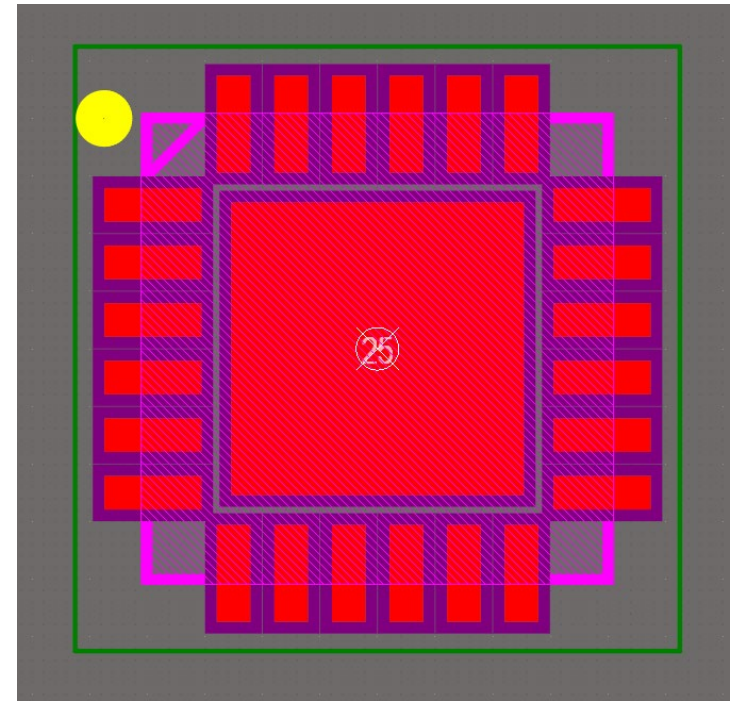


Figure 1. Standard Freescale SOIC Offerings

<https://buildember.com/2024/06/24/component-footprints-what-should-you-include/>
<https://www.nxp.com/docs/en/application-note/AN2409.pdf>

Footprint Libraries

- Very similar to component libraries, although only containing footprints
- Can be utilized across multiple projects
 - Managing a library of very common footprints can save a lot of time
 - SamacSys is also a good tool to help generate footprints quickly



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-KiCAD-Crash-Course>