

KICAD is a package of PCB development software.

High level overview:

Schematic:

- layout circuit components called "symbols"
- make electrical connections
- ERC "Electronic Rules Checker"
- footprint assignment
 - ↳ Foot print is the layout the symbol takes up on the PCB

If a symbol or footprint doesn't exist:

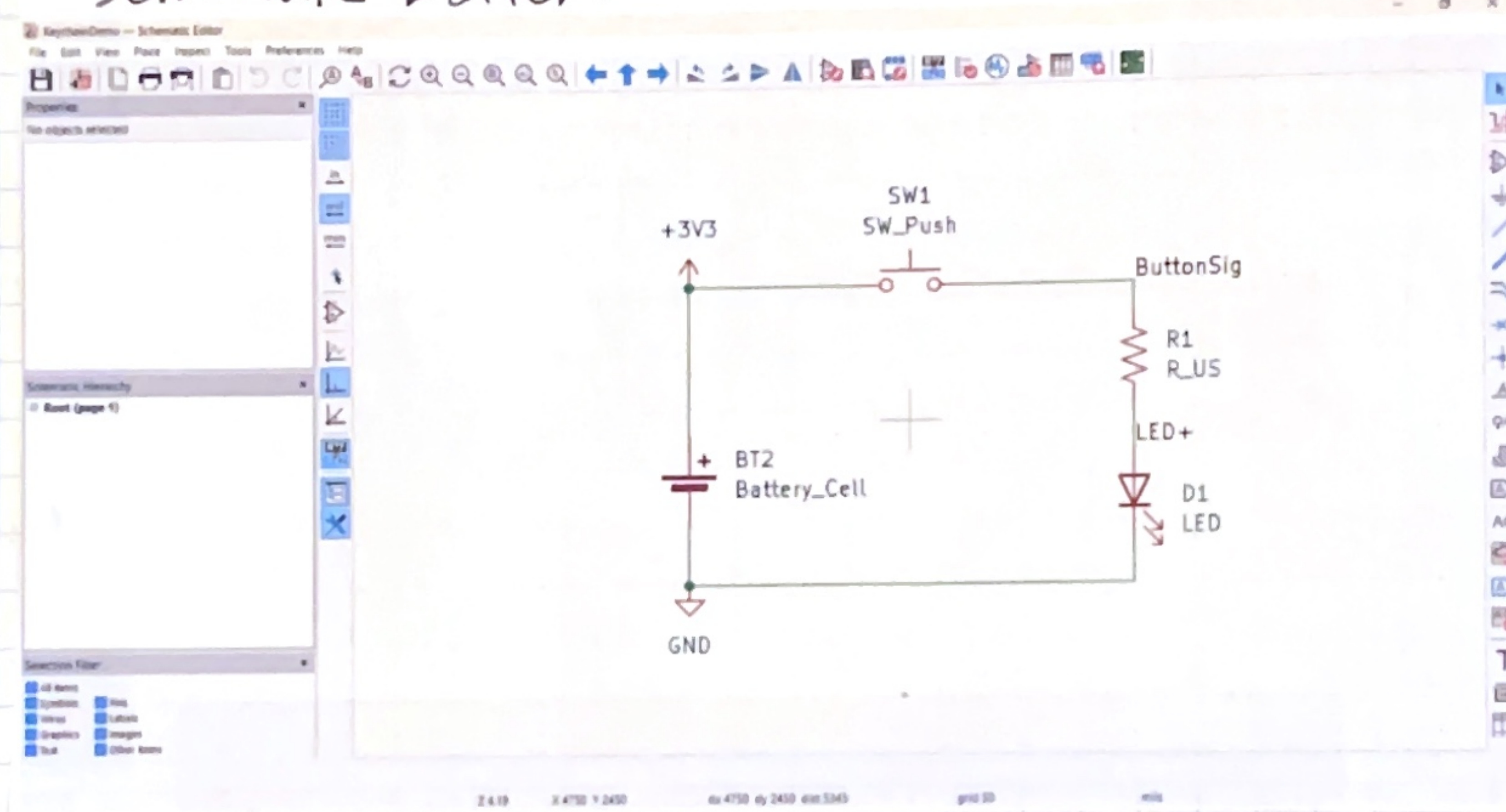
Use the data sheet and the symbol editor/footprint editor to create your own.

PCB Editor:

- Update components from schematic
- "Edge Cuts" layer for the board outline
- Layout footprints
- Route traces on front or back copper layer.
- Power and Ground Planes or "Filled Zones" (optional)
- 3D Model Viewer
- Export Gerbers



Schematic Editor:



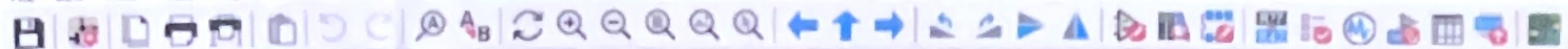
06/09/25
Wesley
Cochran

KICAD Workflows

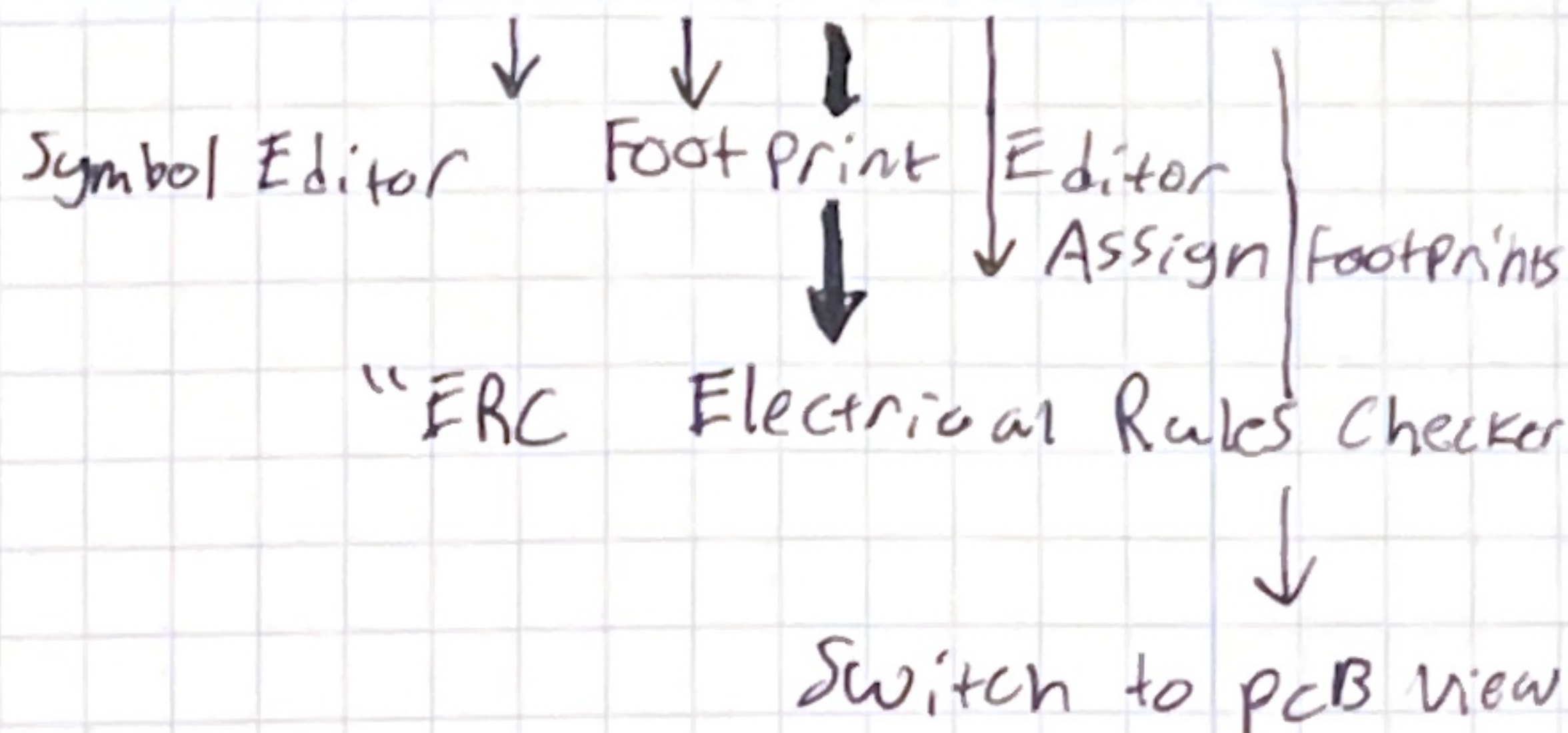
95

KeychainDemo - Schematic Editor

File Edit View Place Inspect Tools Preferences Help



Properties



- Select Items



- Place Symbols



- Place Power Symbols



- Draw Wires



- Draw Busses



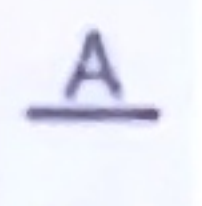
- No connect Flags



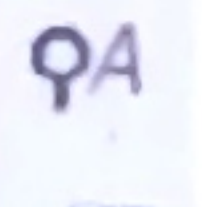
- Place Junctions



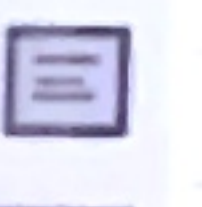
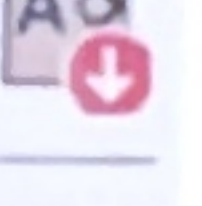
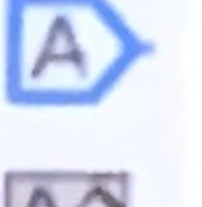
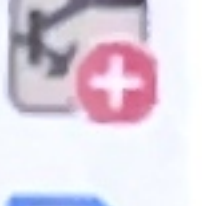
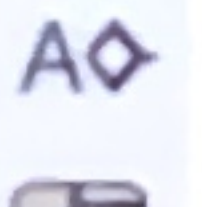
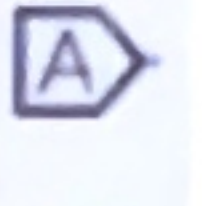
- Place Net labels

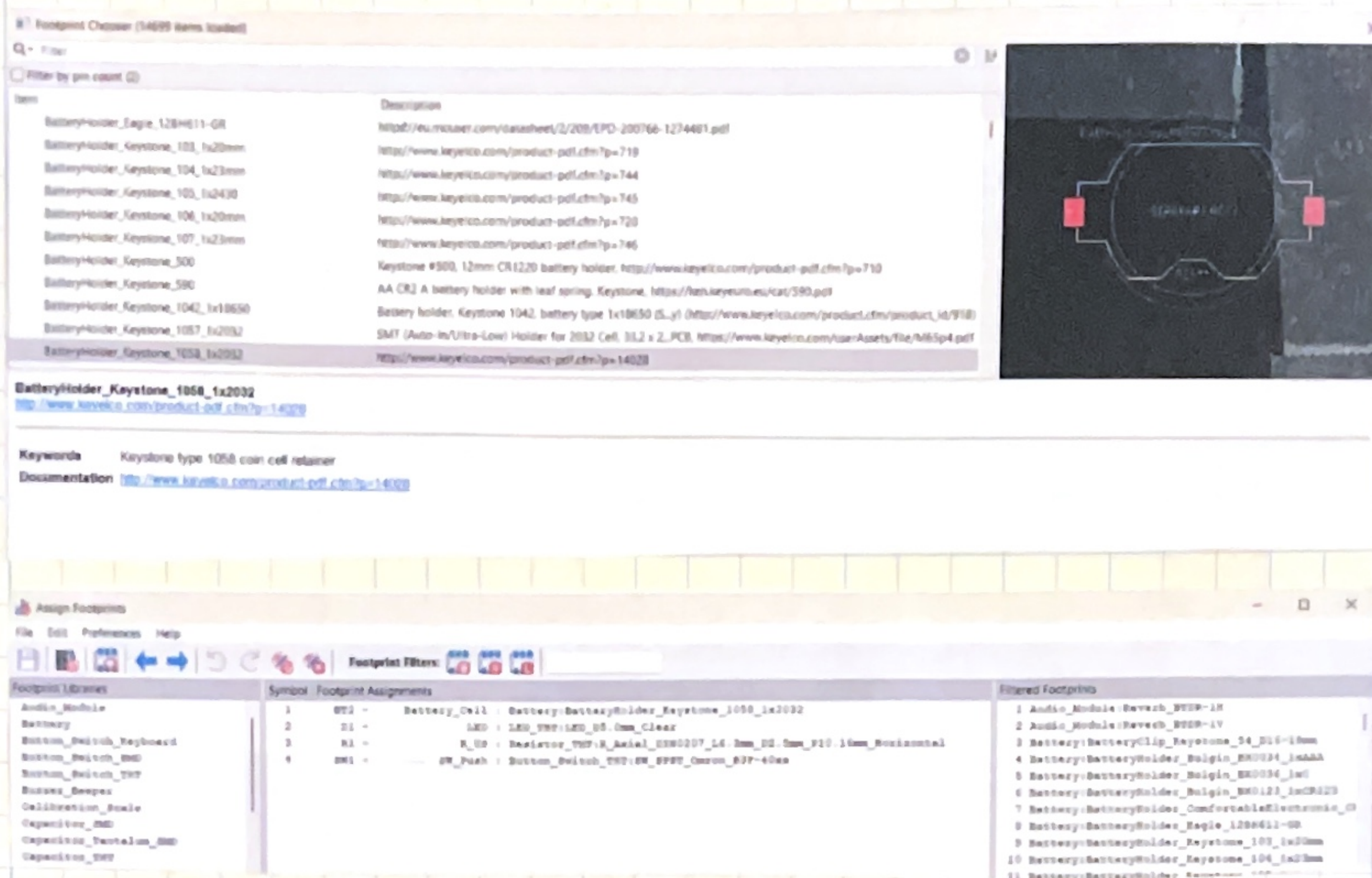


- Global labels



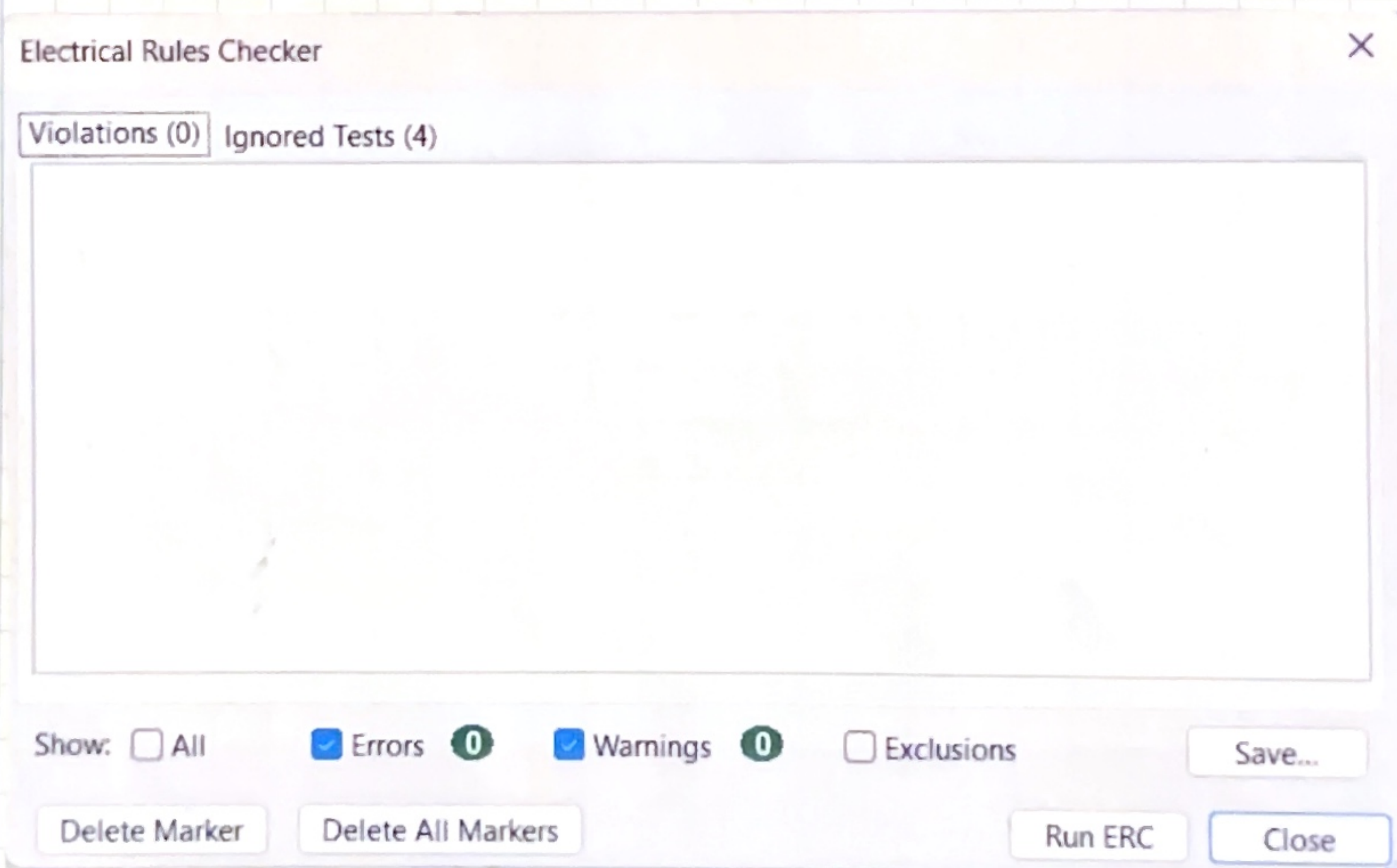
- Text





To assign footprints: Make sure the footprint matches the hardware you have.

Before moving on to the PCB editor, make sure the ERC has no errors

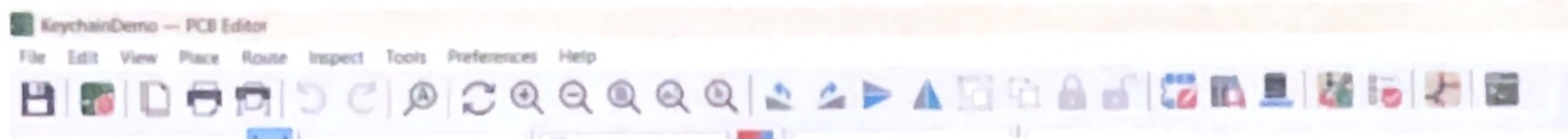


06/09/25
Wesley
Cochran

KICAD Workflows

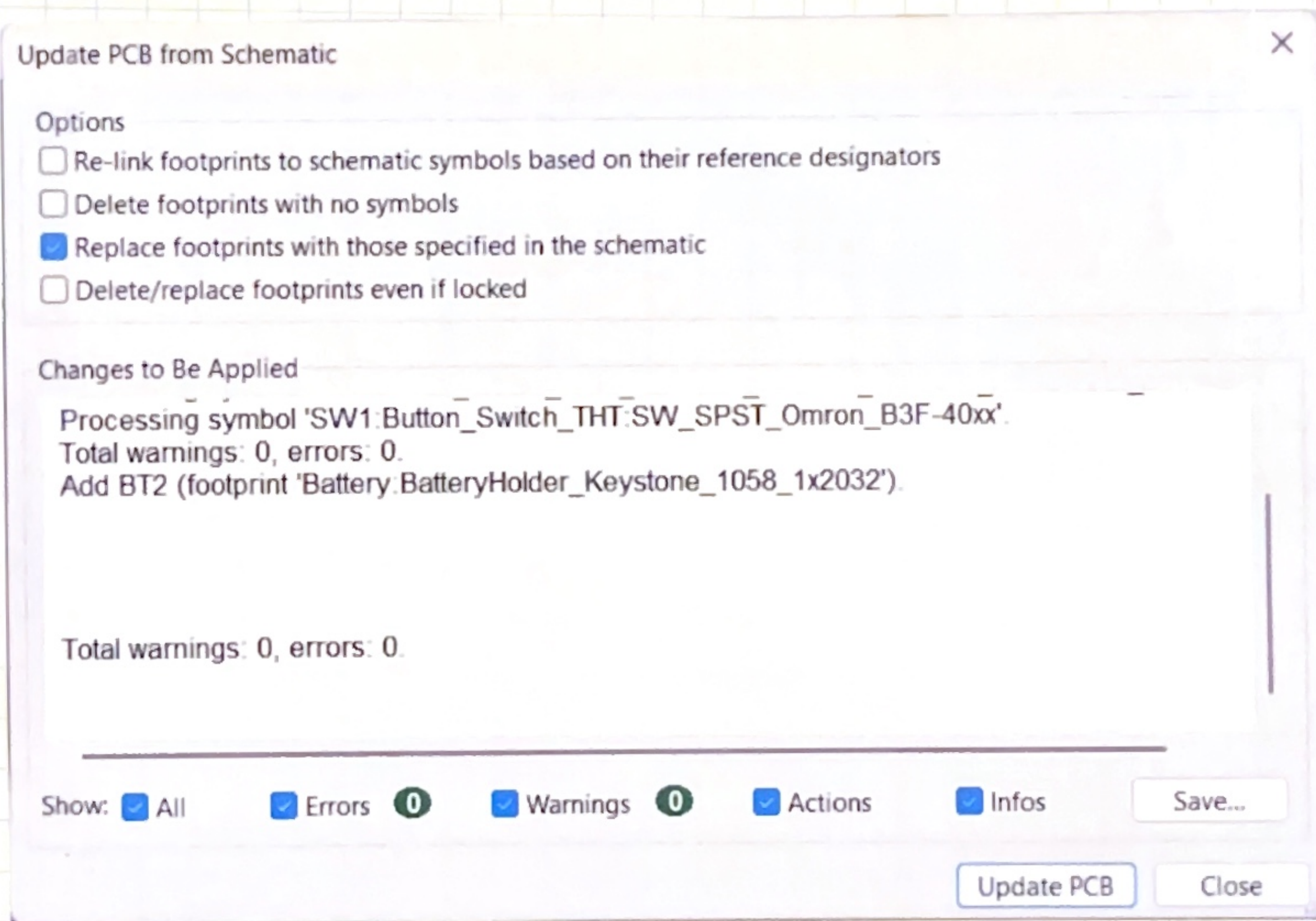
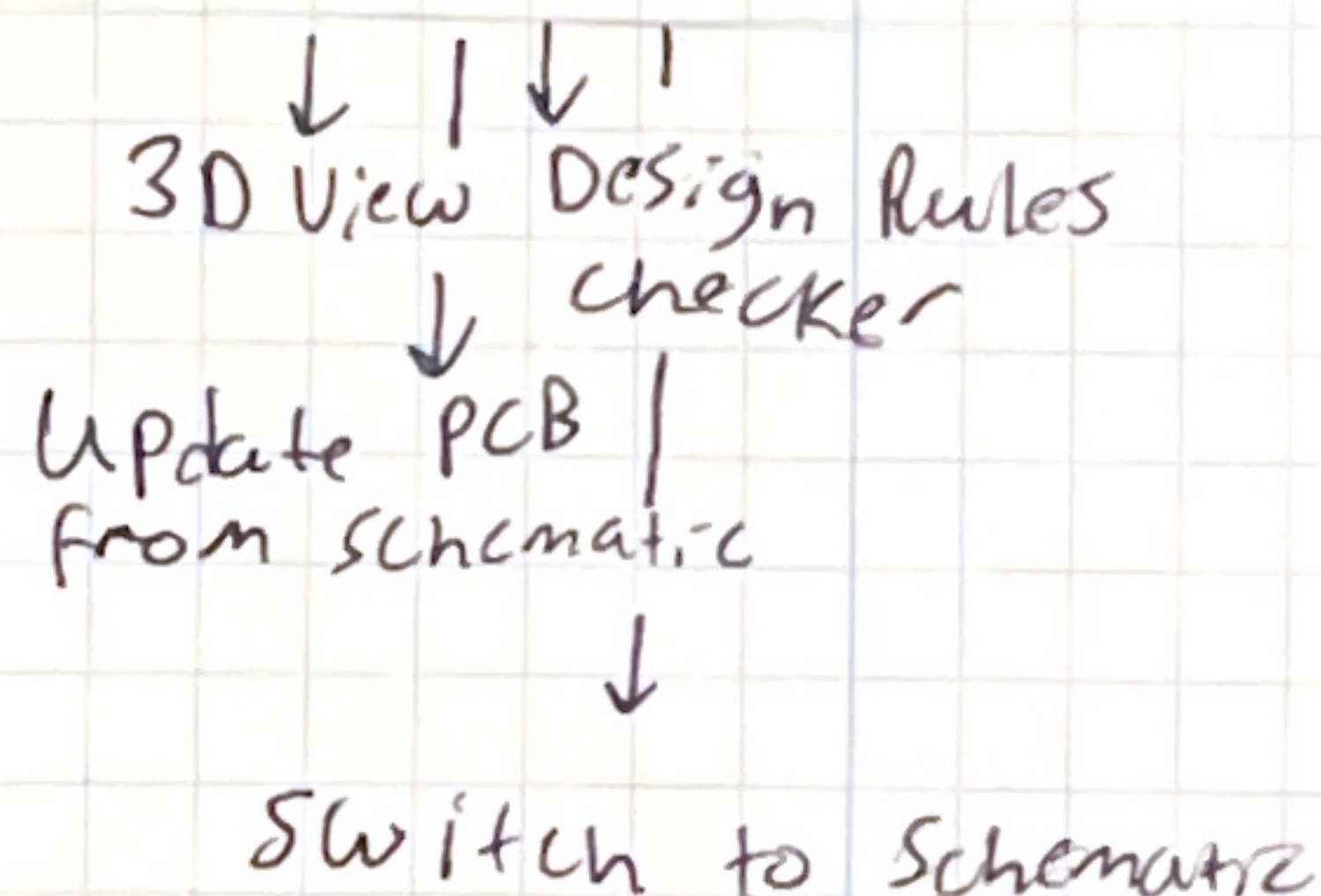
97

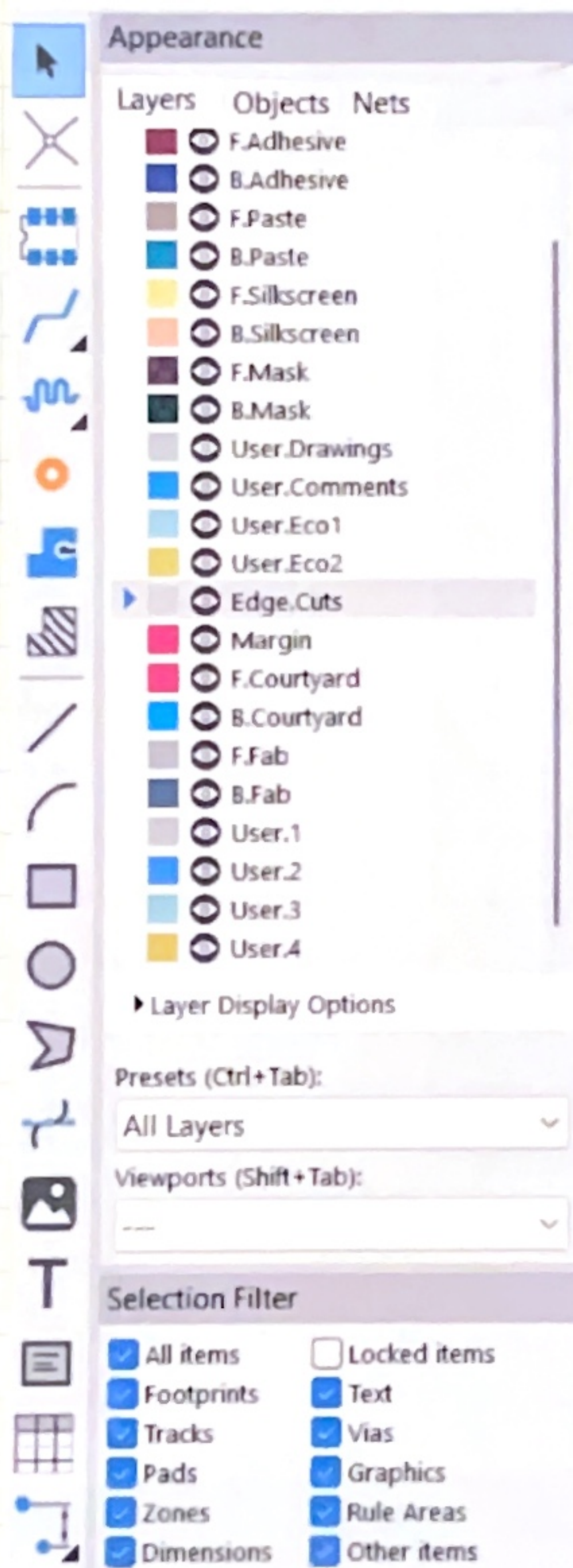
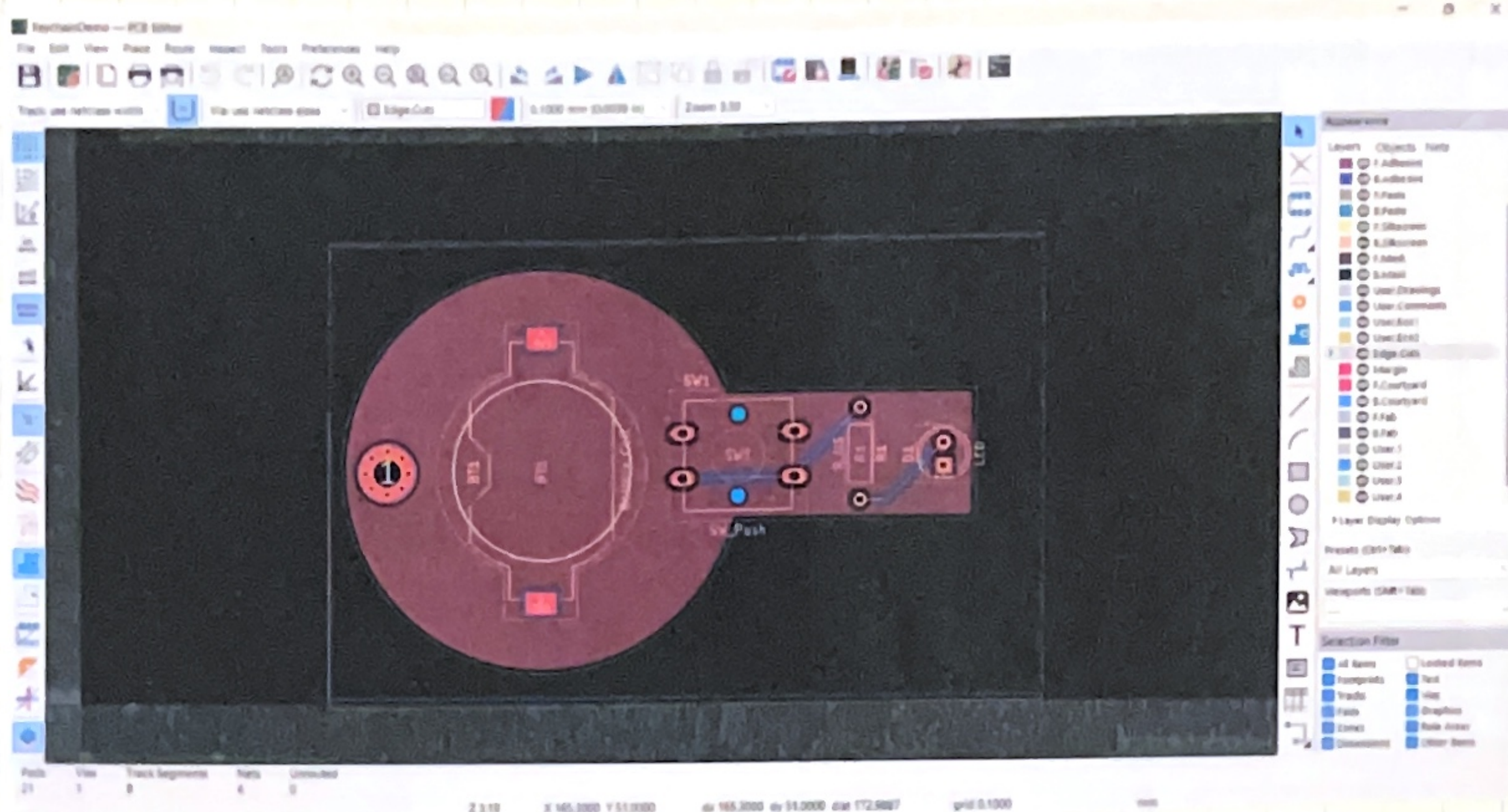
PCB Editor:



After switching to the PCB editor, it might be blank.

Always click update PCB from schematic after making changes.





→ ~~F.Cu~~, front copper
↳ wires go here.

→ F. Silkscreen, Front Text

→ Edgecuts used for board outline

