Using an Imported Library in EAGLE

The following procedure is done using Windows 10 OS and EAGLE 9.0.1. It may not apply directly to other operating systems or other EAGLE versions.

1.) Make a folder name *ENGR_301* in your computer. As an example, create the folder inside *C:\Users\santo\Documents\EAGLE*.

2.) Download the *EAGLE_misc* from iLearn and save it to your computer, e.g. the */Downloads* folder. Unzip the file. *EAGLE_misc* contains the additional components (.lib), the design rule (.dru) and the CAM file settings (.cam) that will be used for the project.

3.) Move the *engr301-proj.lbr* to *ENGR_301* folder, which was created from Step 1.

4.) Open EAGLE. In EAGLE Control Panel, go to *Options > Directories*. In the Libraries text field, you want to copy the pathname of ENGR_301. This will allow EAGLE to locate and import other libraries you plan to use.
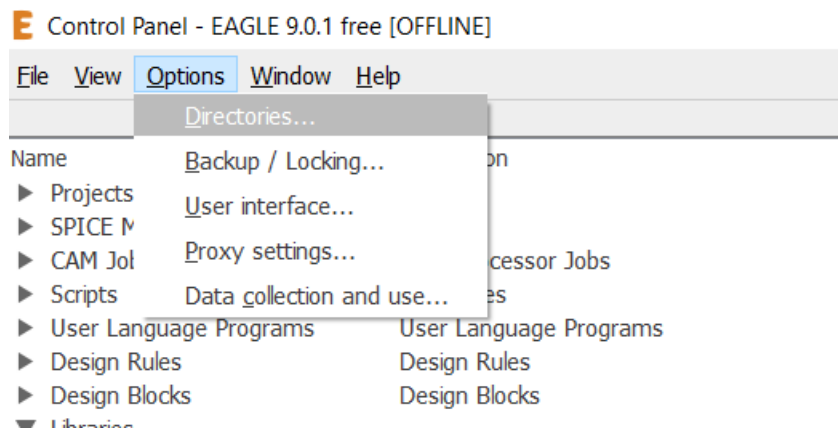

Figure 1

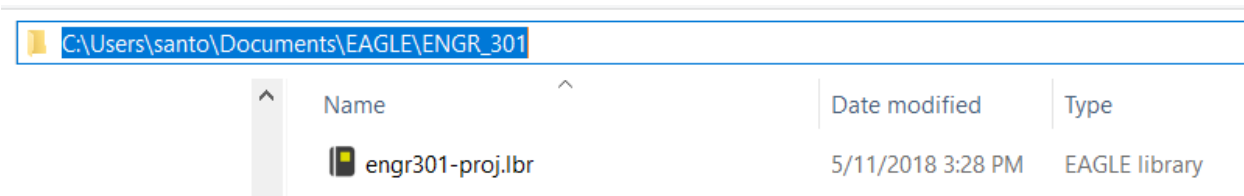In the meantime, go to the *ENGR_301* folder and copy the pathname.

Figure 2

Before you paste the pathname to the Libraries text field, insert semicolon (;). Then paste it to the text field. Click OK. See Figure 3.
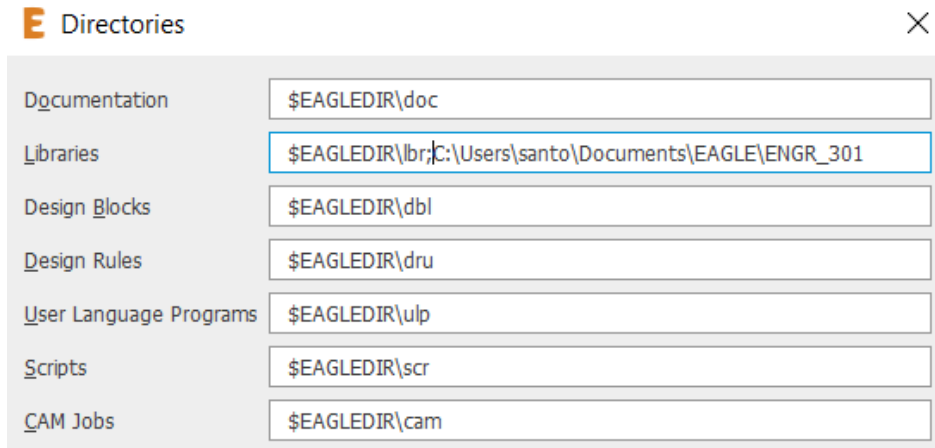
Figure 3

5.) In the EAGLE Control Panel, go to *Libraries > ENGR_301*. Select and right-click *engr301-proj.lbr > use*. The status will turn green, which means the library is ready to use. Also, you may want to enable the default libraries since we're going to use some of them. Right-click *Managed Libraries > use all*.
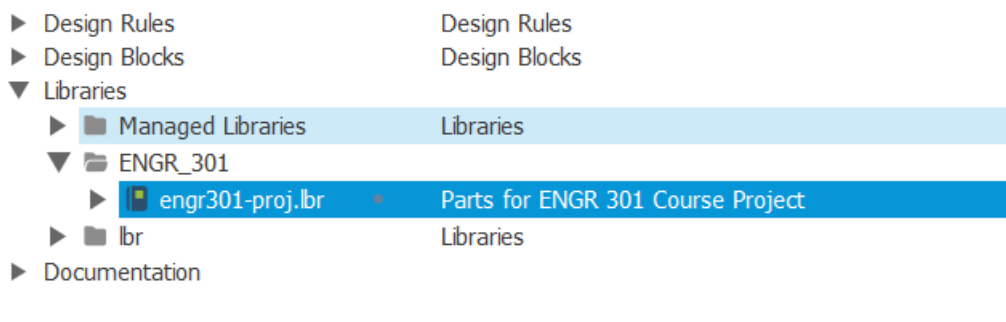

Figure 4

Figure 5

5.) To test the new components, you need to create a new schematic sheet. In the EAGLE Control Panel, go to *File > New > Schematic*.

6.) Go to "add" tool on the left pane right below the wrench icon, or type "add" in the command line.
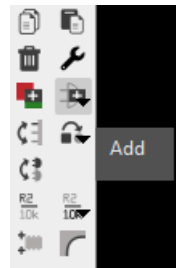

Figure 6

 Scroll down to engr301-proj. You should be able to find the regulators, NPN and PNP Power Transistors, OP275, etc. needed for the Final Project.
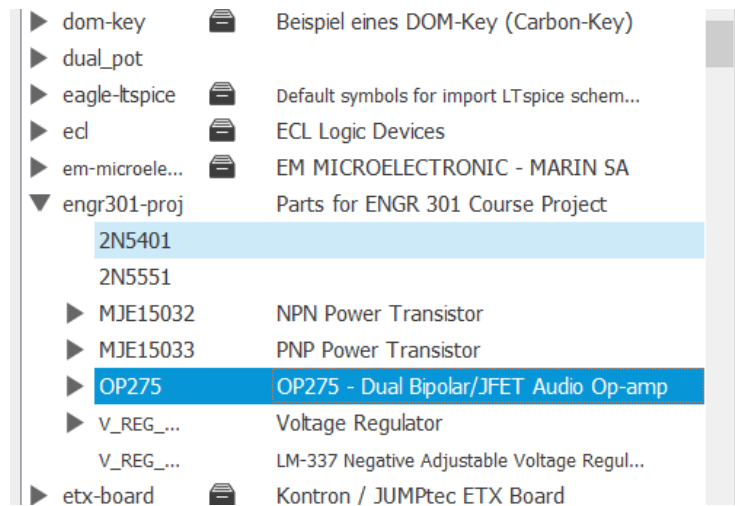

Figure 7

Some tips:

1.) We are mainly using **4** libraries…

- engr301-proj: for regulators, NPN, PNP, OP275
- rcl: for resistor and capacitor
    - C-US, C-EU, R-US
- diode: for 1N4148, 1N4004
- con-phoenix-254: to connect external wiring to the board

2.) When selecting the components, for example a capacitor, make sure that the component you will use on the board can be purchased from a vendor such as Digikey, Jameco, Amazon, etc.

3.) You can measure the dimension of a component on the board layout. You can do this by generating the board first from the schematic by selecting *Generate/switch to board.* Then use the "dimension" tool. By default, the unit is in mm, but you can also modify the displayed unit on the top pane.
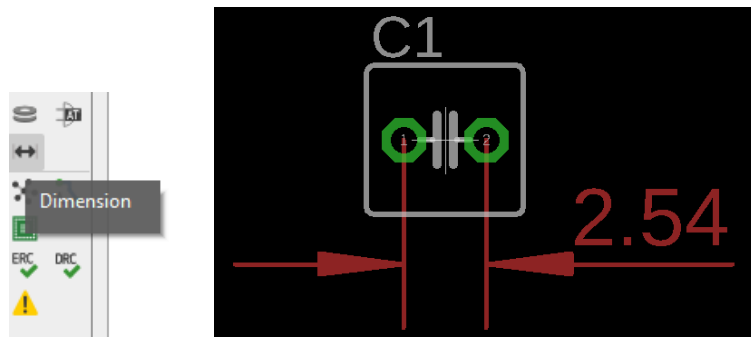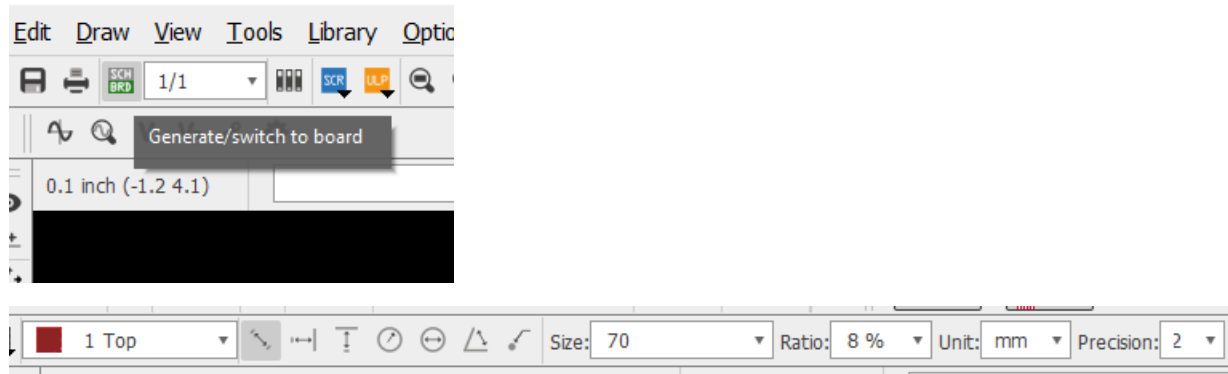

Figure 8