PCB Practice Project:

1. Extract the zip file
2. Open the KiCad Project file
   1. Purple square
3. Schematic Editor:
   1. Make sure libraries and footprints are imported!
      1. Make sure you see the VB-PS-Adapter
      2. Right click on components, click properties, and make sure each has a footprint
   2. Run a design rules check
      1. Write down the warning so I know you did it right
      2. You do not have to fix it.
   3. Press the green PCB button
      1. labeled open PCB in board editor
4. PCB Editor
   1. Press the button labeled “update PCB with changes made to schematic”
      1. A pixelated image of a green and black cartoon character

         Description automatically generated
   2. Place origin
   3. Draw edge cuts for schematic.
   4. Orient footprints.
   5. Route traces
   6. Check design rules.
   7. Export Gerber files
5. Other