Open Builds Discord – Introduction to Eagle Schematic and PCB editor.

In this module I will introduce you to the basic process to create a schematic and PCB using the Eagle layout editor from Autodesk. Eagle is a mature software, with great integration into designs with Fusion360. The ECAD tool in Fusion360 is similar to Eagle stand alone, however, it is not mature yet and has some serious issues that need resolved.

Eagle is free, although for hobbyist the board size is limited to 80sq cm and 2 layers. If you are a student, it will remove this limitation. You can download Eagle from this weblink:
https://www.autodesk.com/products/eagle/free-download

Other things you may need to fully take advantage of this tutorial:

Autodesk Fusion 360

A copy of (insert link to datasheet for custom part)

The scope of this tutorial will help you to do the following:

1 – Create a schematic for a circuit using the schematic editor

- a- Using Eagle built in libraries
- b- Create a custom part library for your designs
- c- Import from the SamacSys online libraries

2- Create a circuit board from your schematic

- a- Component placement
- b- Routing the board
    - i- Autorouter
    - ii- By hand
- c- Design rule check
- d- Adding 3D component packages
- e- Pushing to Fusion360
- f- Creating file list for PCB fabrication

(working copy)