

A. Schematics

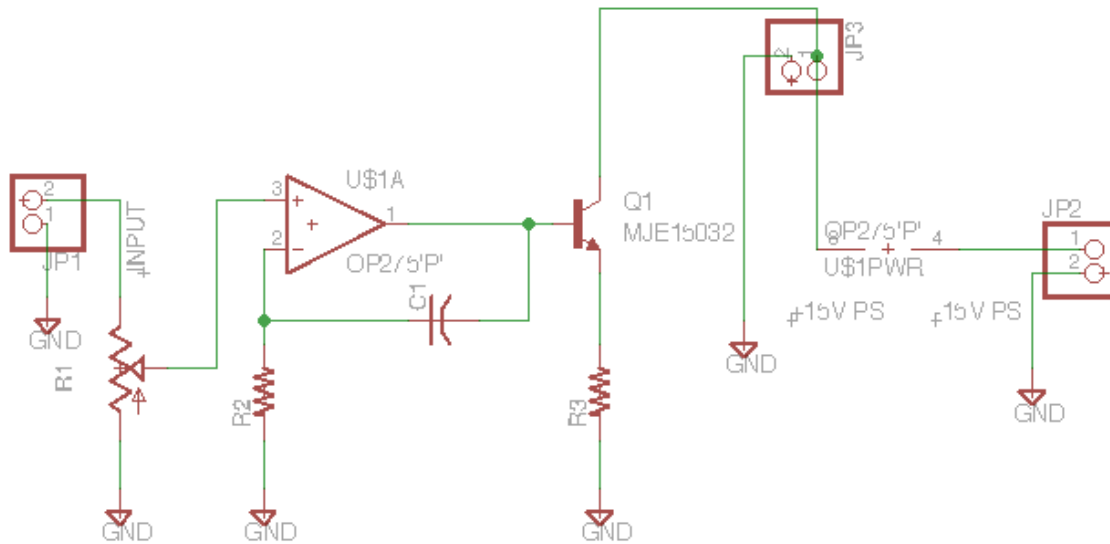


Fig.1 Sample Schematic View

1. Create a new project

Highlight project >> file >> new >> project >> type engr301project

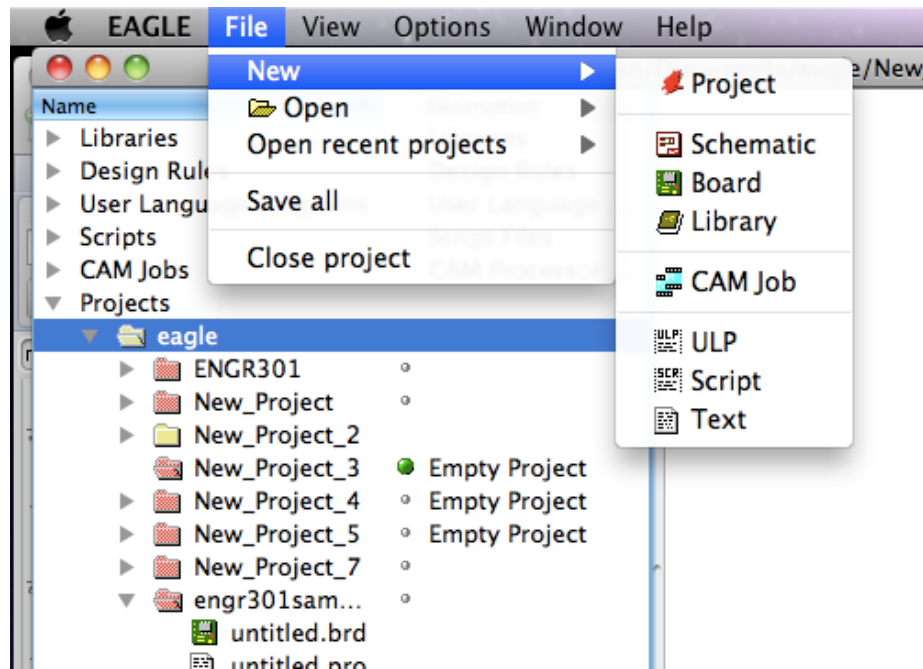


Fig.2 Create a Project

2. Create a schematic

Highlight engr301project >> file >> Schematic >> type engr301project

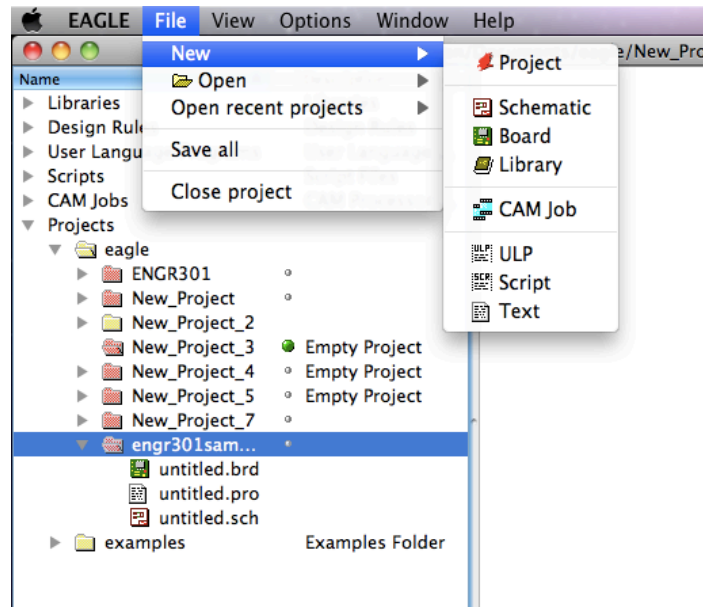


Fig.3 Create a Schematic

3. *Extend library*

- Download the library file named engr301-pro.lbr on iLearn
- Drag the file on eagle library:
Eagle6.4 >> lbr (drag the file in)
Highlight library >> right click >> use all

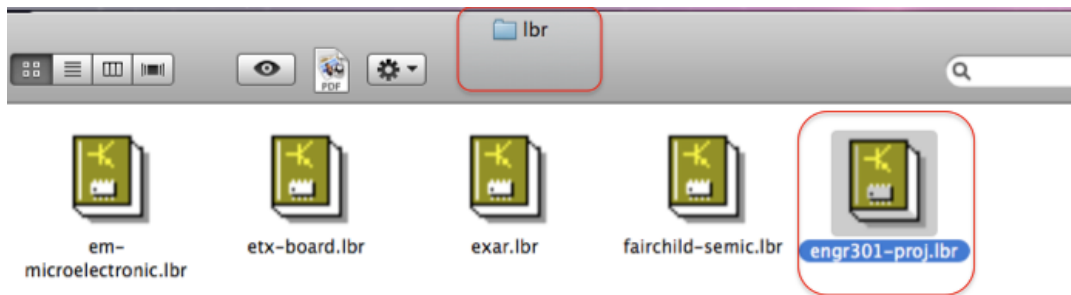


Fig.4 Extend Library

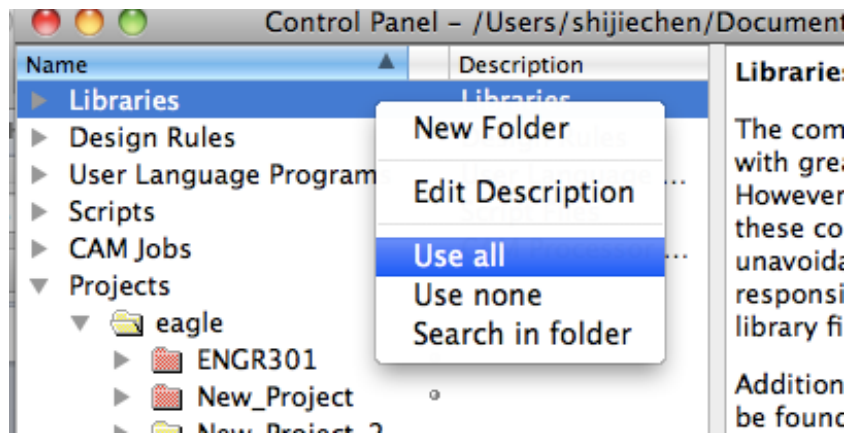


Fig.5 Use All Library File in Schematics

4. Add components

A. Op-amp

Type **add** on command window >> eagle301-pro >> OP275'P' >> ok >>
place on schematic window >> exist >> delete one op-amp

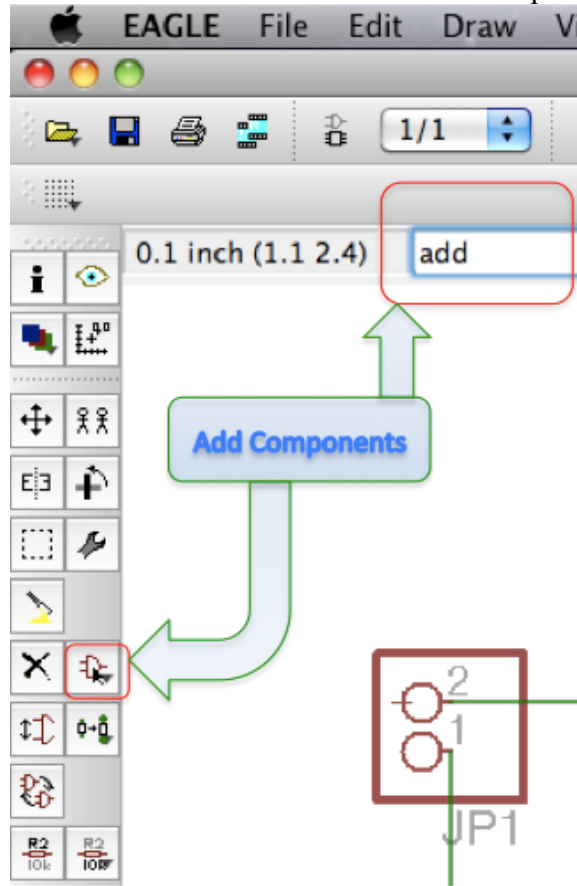


Fig.6 Add Components

B. Terminal block

Type command **add** >> pinhead >> PINHD-1X2 **Potentiometer**

Type command **add** >> pot >> TRIM-US >> TRIM_US-B25U

C. Resistor

Add >> rcl >> R-US_0204/7(R-US) >> ok >> right click on component >> value

D. Capacitor

Add >> rcl >> C-US >> C025-030X050(C-US)

E. Transistor

Add >> engr301-proj >> MJE15033

Add >> transistor >> PNP >> 2N5401-NPN-TO92-CBE

Add >> transistor >> PNP >> 2N5551-NPN

Add >> engr301-proj >> MJE15032

(Used in final project)

F. Ground

Add >> supply2 >> GND

5. Wire

Type command **net** to connect components

(optional) type **name** >> click on wires >> give wires a name

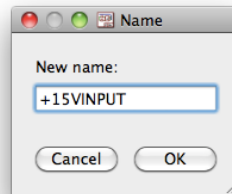


Fig.7 Give a Wire Name

6. Class

Type command **class** on command window >> enter parameter >> ok (Fig.8)

This process gives a definition of wire width on layout. Wires on layout automatically generate following the definition when using **autorouter**.

Wire width selection: right click on wire >> property >> enter parameter (Fig.9)

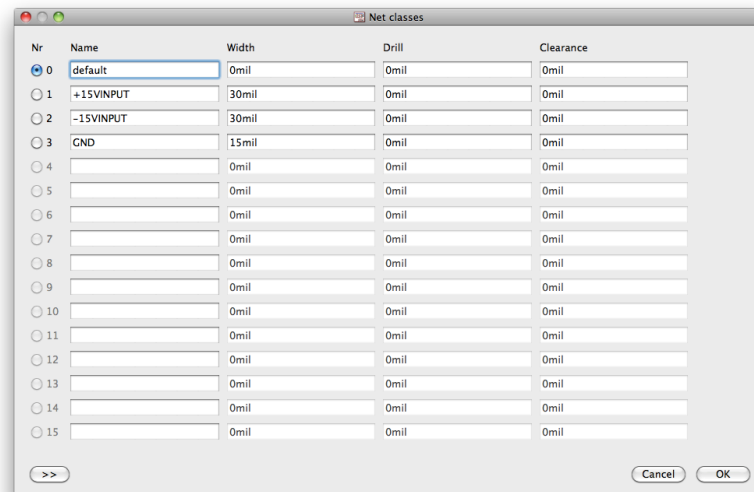


Fig. 8 Command **Class** Parameter

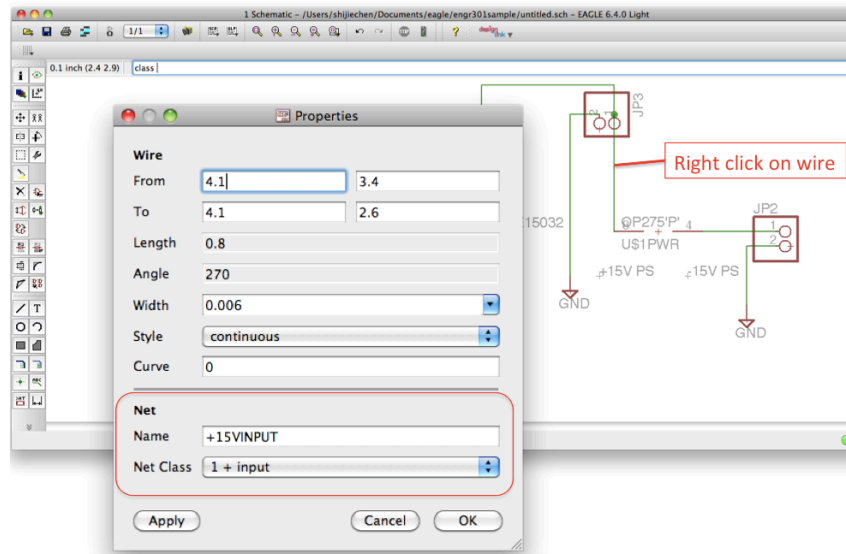


Fig.9 Net Class

7. Command function

Move a group of components:

Move >> group >> select group components >> right click >> move group

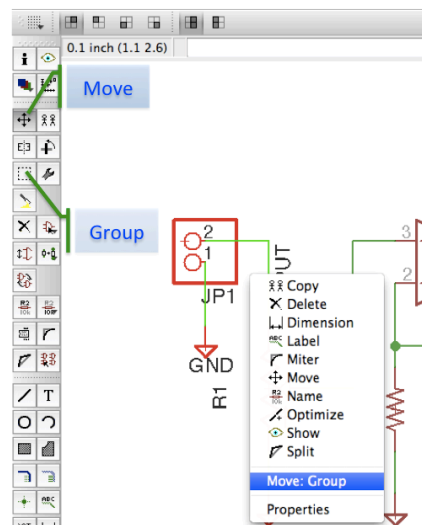


Fig.10 Components Group Movement

B. Layout

1. Type board on schematic command window to generate layout >> ok
2. PCB size (1.5X1.5 inches)
Click on the icon MOVE >> left click on boundary rectangle to reduce its size appropriately to 1.5X1.5 inches
Right click on gray line to check demission >> enter position parameters:
Line 1 (0,0;0, 1.5); Line 2 (1.5,0; 1.5,1.5); Line 3 (1.5,0; 1.5,1.5);
Line 4 (1.5,1.5; 1.5,1.5)

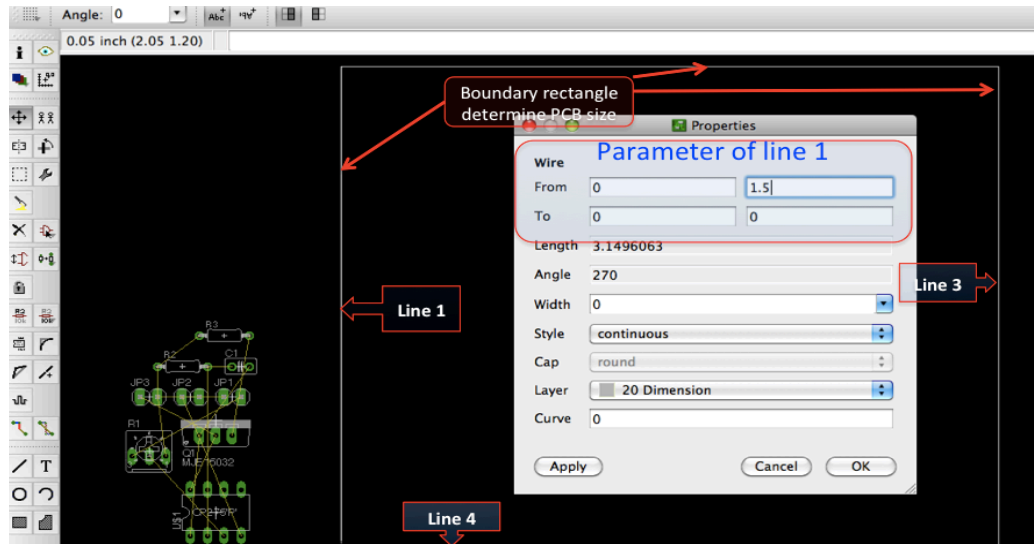


Fig.11 PCBoard Size

3. Design Rule

Download **Design Rule File** on ilearn >> DRC >> Load >> the file address >> choose **engr301.dru** >> open >> apply

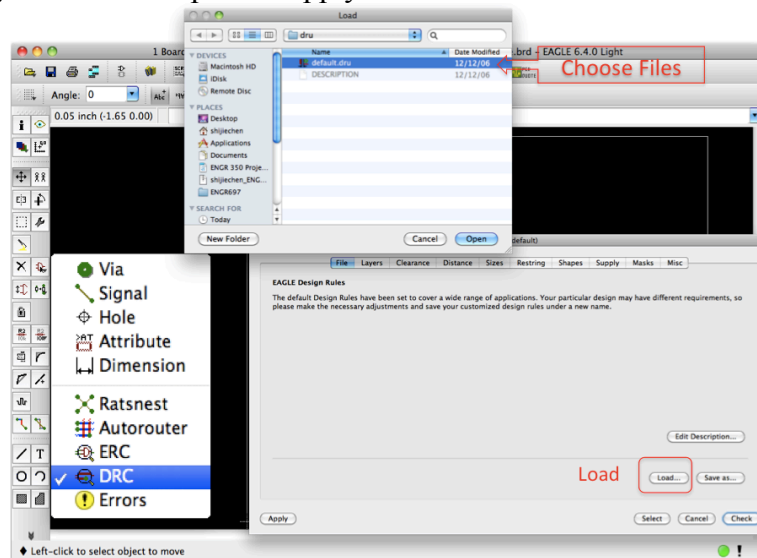


Fig.12 Load Design Rule

Different vendors have their own design rule. Make sure your design rule is compatible with vendor's design rule

4. Grid

Type **grid** on demand box >> size: 0.01 inch >> ok

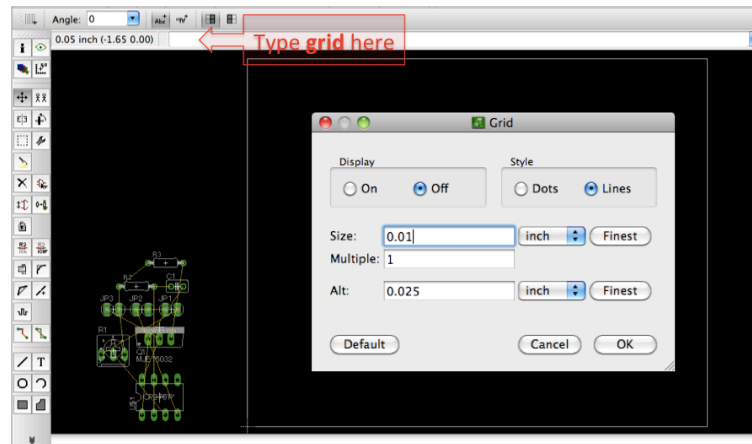


Fig.13 Define Grid Property

5. Hole
Add >> holes >> 2,8
6. Place components
Move >> left click to select components >> left click to place on expect spot
7. Route wire trace
Route trace: type command **route** >> click yellow line (that connected between components) to route trace
Air line indicates a connection between components. Don't delete air line because the deletion make the original circuit changed.
Trace has different color meant wire being mounted in different layer. This case is two-layer PCBoard. Red is top; similarly, blue is bottom.
Delete trace: type command **ripup** >> click on trace to change the trace to be air line (yellow line)

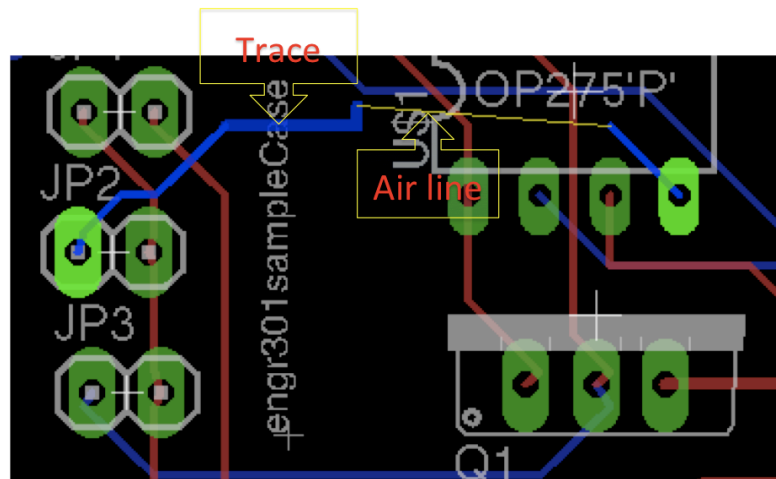


Fig.14 Route Trace on PCBs

8. Modify traces:
Move >> left click on the trace for movement
9. Polygon >> enclosure all the components >> AutoRoute

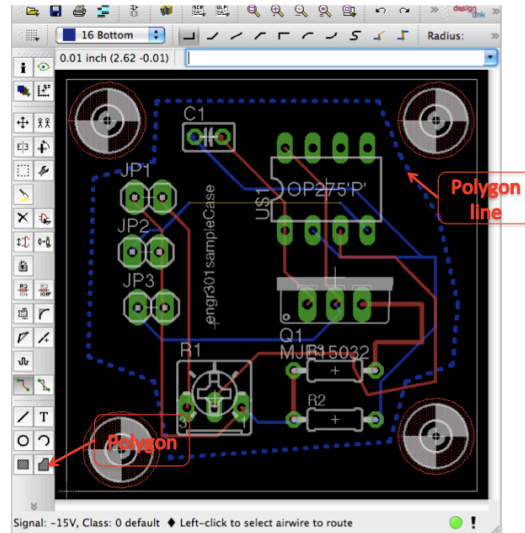


Fig.15 Polygon

10. DRC

DRC enables designer to check the error on PCBs

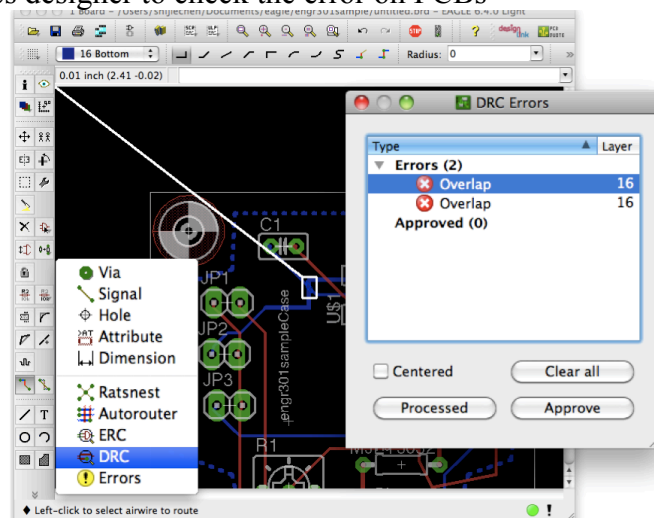


Fig.16 DRC