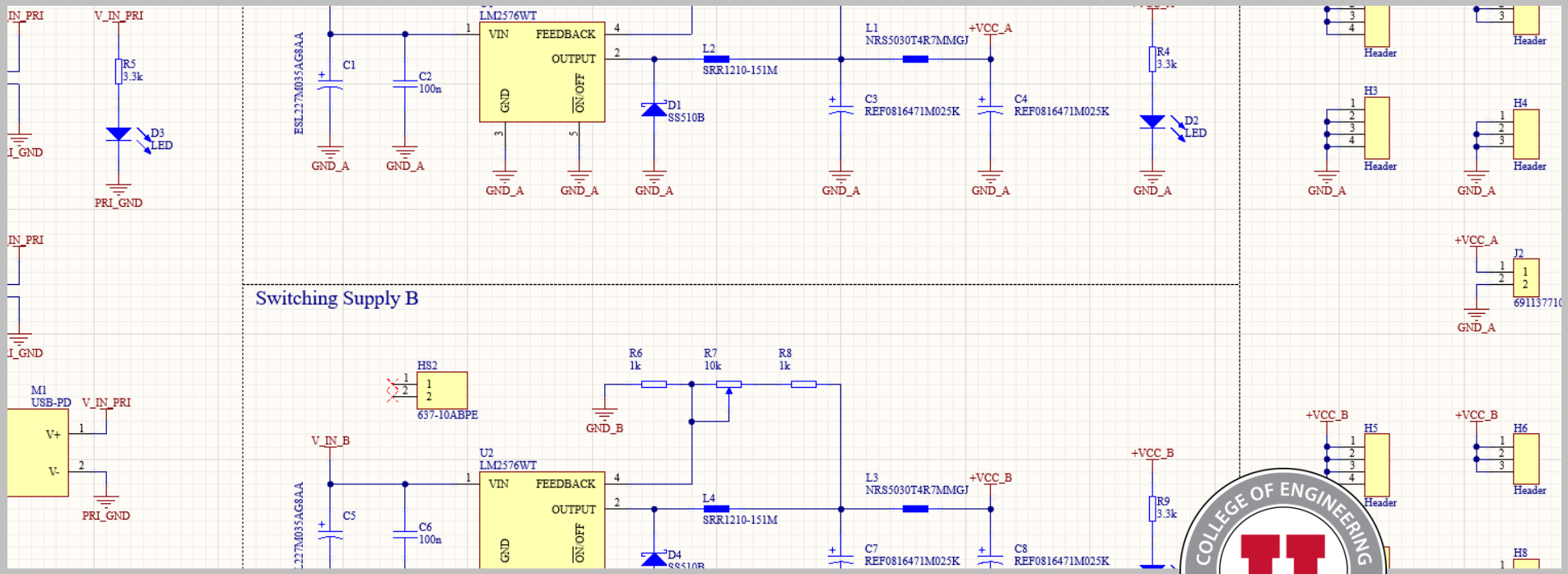
IEEE PCB Design Workshop: (Week 02)

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

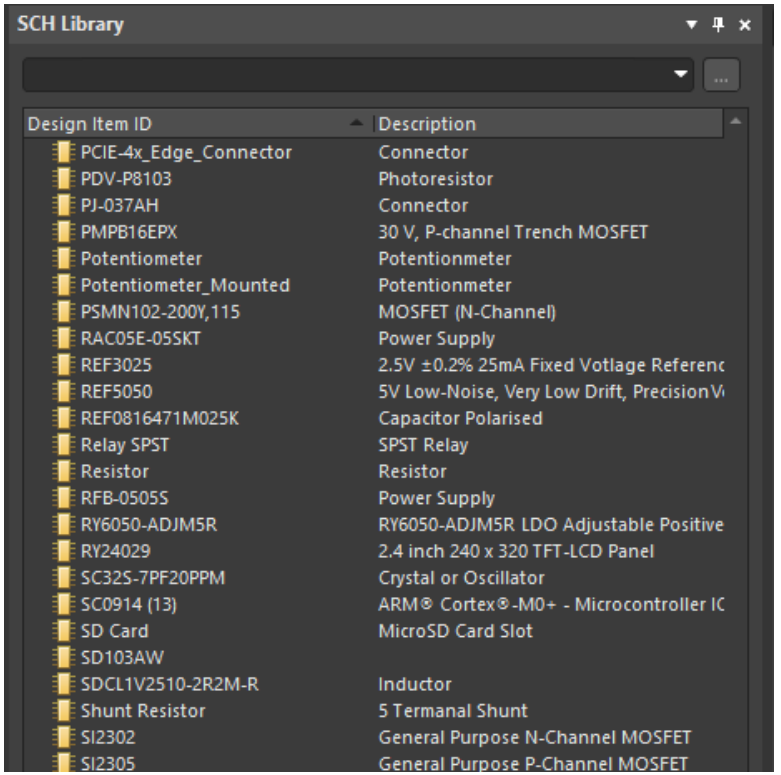
Join the IEEE Discord

- 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

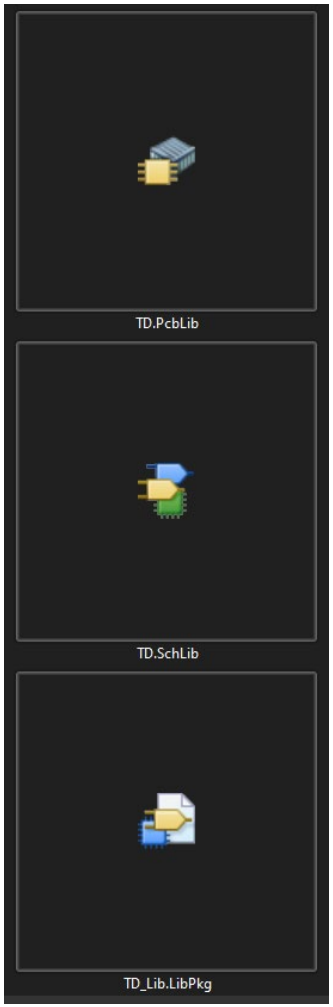
Component Libraries

- 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



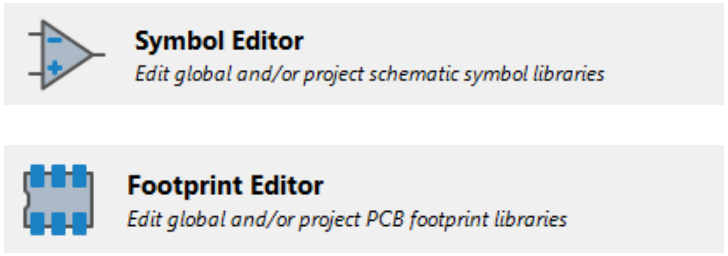
Component Library Types (Altium)

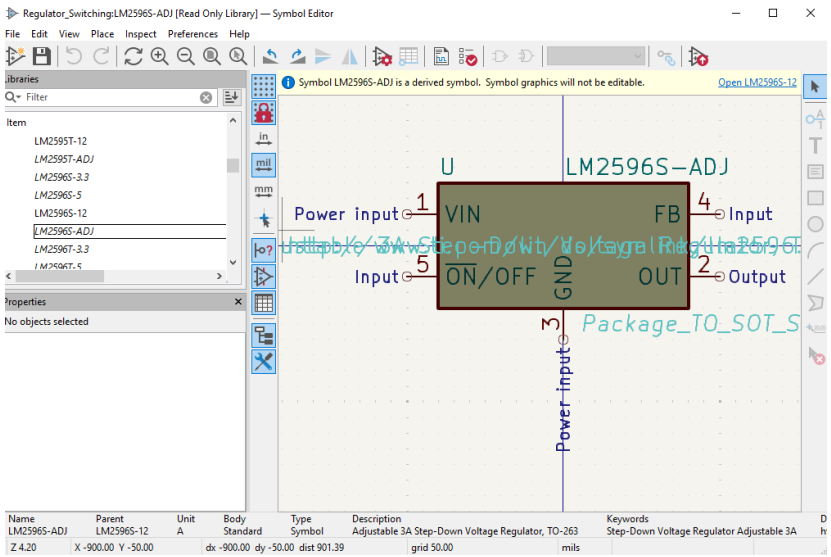
- .SchLib
  - Contains all component schematic symbols
- .PcbLib
  - Contains all footprint information
- .LibPkg
  - Also known as an integrated library
  - Packages multiple .SchLib and .PcbLib libraries together for easier management



Component Library Types (KiCAD)

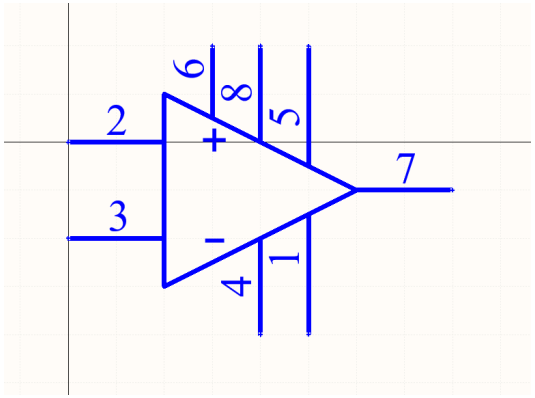
- .kicad_sym
  - Contains all the symbol information for components in the library
- .pretty
  - Contains all the footprint information to be used with components

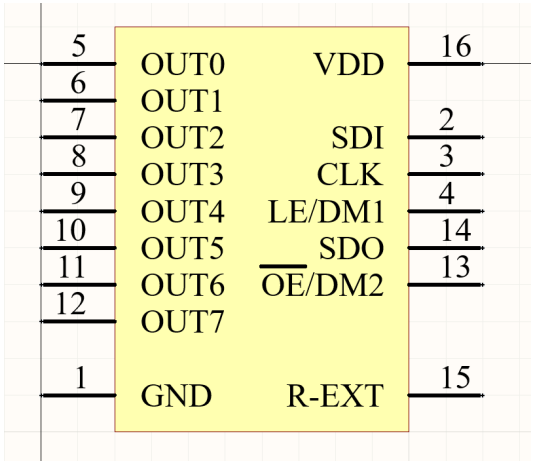




Making Component Symbols

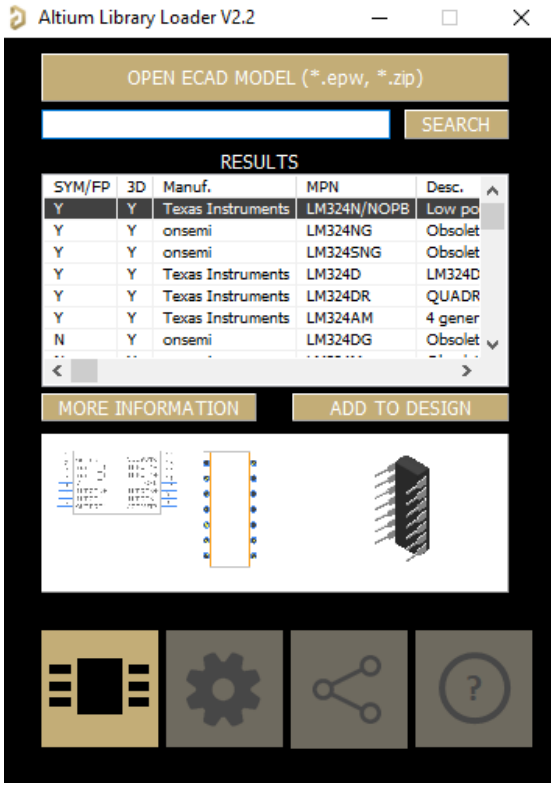
- Making symbols is a time-consuming process
- All pins and designators must be correct, or they will result in errors during layout
- Requires the ability to read a datasheet





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



SamacSys
ELECTRONIC COMPONENT LIBRARY SOLUTIONS

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