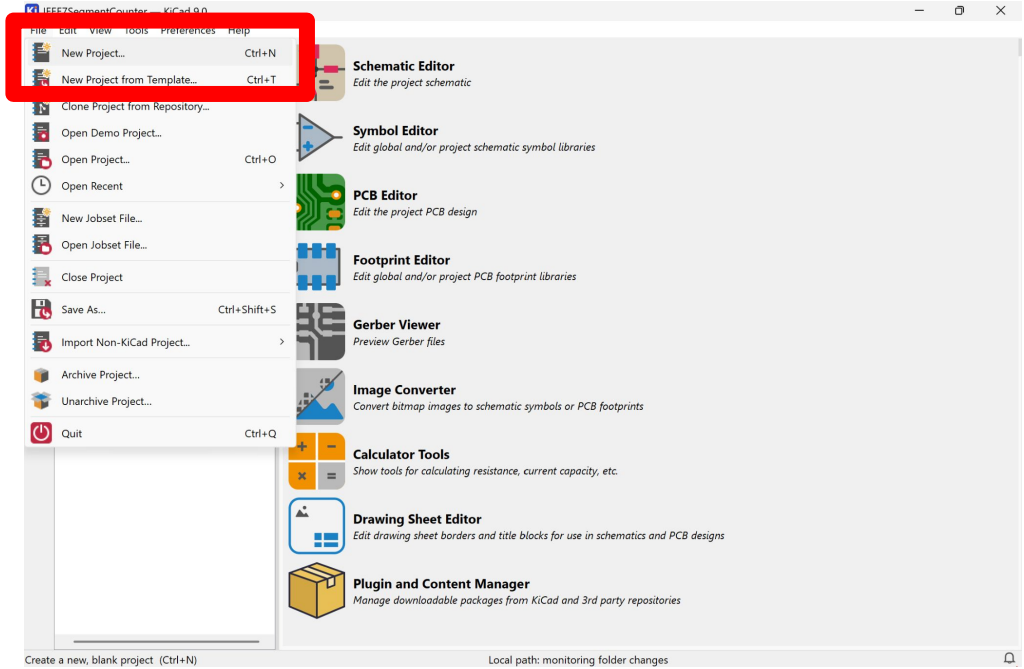
KiCad - Fusion 360 Workshop

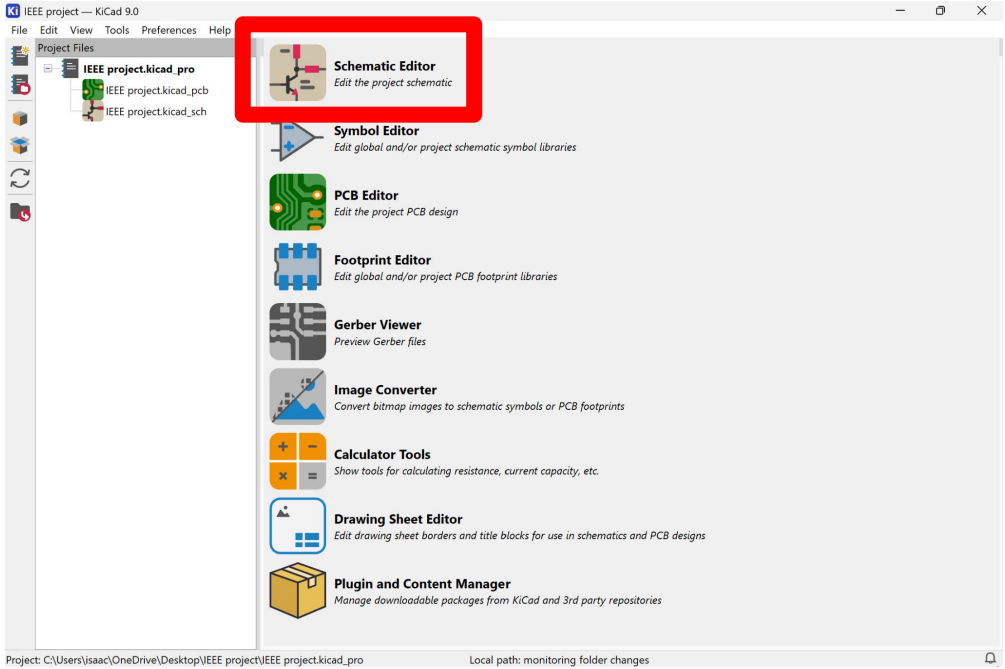
Go to https://github.com/psuieee for completed files

KiCad - Create a project

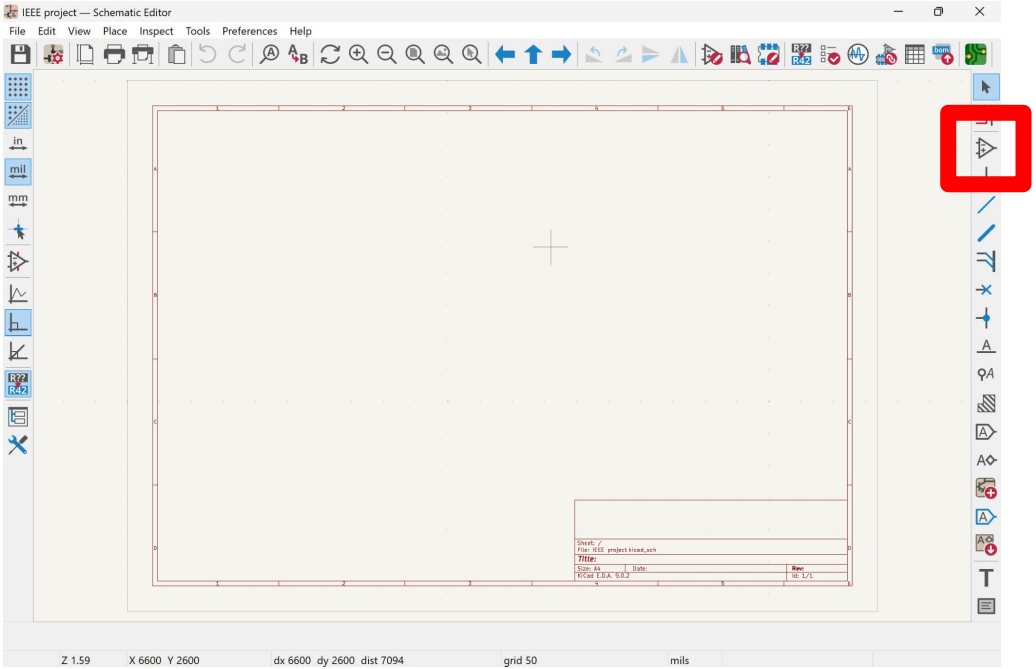


Open Schematic Editor

KiCad - Open Schematic Editor

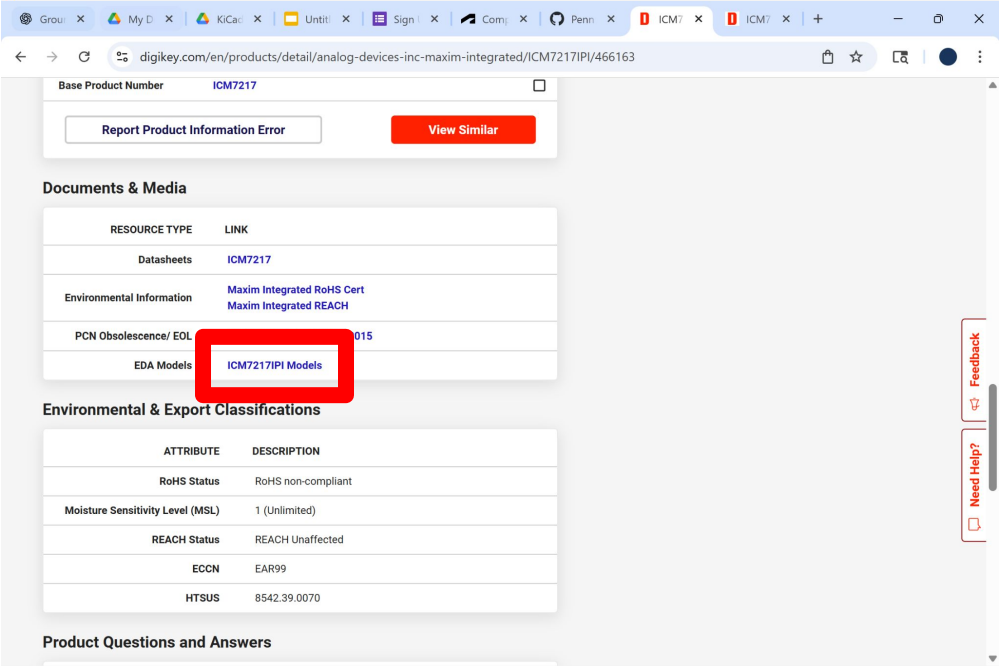


KiCad - Place Symbols



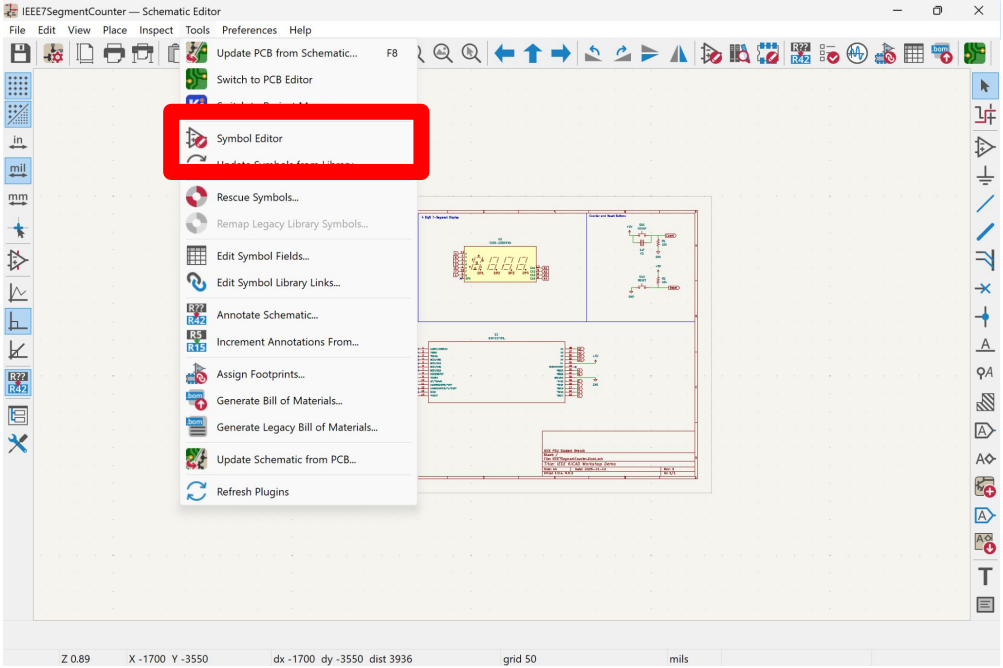
Shortcut (A)

KiCad - Import Symbol



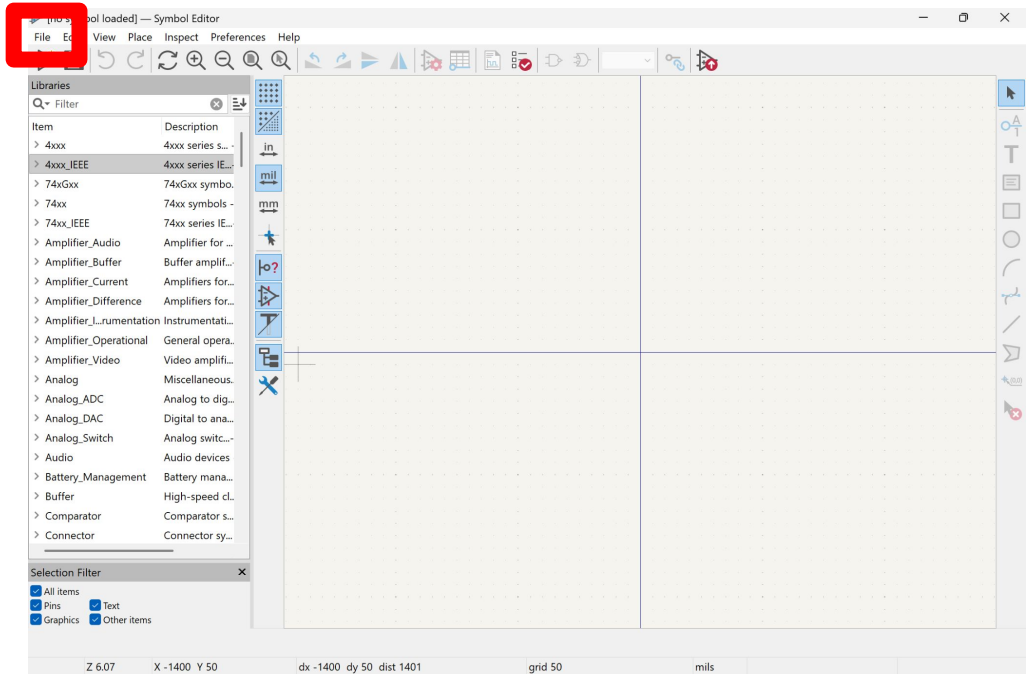
Click Here

KiCad - Import Schematic



Tools ➡ Symbol Editor

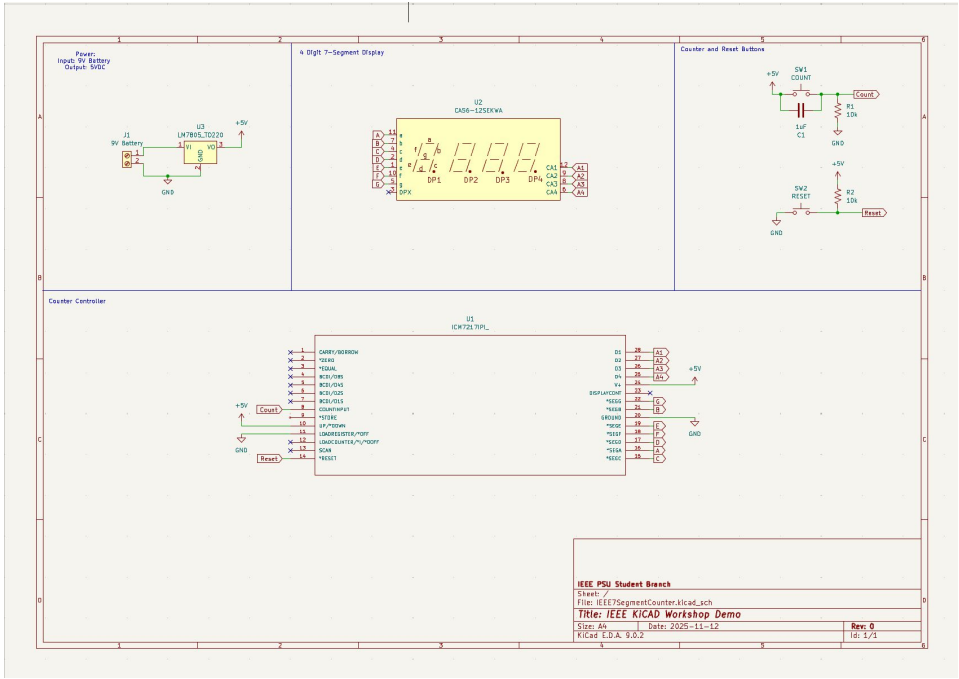
KiCad - Import Symbol



Create a new library to store the symbol

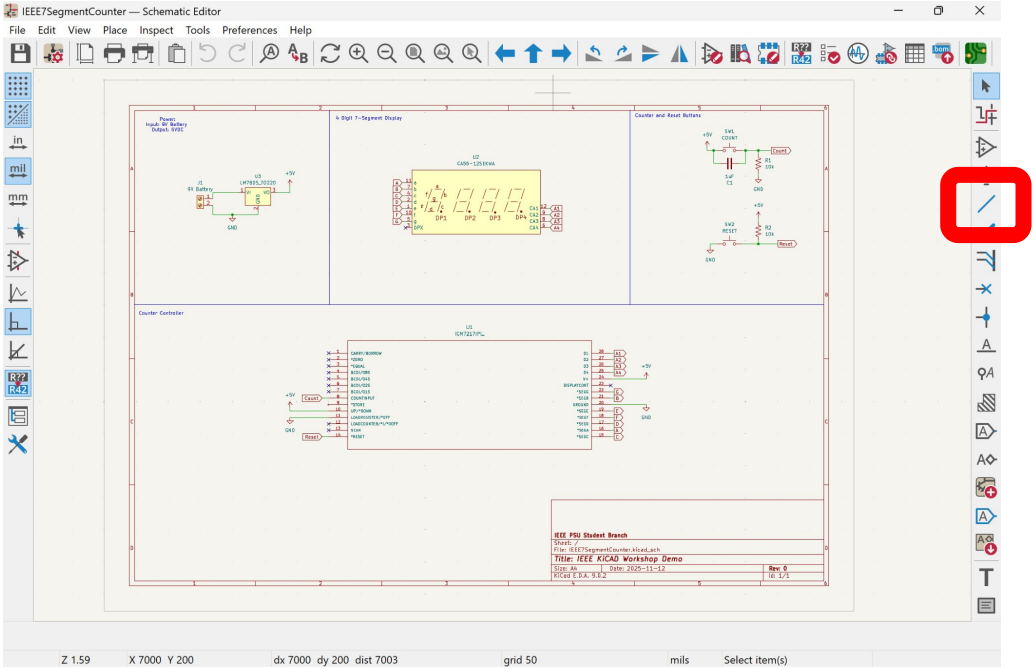
KiCad - Required Symbols (Exact names)

- +5V
- C
- CA56-12SEKWA
- GND
- ICM7217IPI_
- LM7805_TO220
- R_US
- SW_Push
- Screw_Terminal_01x02



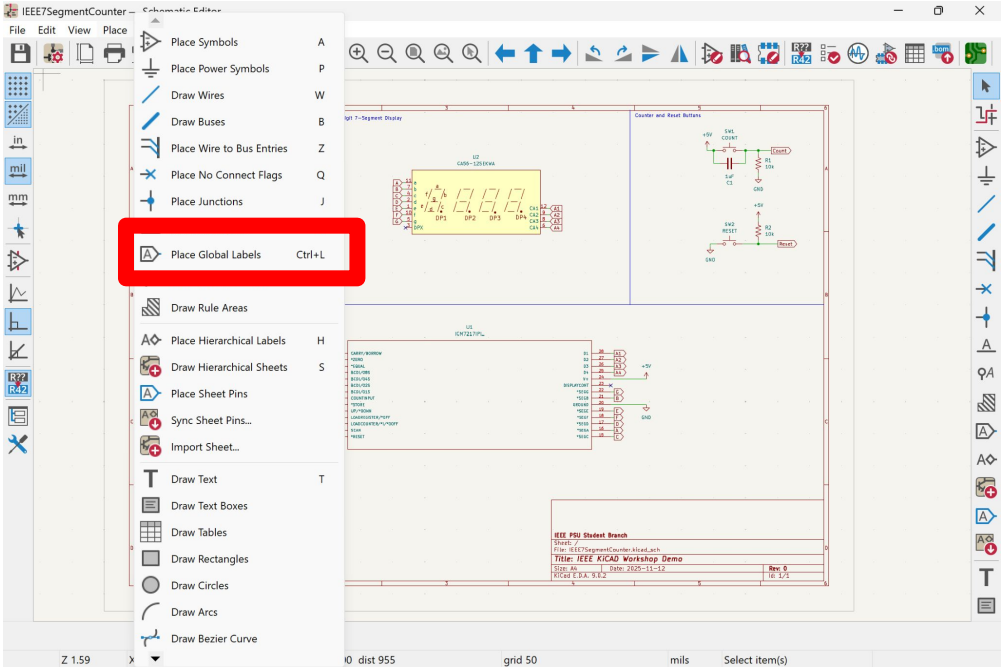
Shortcut (You can copy and paste footprints)

KiCad - Placing Wires



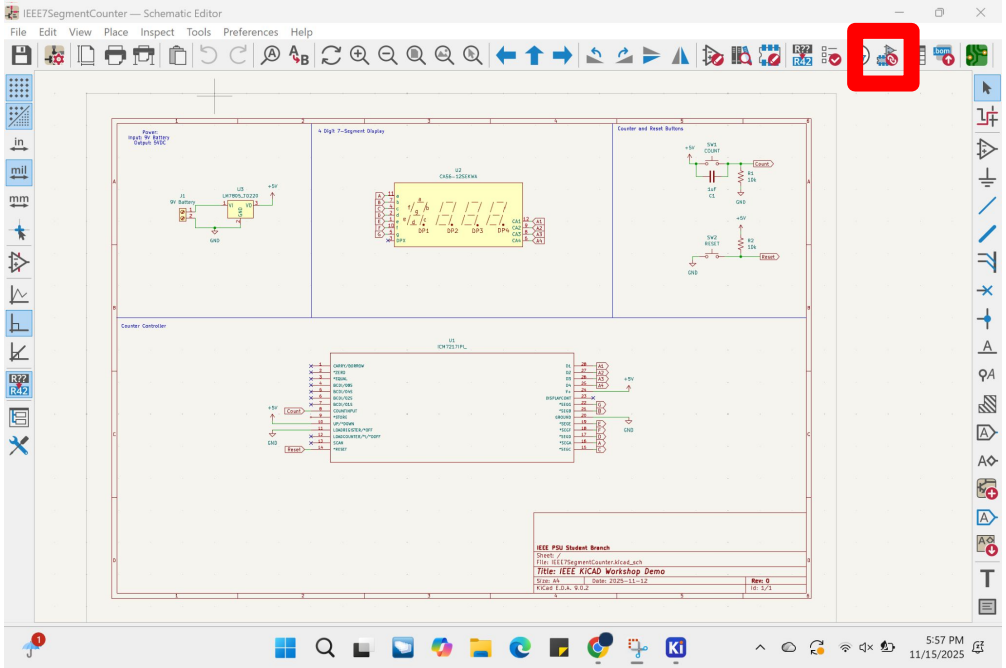
Shortcut (W)

KiCad - Placing Global Connectors

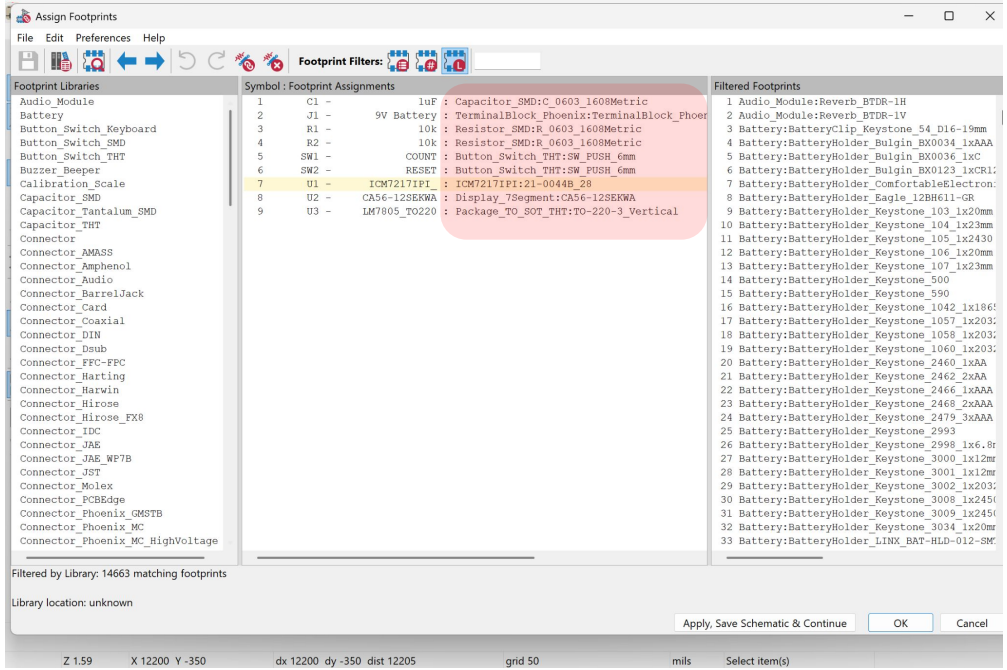


Shortcut (Ctrl + L)

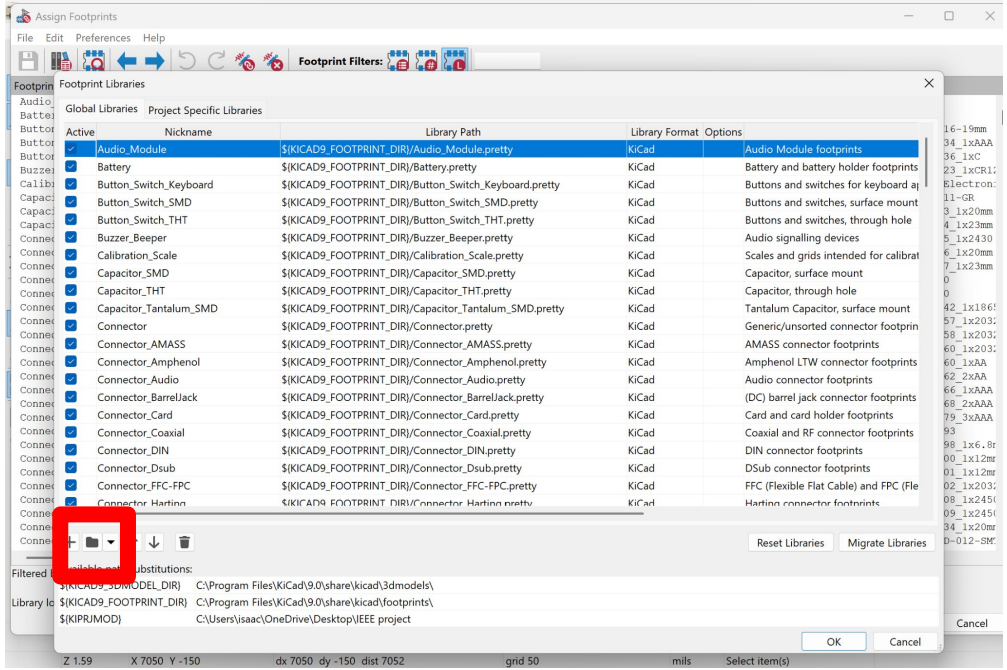
KiCad - Adding Footprints



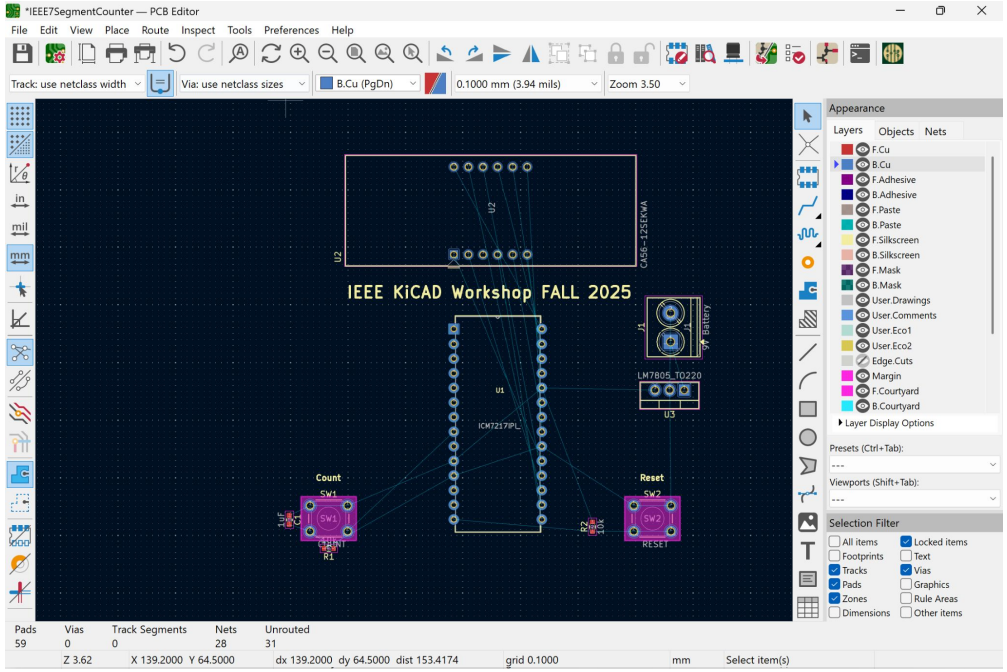
KiCad - Needed Footprints



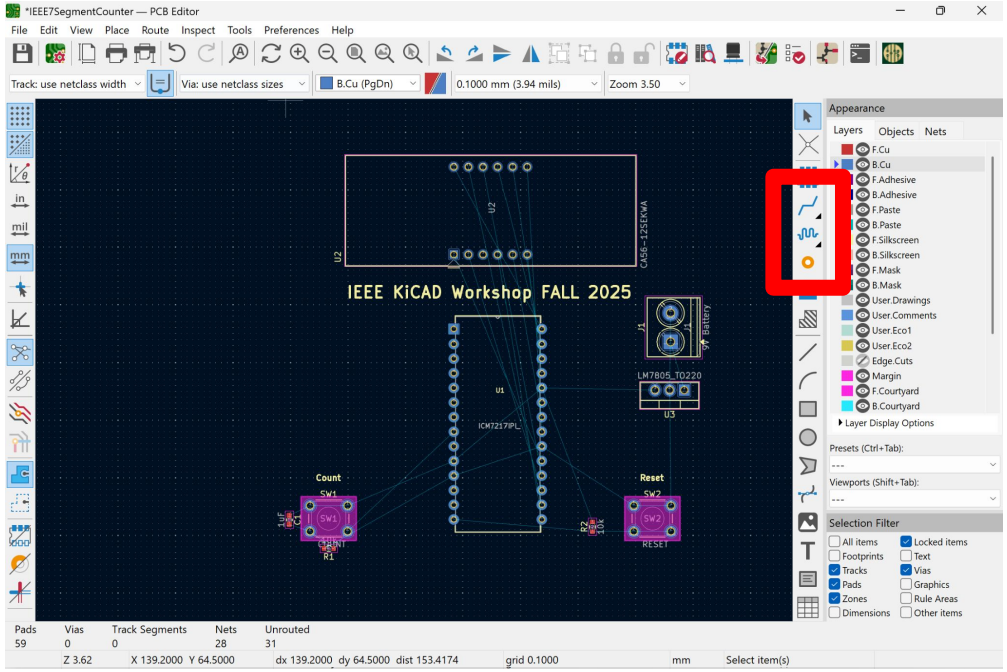
KiCad - Import Footprint



KiCad - Place Footprints

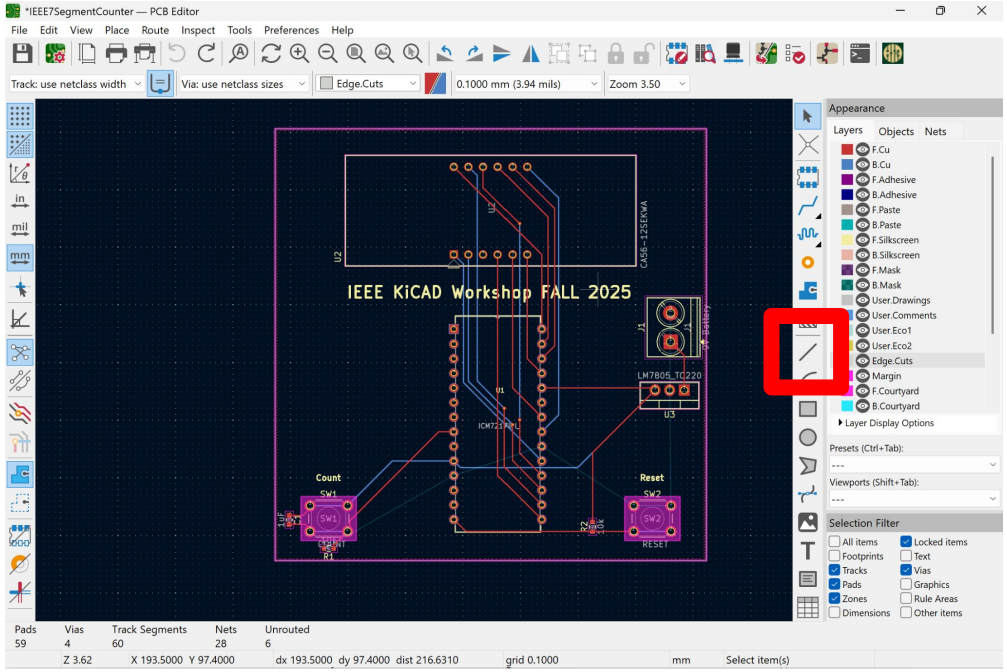


KiCad - Route Tracks & Vias



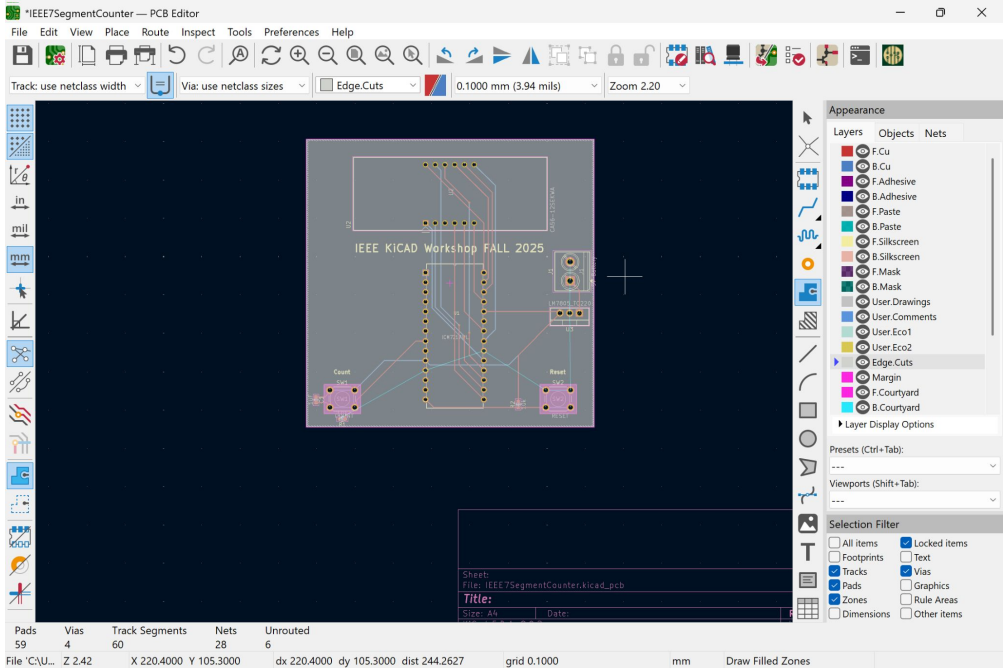
Shortcut (X)

KiCad - PCB outline



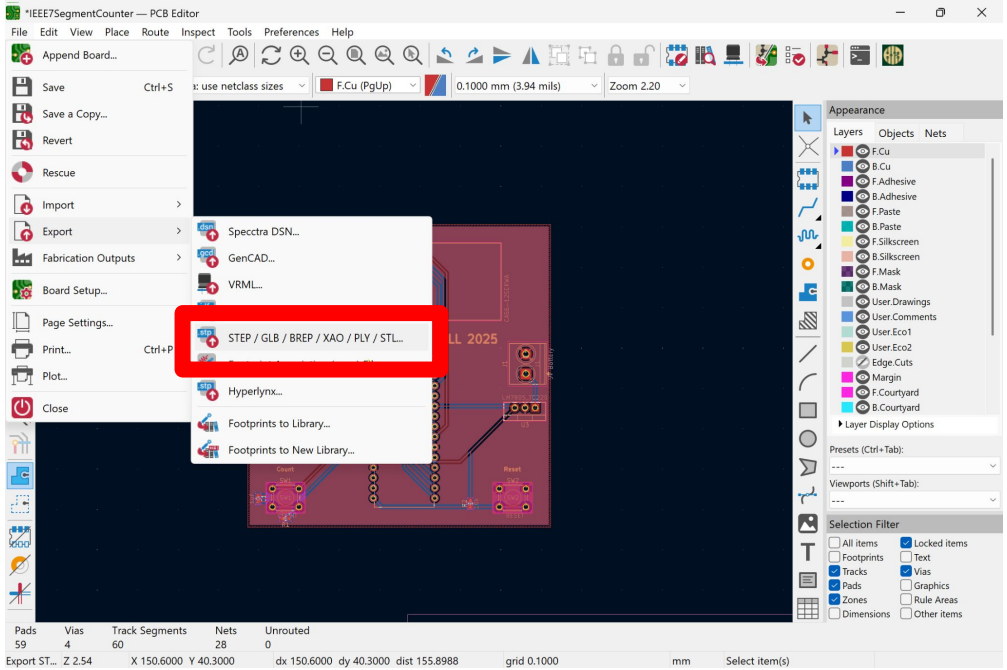
Requires Edge Cut selected under Appearance

KiCad - Ground Plane



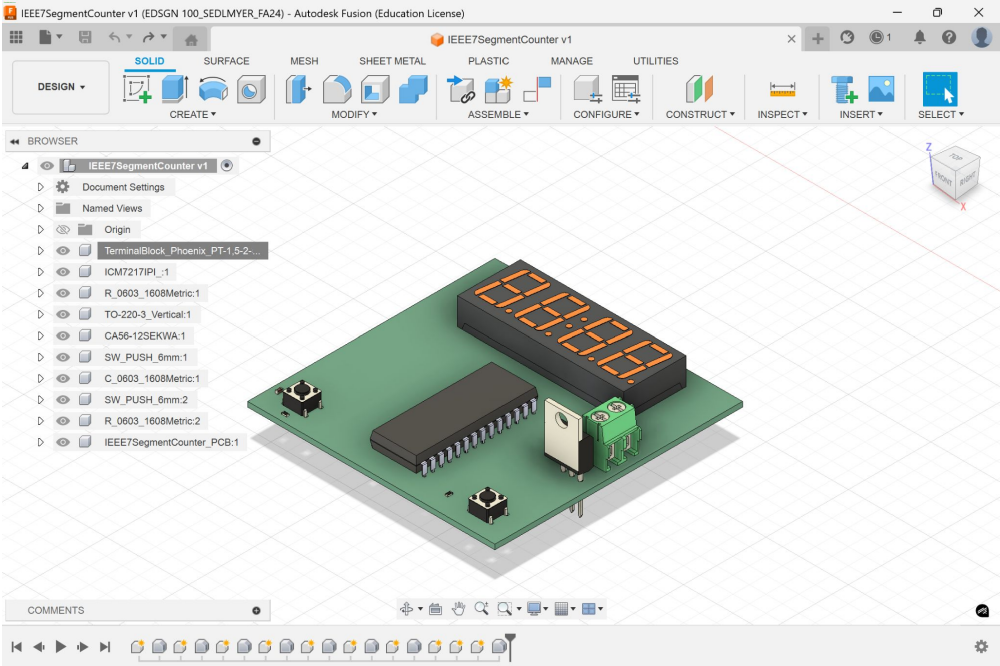
Outline the PCB & Press B to Fill

KiCad - Exporting the File

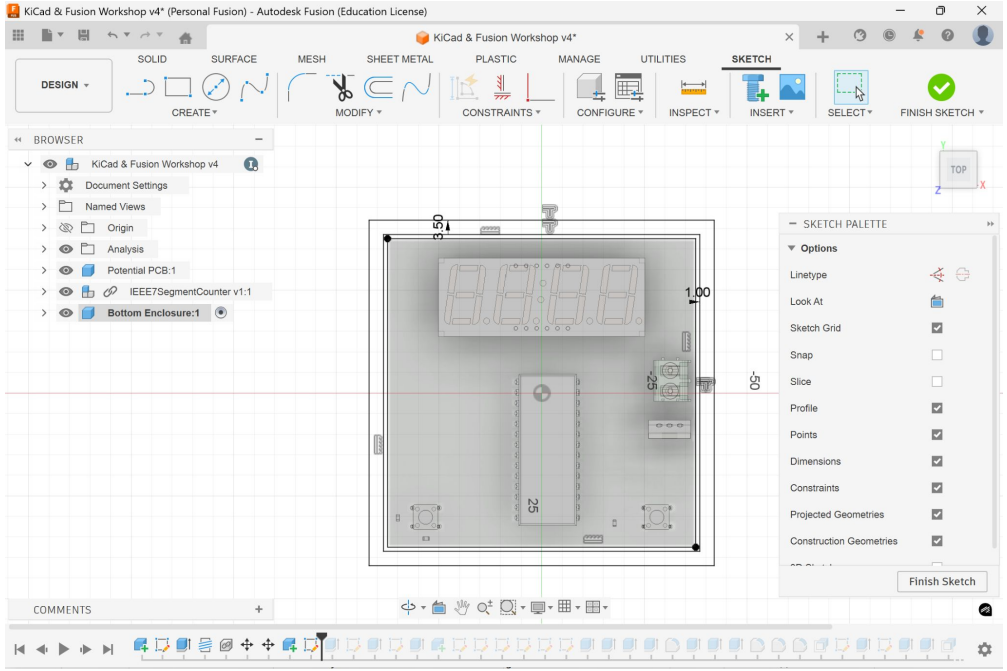


Export it as an STL

Fusion 360 - Open the STL

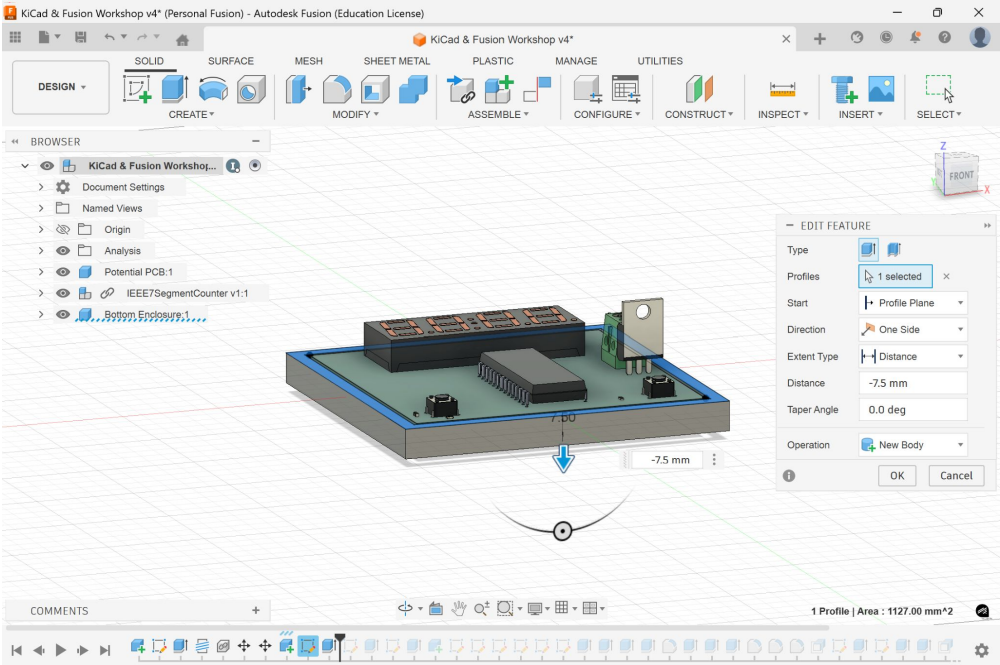


Fusion 360 - Create a sketch



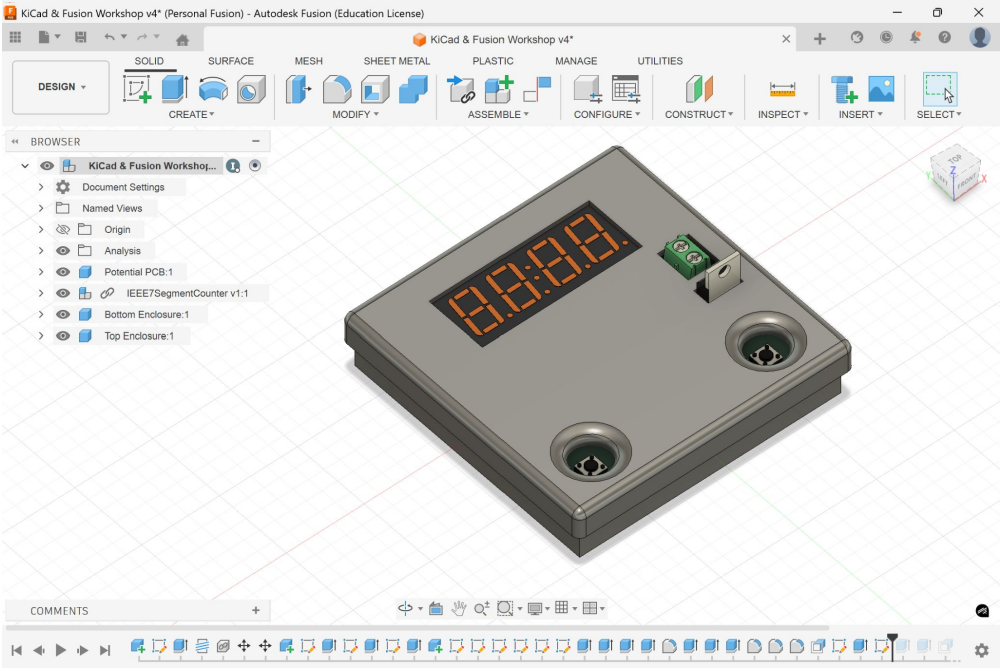
Dimension Shortcut (D)

Fusion 360 - Extrude Base



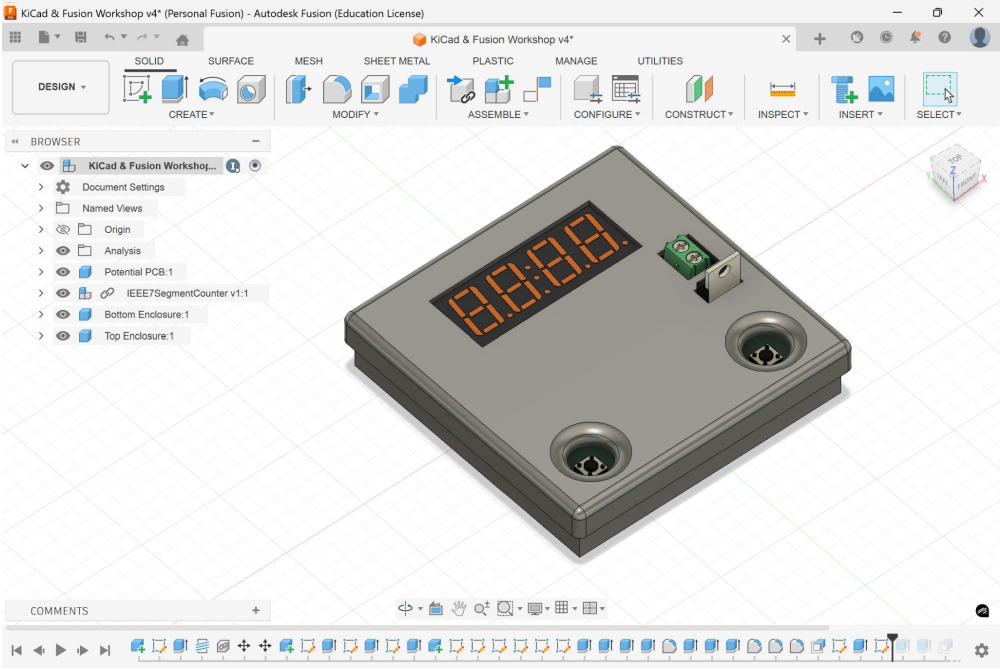
Shortcut (E)

Fusion 360 - Continue to design Case



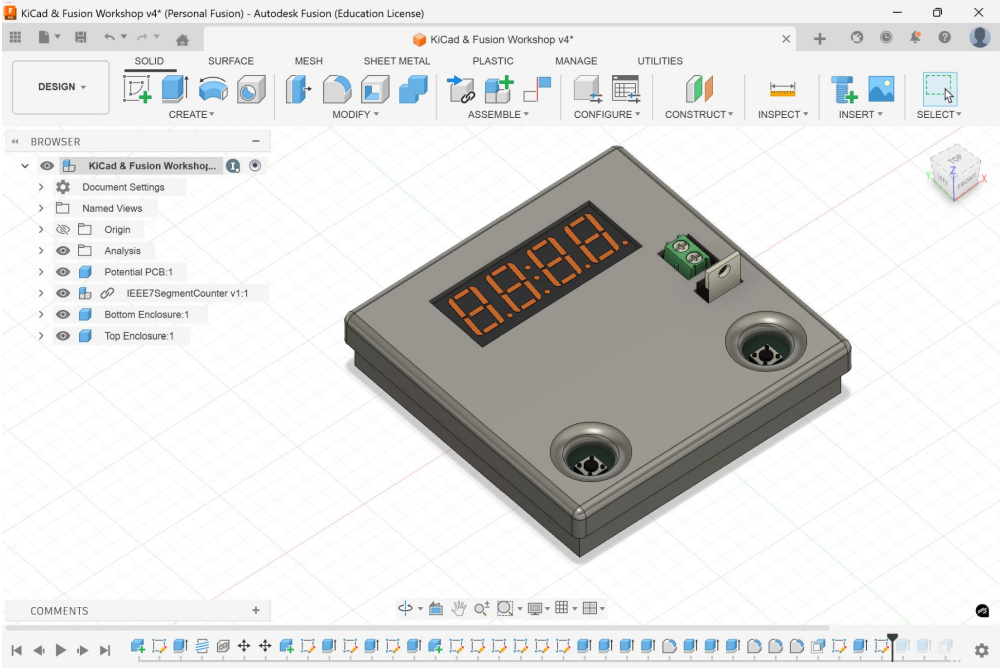
Use the completed version as a reference

Fusion 360 - Continue to design Case



Fillet Shortcut (F) - Use the completed version as a reference

Fusion 360 - Continue to design Case



Fillet Shortcut (F) - Use the completed version as a reference