



generate

Intro to Kicad Designer - SPRING 2023



PREFACE

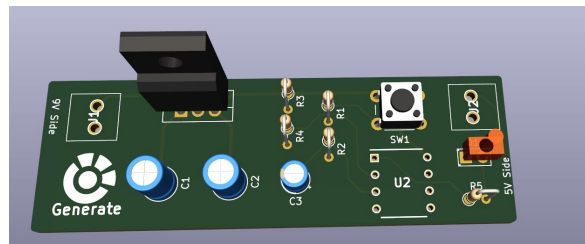
The goal of this presentation is to help you understand the ECAD tool Kicad and will provide insight into the following

- Creating a full schematic in KiCad
- Importing parts from mouser/Digikey and Octopart
- Brief Insight into how PCB design is performed
- This workshop serves as a “part 1” where later we will turn the schematic we design in this part into an actual PCBA (printed circuit board assembly).



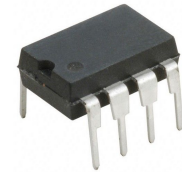
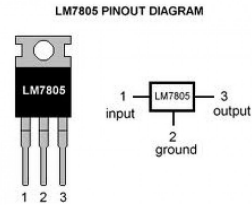
WHAT IS KiCad?

- KiCad is an industry standard ECAD software that allows an engineer to design schematics, PCB boards, as well as board stackups.
 - Similar to ORCAD, Altium, or Free alternatives Eagle, Easy EDA
- Allows you to design schematics and then import those parts into a pcb document to then design and export to fab house to bring your design to life

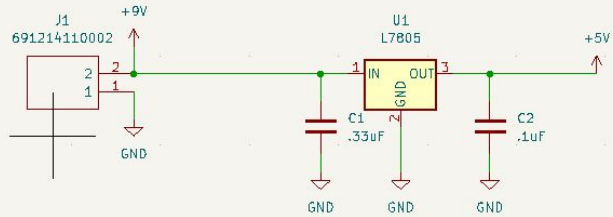


LET'S DESIGN A SIMPLE SCHEMATIC IN KiCad!

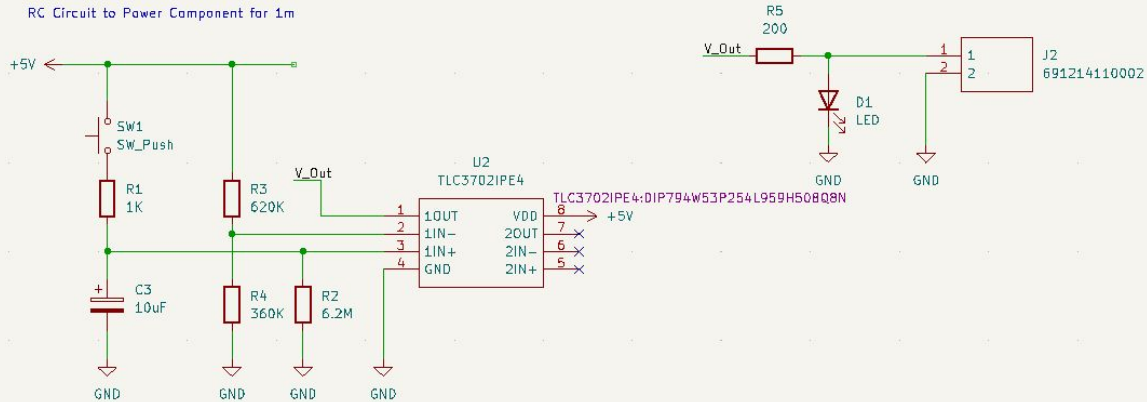
- We will be using a student license of KiCad to build a circuit that takes a 6-9V battery input, regulates it down to 5V, and the powers on anything for one minute at a time
- Important components include
 - TLC3702 comparator IC
 - L7805 Voltage Regulator
 - Terminals



Voltage Regulator



RC Circuit to Power Component for 1m



Schematic Files

-



IMPORTANT FILES:

PCBA and Schematic Libraries

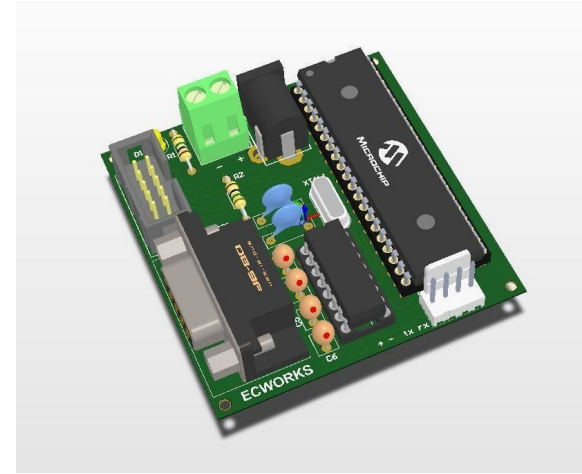
- Schematic Libraries are where the symbols as well as some PCB footprint data for components in your project are stored.
 - Here you can find important information such as the schematic symbol, price, supplier link, and other useful information
- PCB Libraries are where PCB footprints can be created and stored using a footprint wizard or by importing them
 - Sometimes when importing a component, the footprint will not transfer. You are able to load footprints then from this library.



IMPORTANT FILES:

PCB Files

- PCB files are where actual board layout occurs and where routing between parts occurs
 - Routing describes the practice by which “traces” are placed down on a board to form electronic connections
- There are plenty of good practices for PCBA routing, but that is for the next workshop!

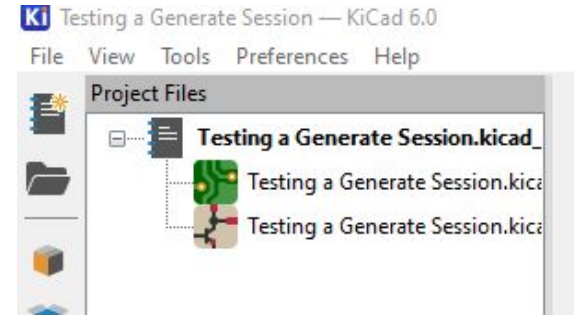


Open KiCad!



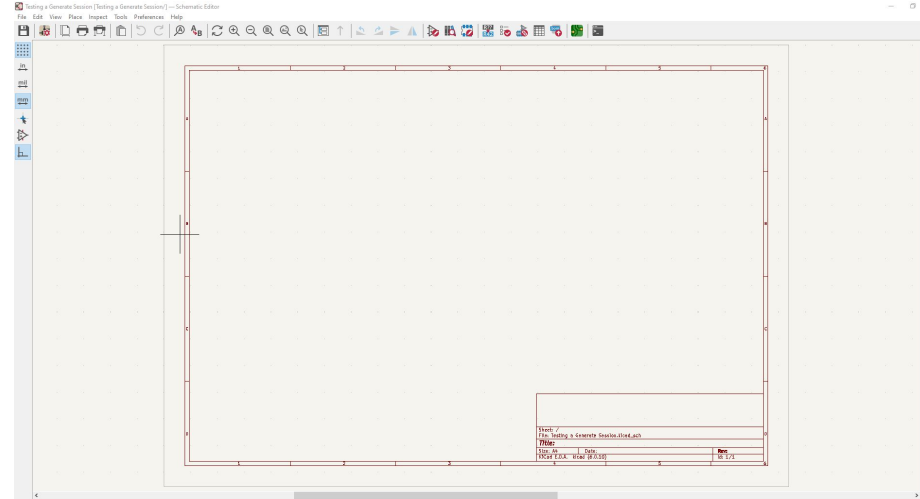
HOW TO START A PROJECT

- Upon opening up KiCad, select file in the top right and select new project and name it something like “New Project”
 - I recommend first making a KiCad folder with the following Subfolders
 - Projects
 - Imported Parts
- This will create a new Kicad Project that will have both a kicad.sch (schematic file) and a kicad.pcb file (pcb file)



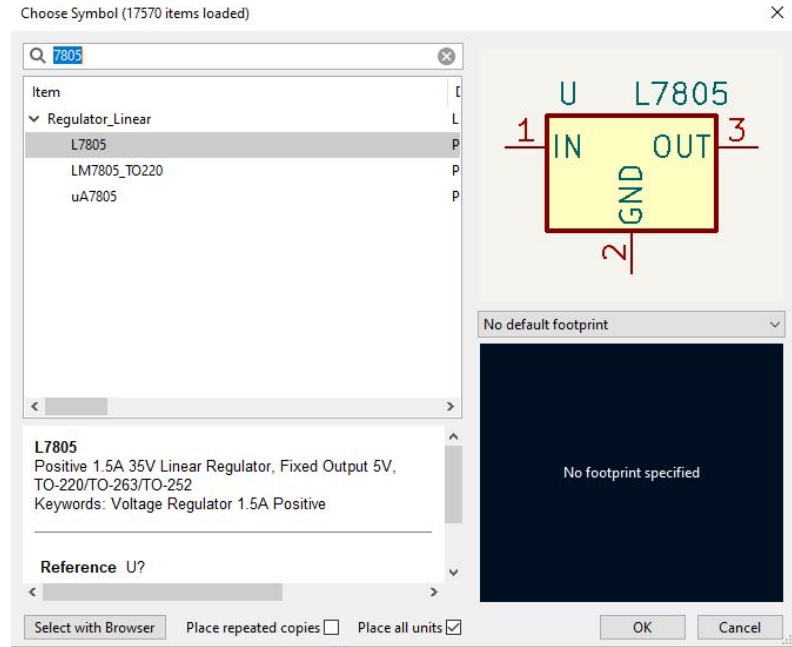
HOW TO START A PROJECT

- The first step to building a PCB is to design the corresponding schematic first. Select the schematic file and we should be ready to start creating our schematic!

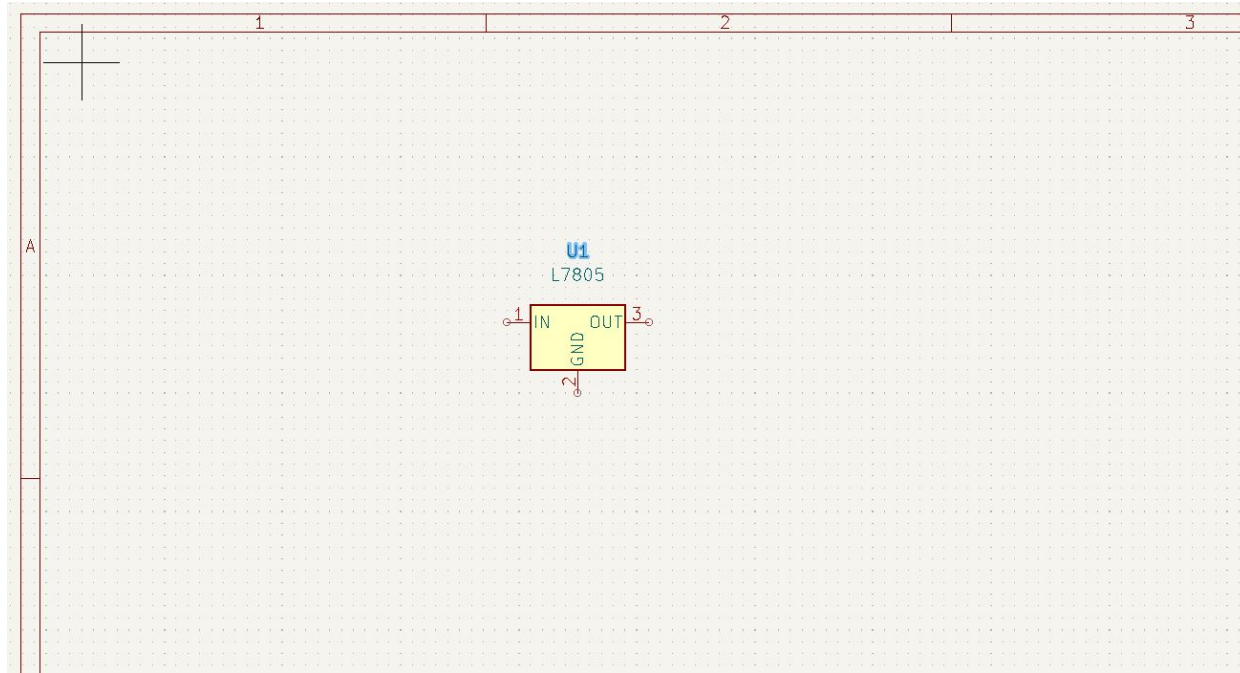


LAYING OUT IMPORTANT COMPONENTS

- The first components I like to layout when creating components are ones I know I am going to import.
- These will be the TLC3702, 7805 Regulator, and the Terminals we will use
 - We can press the hotkey “A” to place a component
 - Sometimes KiCad will have some components in their default library
- Always select a footprint package when placing a component



LAYING OUT IMPORTANT COMPONENTS



LAYING OUT IMPORTANT COMPONENTS

- Some components like the terminals and the TLC3702 compactor cannot be found in the native symbol library so we will have to look for the footprint online and import it using the **SAMAC LIBRARY LOADER**
 - Mouser is a good website for this specifically
- lets search for the 2 Terminal Connector and the Comparator



LAYING OUT IMPORTANT COMPONENTS

- Select the ECAD Model and log into your SamacSys account to download the component
- the download will be a library .zip file. Extract the file into your exported part folder.
- To load this into our schematic, head to the preferences tab at the top of the screen and select manage symbol library



LAYING OUT IMPORTANT COMPONENTS

- Select the folder ICON and navigate the to the folder with the component. Navigate the extracted zip file until you see the Kicad File.
 - Select the .lib file
- It should now be imported if you look up the number of the component you should be able to place it



LAYING OUT IMPORTANT COMPONENTS

- When you importing a part it is **VERY** important you also import the footprint as well.
- Open the PCB file in your project and head to the preference tab and select manage footprint library
- Select the **DROP DOWN ARROW** on the folder Icon and select legacy kicad file.
 - Navigate to the folder the component was stored in and select the .mod file



