

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
 - ↳ Footprint is the layout the symbol takes up on the PCB

If a symbol or footprint doesn't exist:

use the datasheet and the symbol editor/footprint editor to create your own.

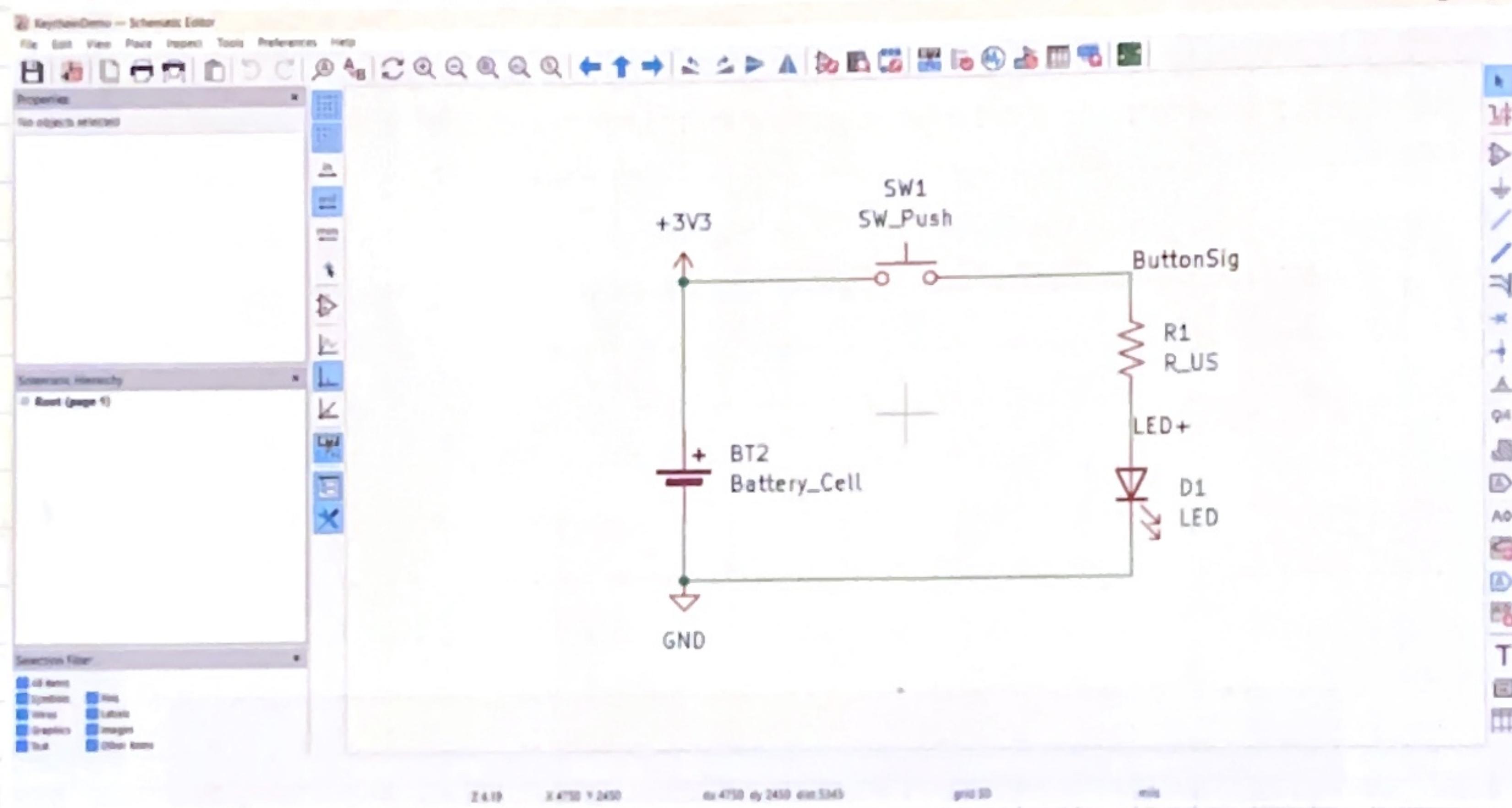
PCB Editor:

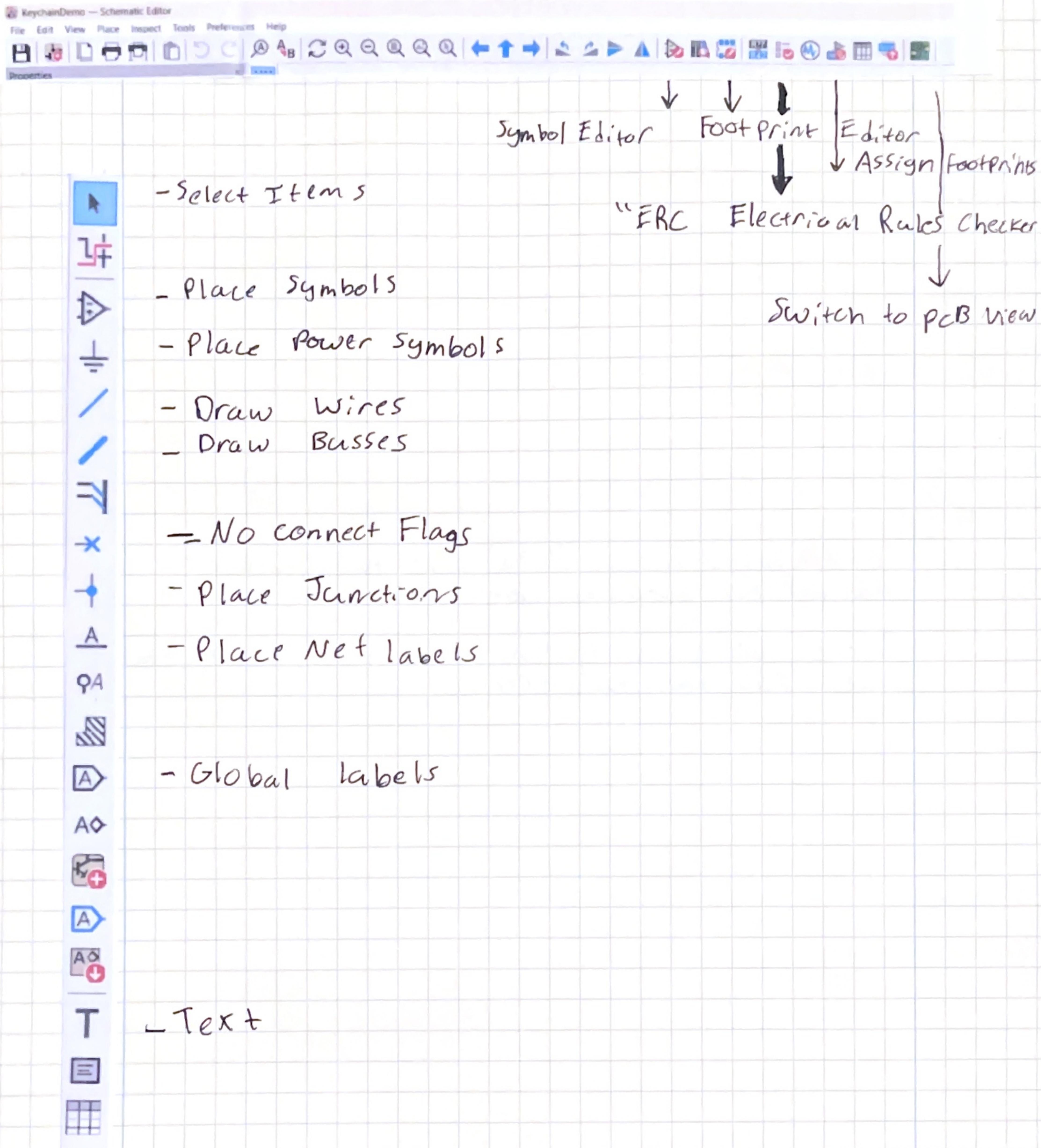
- ↳ update components from schematic
- ↳ "Edge Cuts" layer for the board outline
- ↳ Layout footprints
- ↳ Route traces on front or back copper layer.
- ↳ Power and Ground Planes or "Filled Zones" (optional)
- ↳ 3D Model Viewer
- ↳ Export Gerbers

06/09/25
Wesley
coke



Schematic Editor:





KICAD Workflows

06/09/25
Wesley
coche

Recent Changes (34699 items listed)	
Item	Description
BatteryHolder_Eagle_128H611-GR	http://eu.mouser.com/datasheet/2/208/EPD-200766-1274481.pdf
BatteryHolder_Keystone_103_1x20mm	https://www.keyleco.com/product-pdf.cfm?p=719
BatteryHolder_Keystone_104_1x23mm	https://www.keyleco.com/product-pdf.cfm?p=744
BatteryHolder_Keystone_105_1x24mm	https://www.keyleco.com/product-pdf.cfm?p=745
BatteryHolder_Keystone_106_1x20mm	https://www.keyleco.com/product-pdf.cfm?p=720
BatteryHolder_Keystone_107_1x23mm	https://www.keyleco.com/product-pdf.cfm?p=746
BatteryHolder_Keystone_500	Keystone #500, 12mm CR1220 battery holder. https://www.keyleco.com/product-pdf.cfm?p=710
BatteryHolder_Keystone_590	AA (R6) A battery holder with leaf spring. Keystone. https://www.keyleco.eu/cat/590.pdf
BatteryHolder_Keystone_1042_1x18650	Battery holder, Keystone 1042, battery type: 1x18650 (5...y) (https://www.keyleco.com/product.cfm/product_id/910)
BatteryHolder_Keystone_1057_1x2032	SMT (Auto-In/Ultra-Low) Holder for 2032 Cell, 31.2 x 2...PCB. https://www.keyleco.com/userAssets/file/M6Sp4.pdf
BatteryHolder_Keystone_1058_1x2032	https://www.keyleco.com/product-pdf.cfm?p=14020

The screenshot shows the 'Assign Footprints' application window. The menu bar includes File, Edit, Preferences, Help, and a toolbar with various icons. The main area has three tabs: 'Footprint Libraries', 'Symbol-Footprint Assignments', and 'Filtered Footprints'. The 'Symbol-Footprint Assignments' tab displays a table with four columns: Row, Symbol, Footprint, and Description. The 'Filtered Footprints' tab lists several footprint names.

	Symbol	Footprint	Description
1	BT2	Battery_Oval	Battery:BatteryHolder_Keystone_1000_1x2032
2	DL	LED	LED:TH1715LED_05.0mm_Clear
3	R_UF	R_UF	Resistor_TH18_R_Axial_050207_14.2mm_Pt2.5mm_Pt0.16mm_Perpendicular
4	SW_Push	SW_Push	Button_Switch_TS71SW_EPST_Ouron_837-60mm

Footprint
1 Audio_Module_Beacon_BT229-1H
2 Audio_Module_Beacon_BT229-1V
3 Battery:BatteryClip_Keystones_34_016-1mm
4 Battery:BatteryHolder_Bulgin_BT0034_1mm
5 Battery:BatteryHolder_Bulgin_BT0034_1ml
6 Battery:BatteryHolder_Bulgin_BT0121_1mCR123
7 Battery:BatteryHolder_ComfortableLithium_03
8 Battery:BatteryHolder_Eagle_1288611-00
9 Battery:BatteryHolder_Keystones_103_1x10mm
10 Battery:BatteryHolder_Keystones_104_1x12mm
11 Battery:BatteryHolder_Keystones_105_1x15mm

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

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

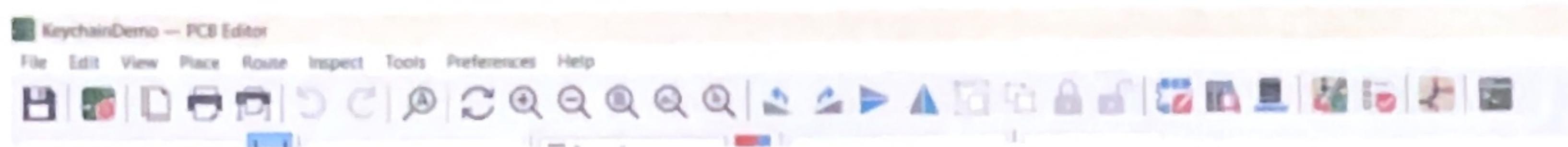
The screenshot shows the Electrical Rules Checker (ERC) interface. At the top left, it says "Violations (0)" and "Ignored Tests (4)". Below this is a large empty rectangular area representing the workspace. At the bottom, there are several controls: "Show:" dropdown with "All" selected, "Errors" checked (0 violations), "Warnings" checked (0 violations), "Exclusions" unchecked, "Save..." button, "Delete Marker" button, "Delete All Markers" button, "Run ERC" button, and "Close" button.

06/09/25
Wesley
code

KICAD Workflows

97

PCB Editor:

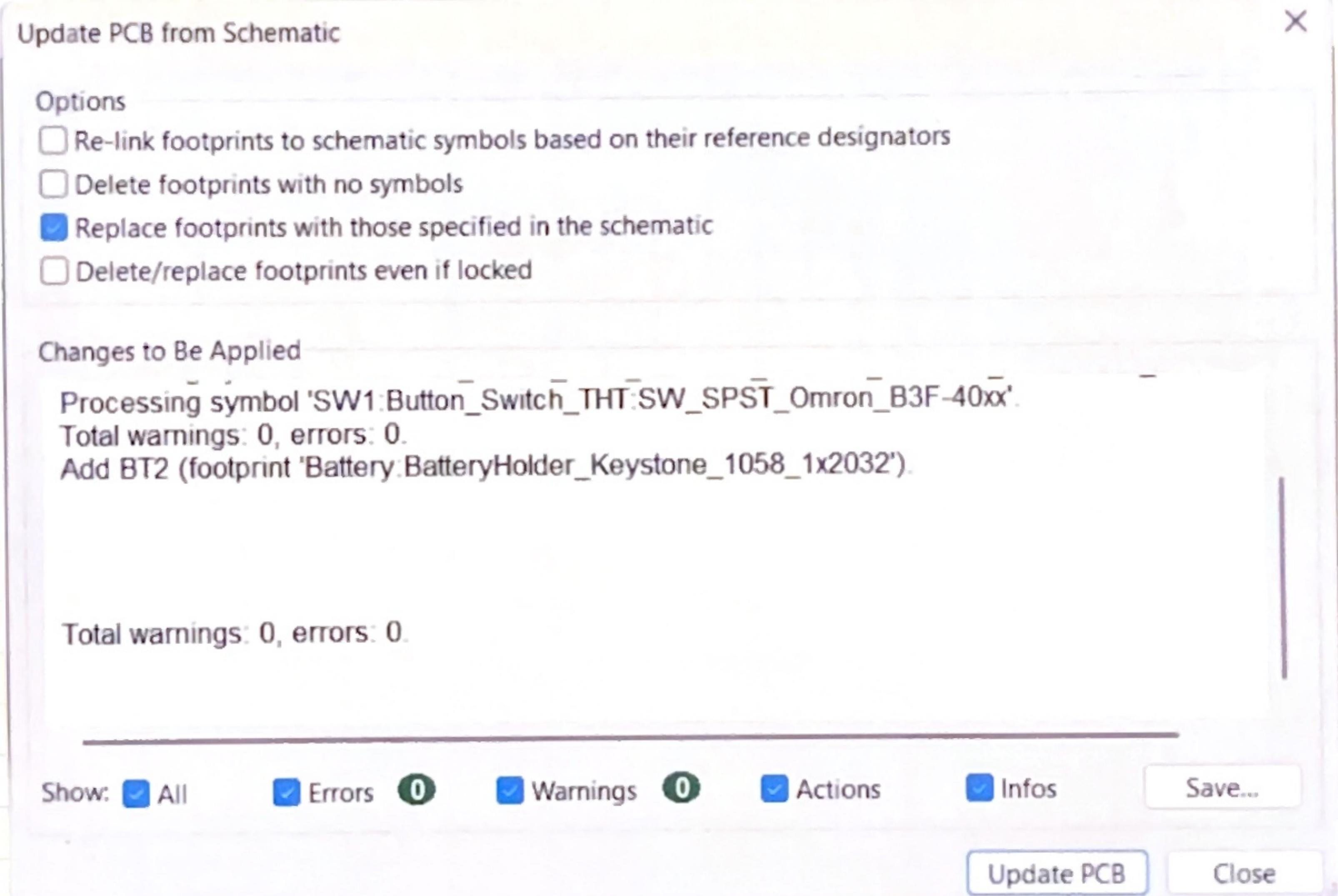


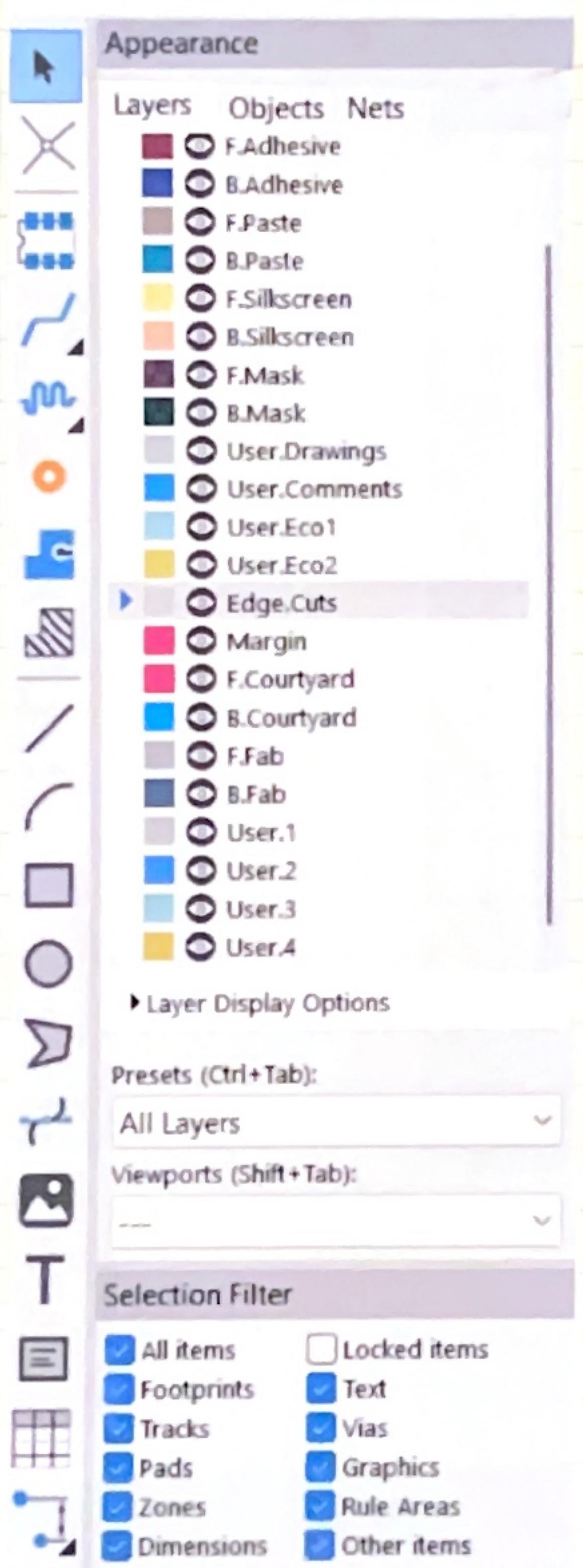
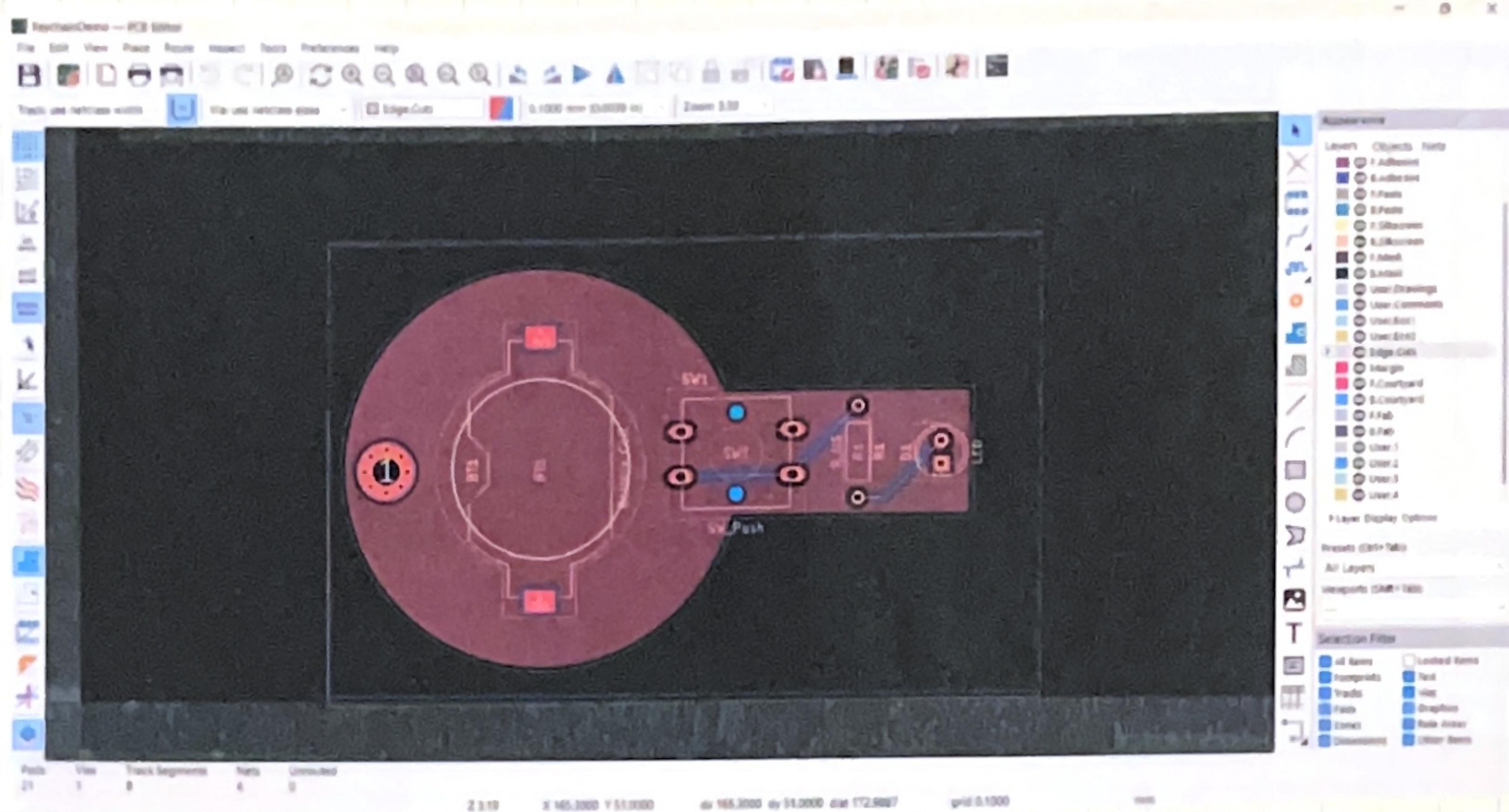
After switching to the PCB editor, it might be blank.

Always click update PCB from Schematic after making changes.

↓ | ↓
3D View Design Rules
↓ checker
Update PCB |
from schematic
↓

Switch to Schematic





→ F.Cu, front copper
↳ wires go here.

→ F. SILK screen, Front Text

→ Edgecuts used for board outline

