<u>Getting Started with Kicad</u> rev 022421

(Note: the terms **folder** and **directory** are synonyms. This document uses directory)

<u>FIRST step</u> Create a directory where you wish to place your design

<u>Second step</u> Install a WWU library of components to be used for this class

Download file aawwu_lib21.tar from the class webpage into your new project directory.

Open a terminal window and navigate to your project directory.

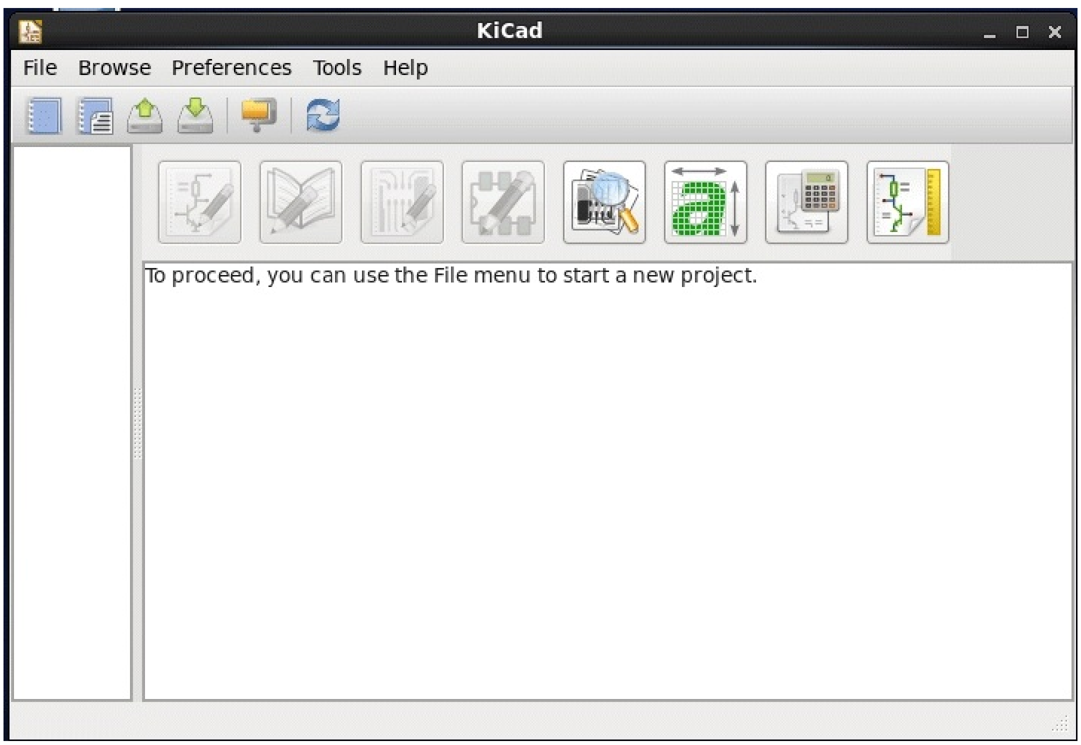
Type in this command: tar -xf aawwu_lib21.tar

This will unpack the library files into your project directory.

<u>Start Kicad</u>

Electronics > Kicad (This may vary depending on the window manager in use)

The project manager window will open and look something like this (if you have used Kicad before it may look different and it may display the name of the prior project)
:



Create a project.

File > New Project > New Project

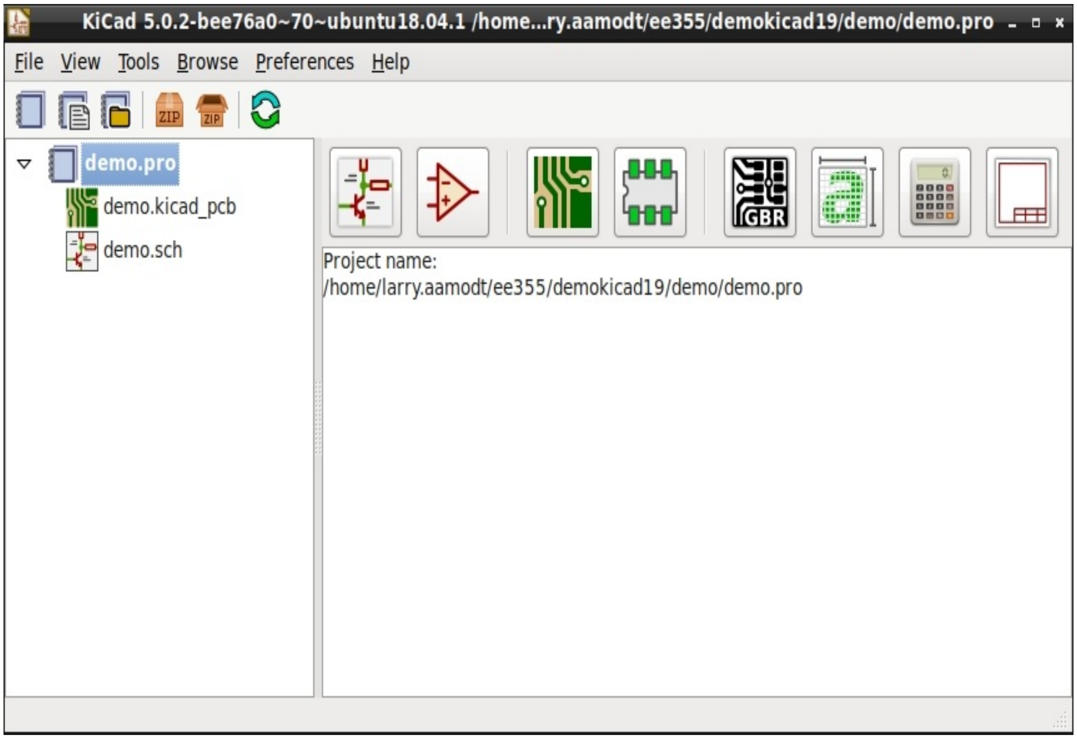
A new window will open.

Navigate to the directory you created for your design and fill in a name for the project.

In the lower left of the window is a little box asking if you wish to create a directory for the project. **Uncheck that box** since you have already created the directory for the project.

A warning message about not creating a directory may appear. That is ok.

The project manager window will then look something like this. The project file ends in .pro extension.



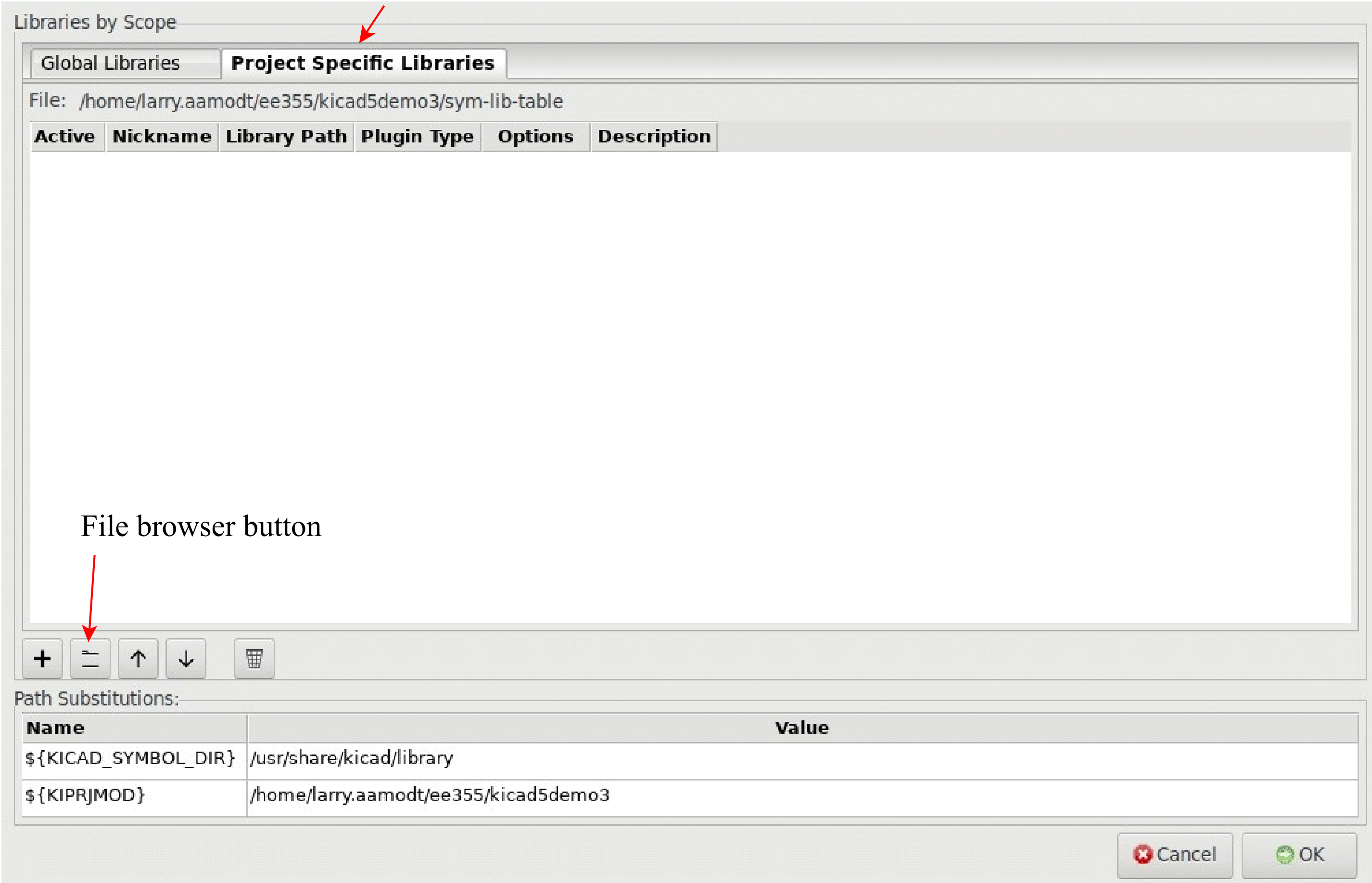
On the Project Manager window there are 8 large icons above where the project name is displayed. These are buttons that will launch a respective tool (place cursor over icon to see tool name):

schematic layout editor (eeschema)
library layout editor
PCB layout editor (pcbnew)
footprint library editor
gerbview - gerber file viewer
bitmap to component - file convertor
pcb calculator
worksheet layout editor

The next thing to do is to click **Preferences** on the tool bar and from the drop-down menu select **Manage Symbol Libraries**

A Libraries by Scope window will open (see the next page).

Click on the **Project Specific Libraries** tab



Then click the button at lower left between the + and up-arrow buttons. This is the file browser button.

A **Select Library** window will open. It should show your project directory, but if needed, navigate to your project directory. You should see the file aawwu.lib

Click on **aawwu.lib** to highlight it. Then click **Open**

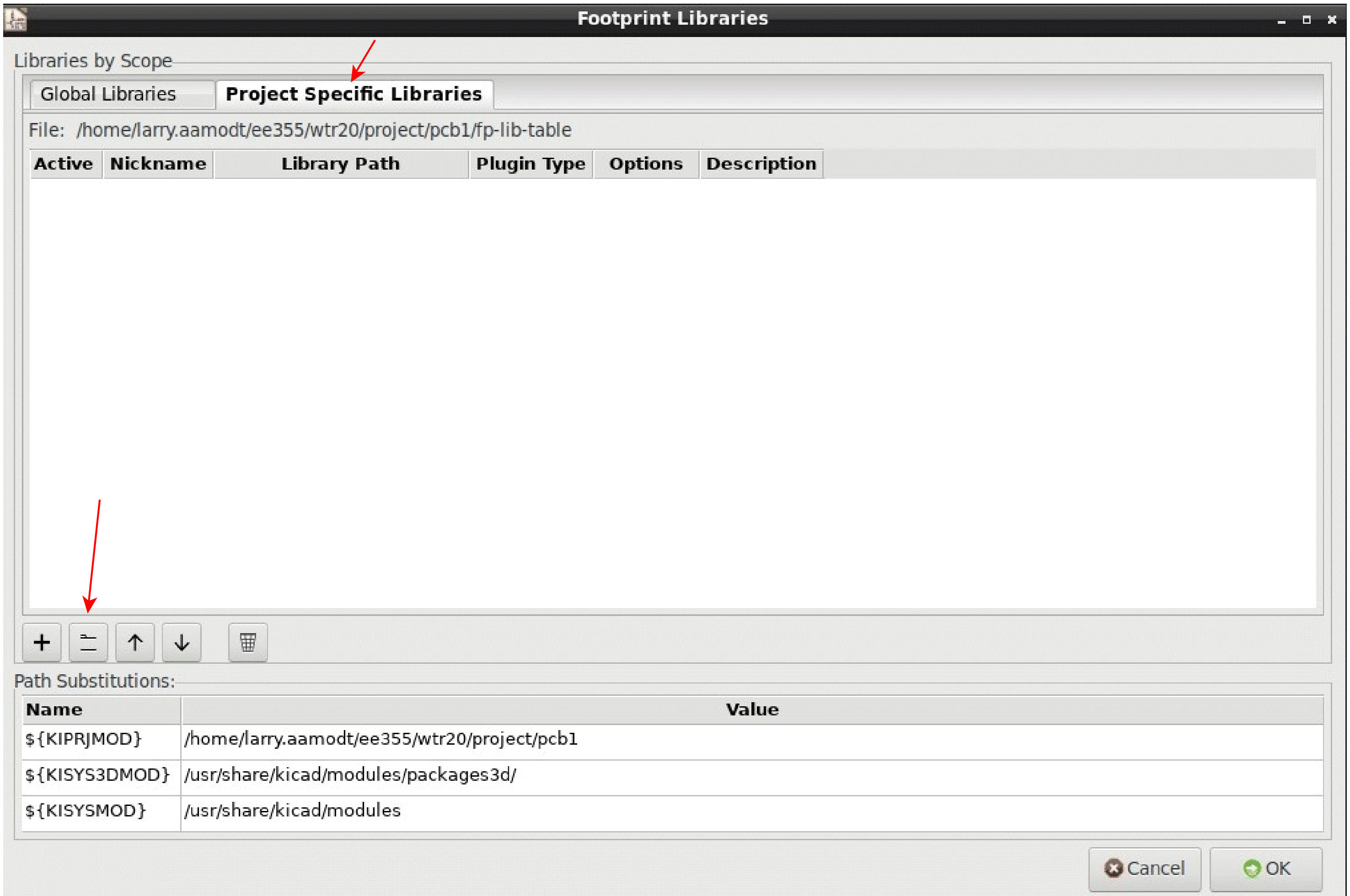
The aawwu.lib file should now appear in the Libraries by Scope window.

Click on **OK** to close the libraries window

Next, again click **Preferences** on the tool bar and from the drop-down menu select
Manage Footprint Libraries

A footprint Libraries by Scope window will open

Click on the **Project Specific Libraries** tab.



Then click the button at lower left between the + and up-arrow buttons. This is the file browser button.

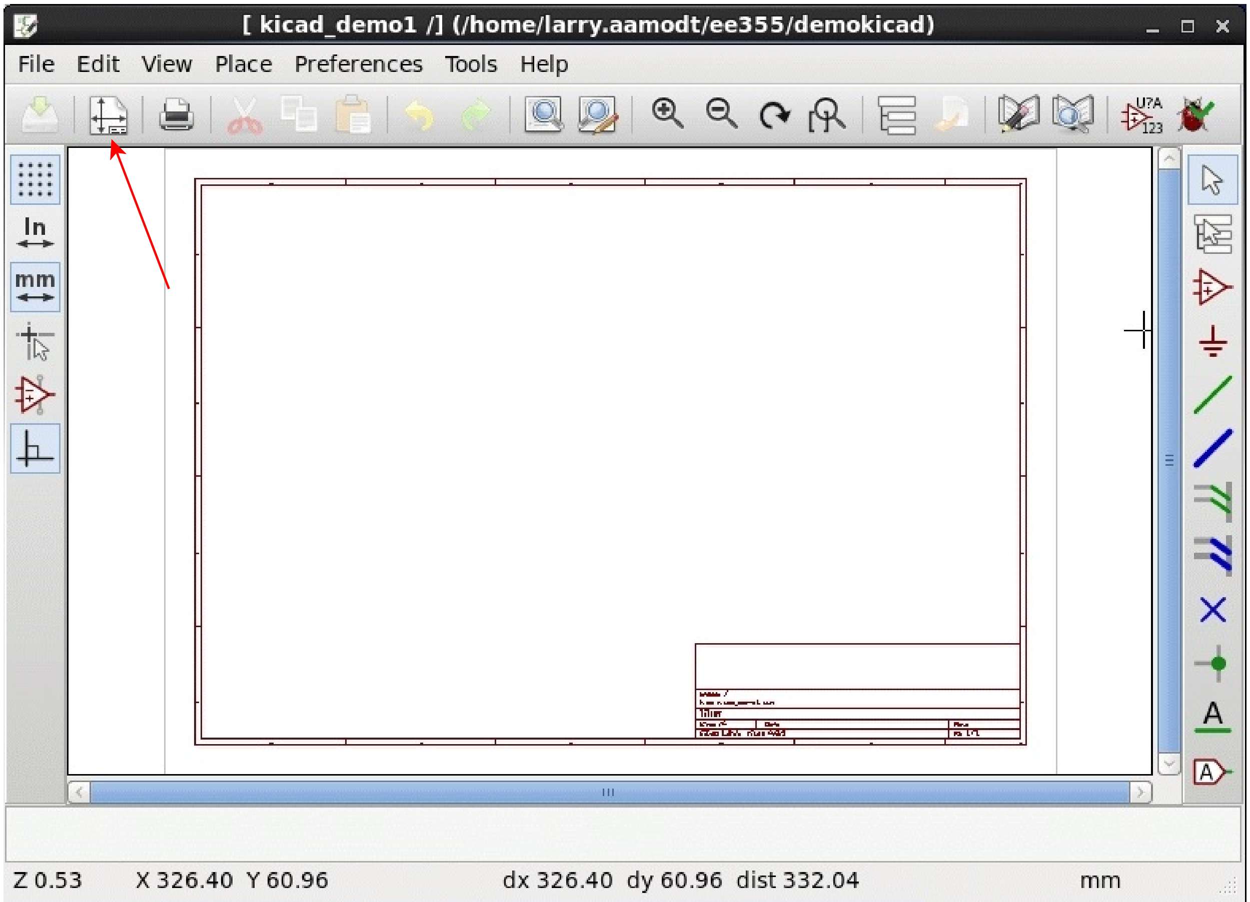
A **Select Library** window will open. It should show your project directory, but if needed, navigate to your project directory. You should see the directory (folder) aawwu.pretty

Click on **aawwu.pretty** to highlight it. Then click **Open**

The aawwu.pretty directory should now appear in the Libraries by Scope window.

Click on **OK** to close the libraries window

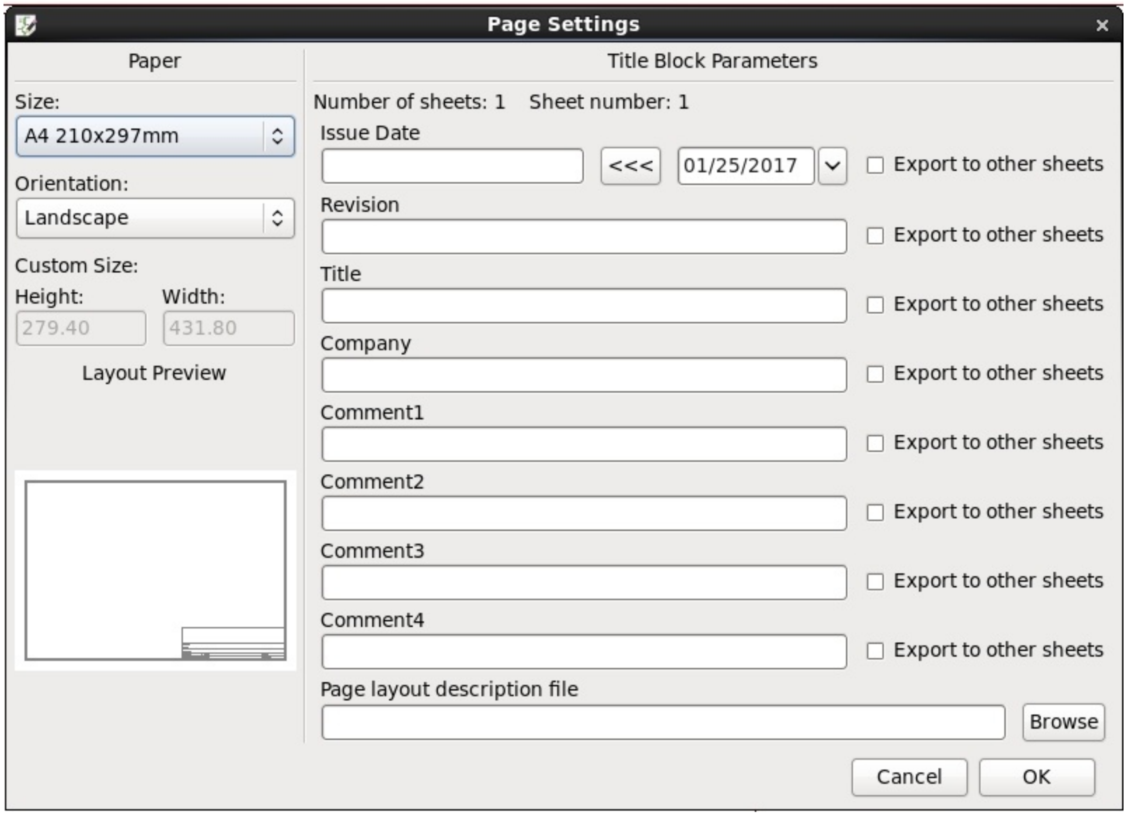
Start the schematic layout editor, eeschema, by clicking it's icon (the left large icon in which a transistor symbol can be seen). Something like this should appear:



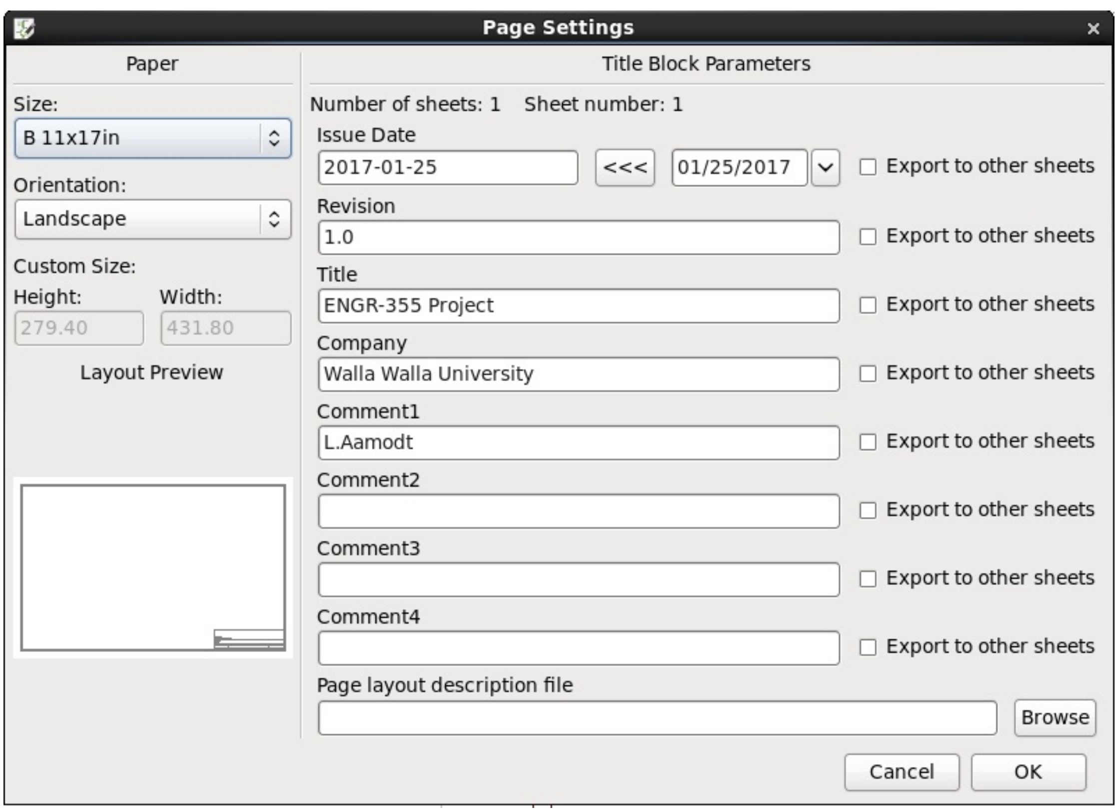
The default sheet size likely will be A4. That is similar to American size A, 8.5 x 11 inches. That may be too small, so change the sheet size to B which is 11x17 inches.

Click the button pointed to by the red arrow above
(or Select **File > Page Settings**   on the tool bar)

and the Page Settings window should open as shown on the next page:



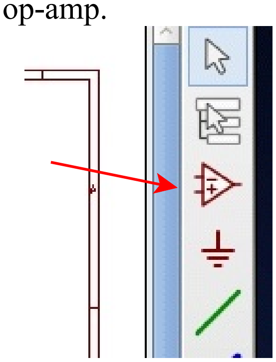
Size can be set with the drop down menu at upper left. Also, various fields can be filled in with data that will appear in the title block. Here is what I filled in (your name is required):



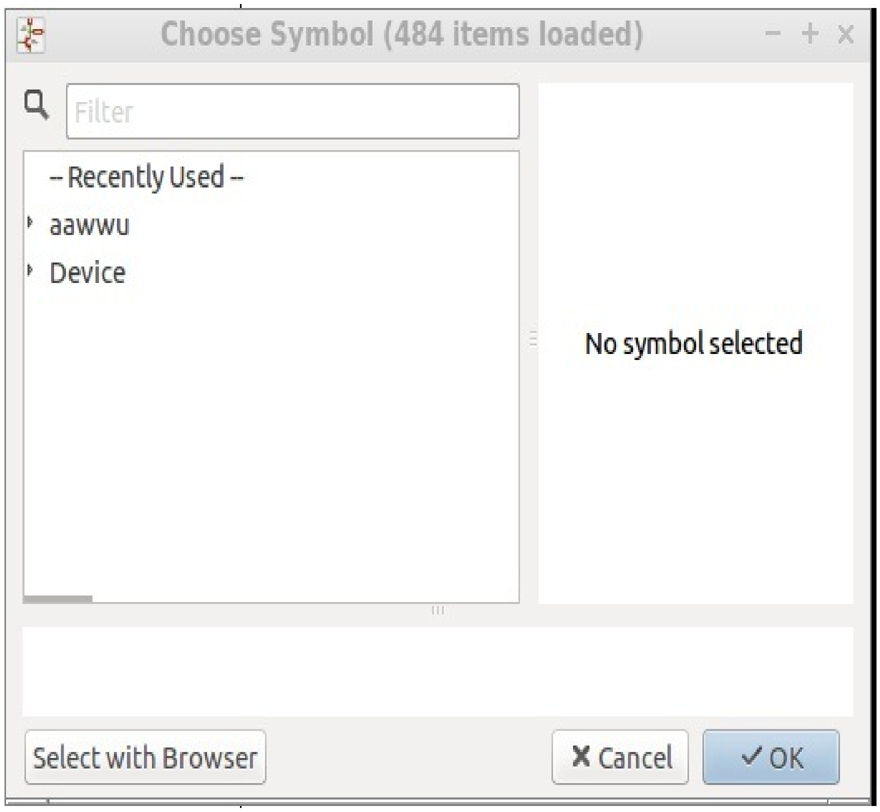
To begin placing components on the schematic sheet click the note the tool bar on the right. There is what appears to be the symbol for an op-amp.

Click the **“op-amp” symbol**.

Then move the cursor over the drawing area of the sheet and click again.



A component selection window should open. A list of libraries will be shown.

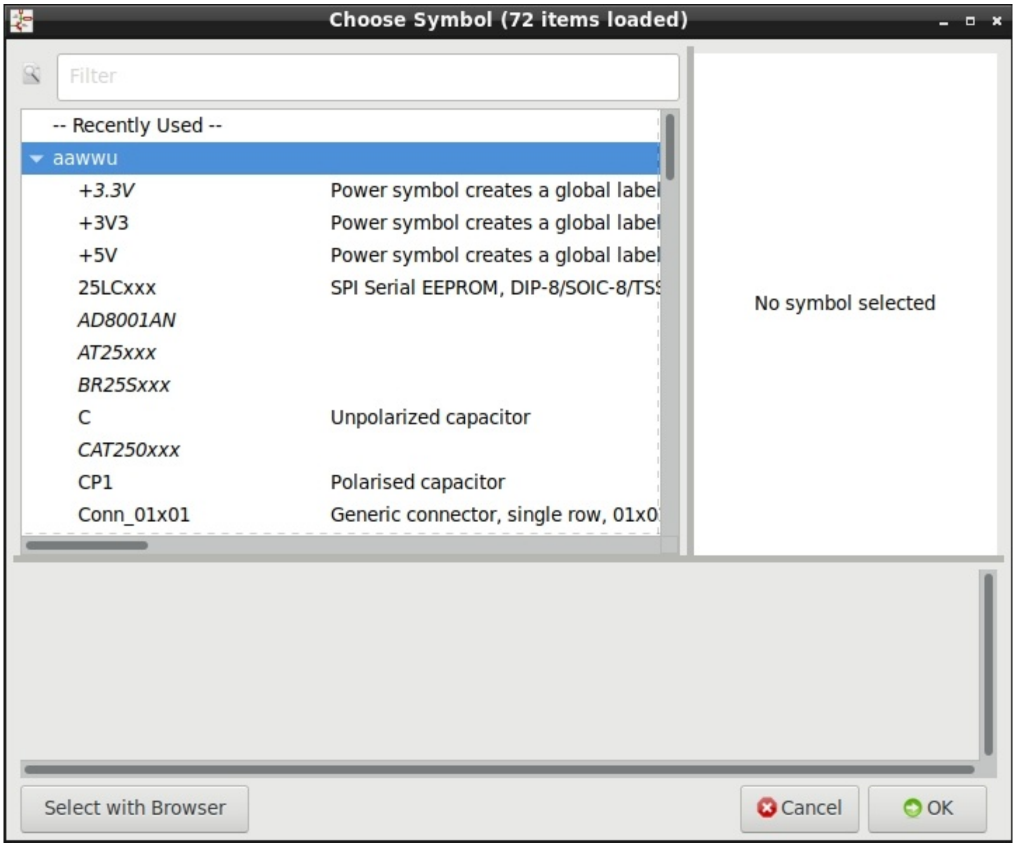


Near the top should be a library named aawwu

Click on **aawwu**

Clicking on the name of a library will open up a list of the components in that library

A list of the components in a library (once used, recent components will be shown at the top):



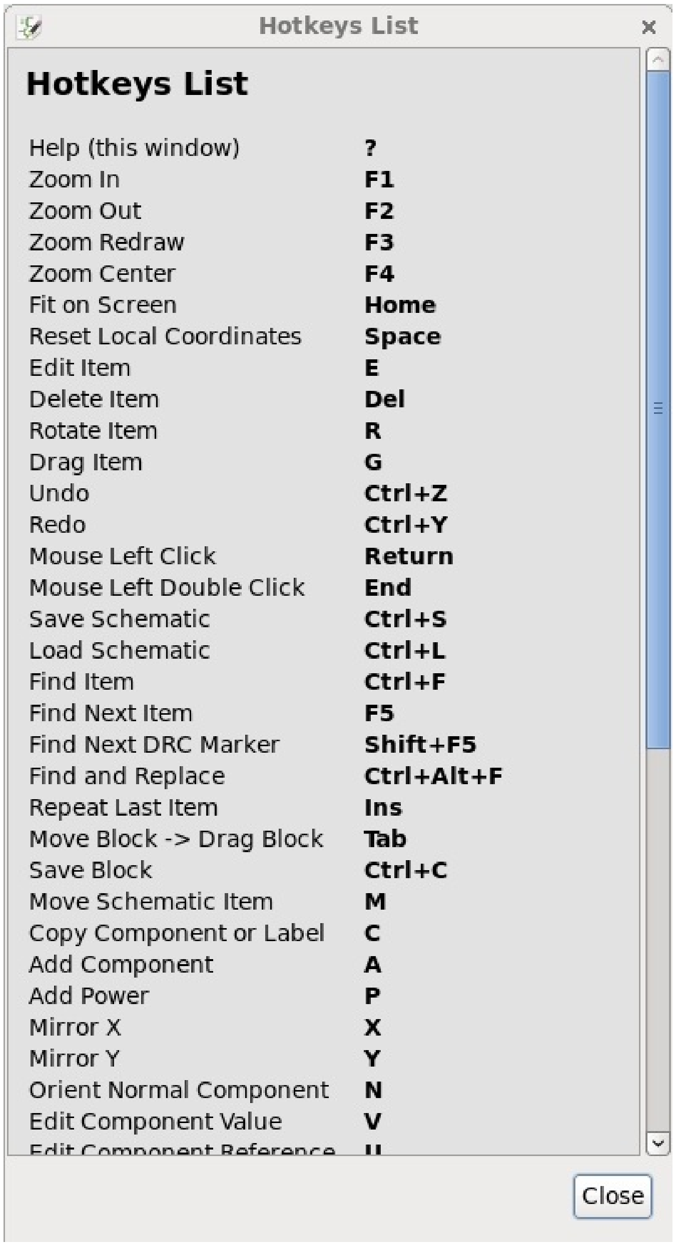
Click on a desired component and its symbol will appear in the pane to the right.

Click OK to select this component and enable placing its symbol on the schematic.

A symbol for the component should be linked to the cursor. Move cursor/component to desired location on the drawing sheet and click to place it.

(NOTE: There may be other libraries in addition to aawwu. For ENGR-355 projects only use parts from the aawwu library).

There are many hot keys. Get a list by pressing ?   A partial list is:



Hotkeys List

Hotkeys List

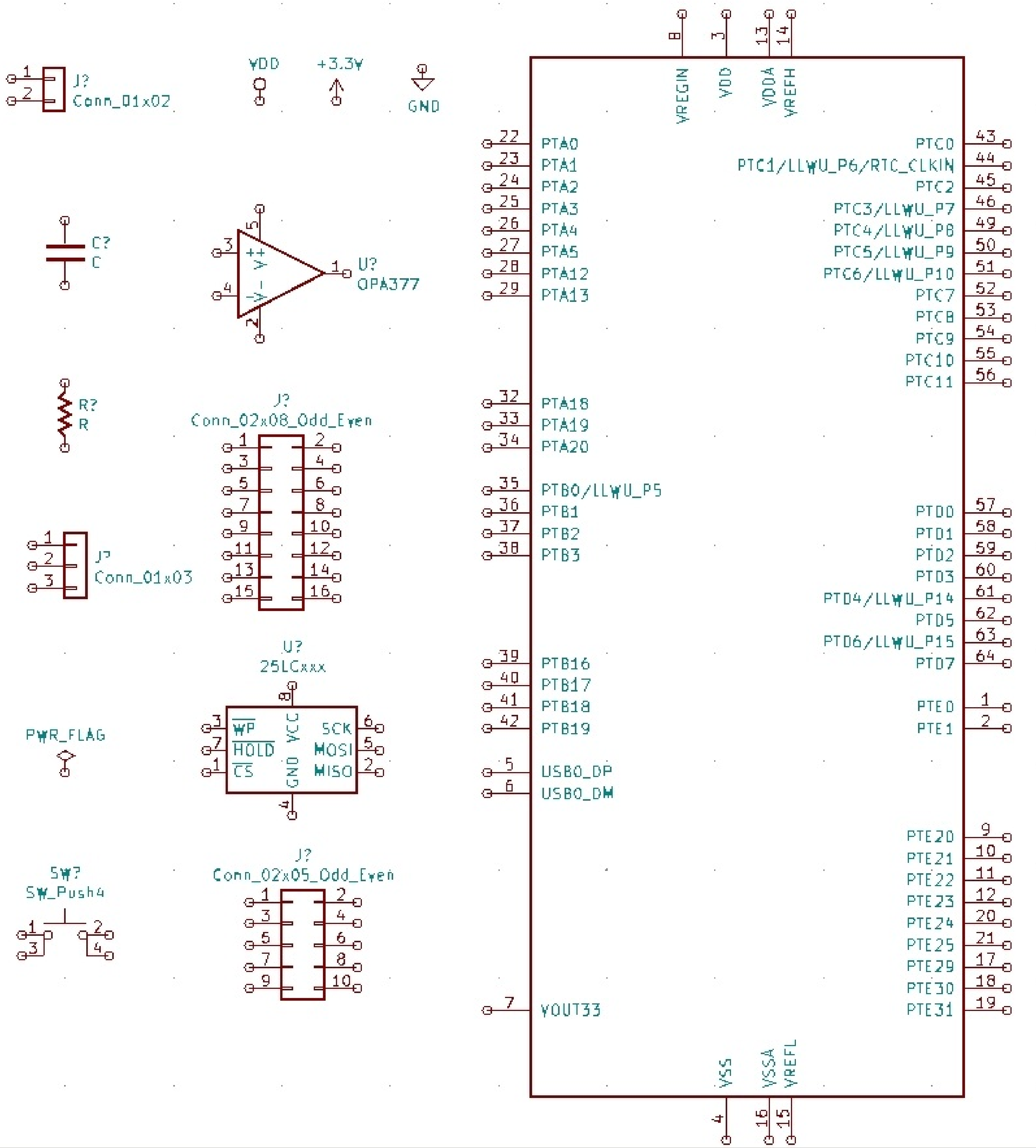
Help (this window)	**?**
Zoom In	**F1**
Zoom Out	**F2**
Zoom Redraw	**F3**
Zoom Center	**F4**
Fit on Screen	**Home**
Reset Local Coordinates	**Space**
Edit Item	**E**
Delete Item	**Del**
Rotate Item	**R**
Drag Item	**G**
Undo	**Ctrl+Z**
Redo	**Ctrl+Y**
Mouse Left Click	**Return**
Mouse Left Double Click	**End**
Save Schematic	**Ctrl+S**
Load Schematic	**Ctrl+L**
Find Item	**Ctrl+F**
Find Next Item	**F5**
Find Next DRC Marker	**Shift+F5**
Find and Replace	**Ctrl+Alt+F**
Repeat Last Item	**Ins**
Move Block -> Drag Block	**Tab**
Save Block	**Ctrl+C**
Move Schematic Item	**M**
Copy Component or Label	**C**
Add Component	**A**
Add Power	**P**
Mirror X	**X**
Mirror Y	**Y**
Orient Normal Component	**N**
Edit Component Value	**V**
Edit Component Reference	**U**

Close

Power connections could be made by drawing lines between component pins that connect to a common power. However, this can make the drawing messy, particularly for ground connections. So typically we use a ground symbol anywhere a node connects to ground and typically a power symbol such as Vdd or Vcc where a node connects to a power source.

After placing needed parts on the schematic, connect wires between them by clicking on the green line in the right-hand tool bar to select the wiring tool and clicking starting point and ending point for desired wires.

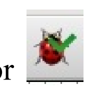
Project parts

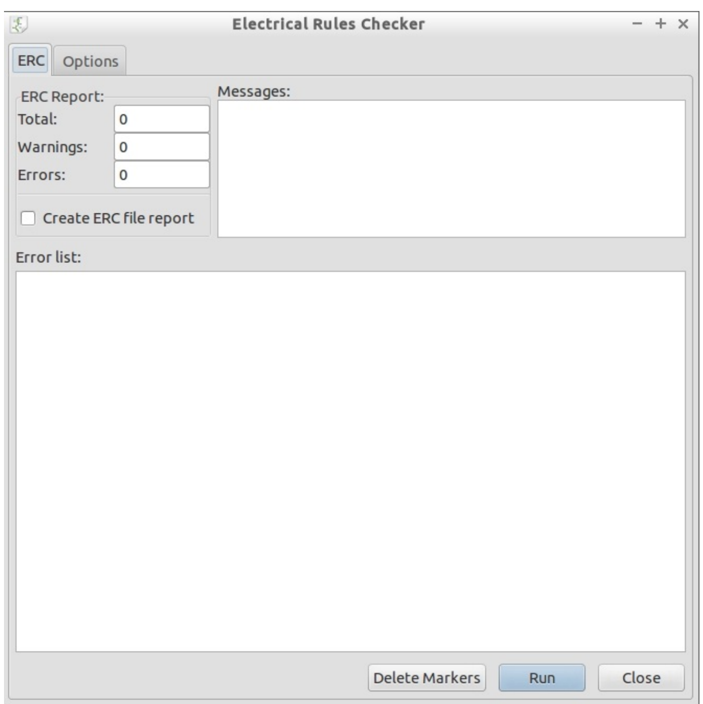


To summarize, these steps are needed: to create the schematic and prepare for circuit board layout:

1) Place parts
2) Draw nets
3) Add power/ground symbols
4) Name important nets (Place -> Local Label)
5) Edit title block (File -> Page Settings)
6) Add comment text as appropriate (Place -> Text)
7) Annotate schematic (Tools -> Annotate Schematic)
   Annotate means to fill in the sequential number portion of component reference designators such as R1, R2, U1, U2, etc.
8) Run the Electrical Rule Checker (often referred to as the Design Rule Checker or DRC). (Tools -> Electrical Rule Checker)
   Note: You likely will have one or more errors. More on this below.
9) Associate schematic symbols and footprints for layout. More on that below. (Tools -> Assign Component Footprints) or edit each symbol and add foot print.
10) Generate a netlist file (Tools -> Generate Netlist File)

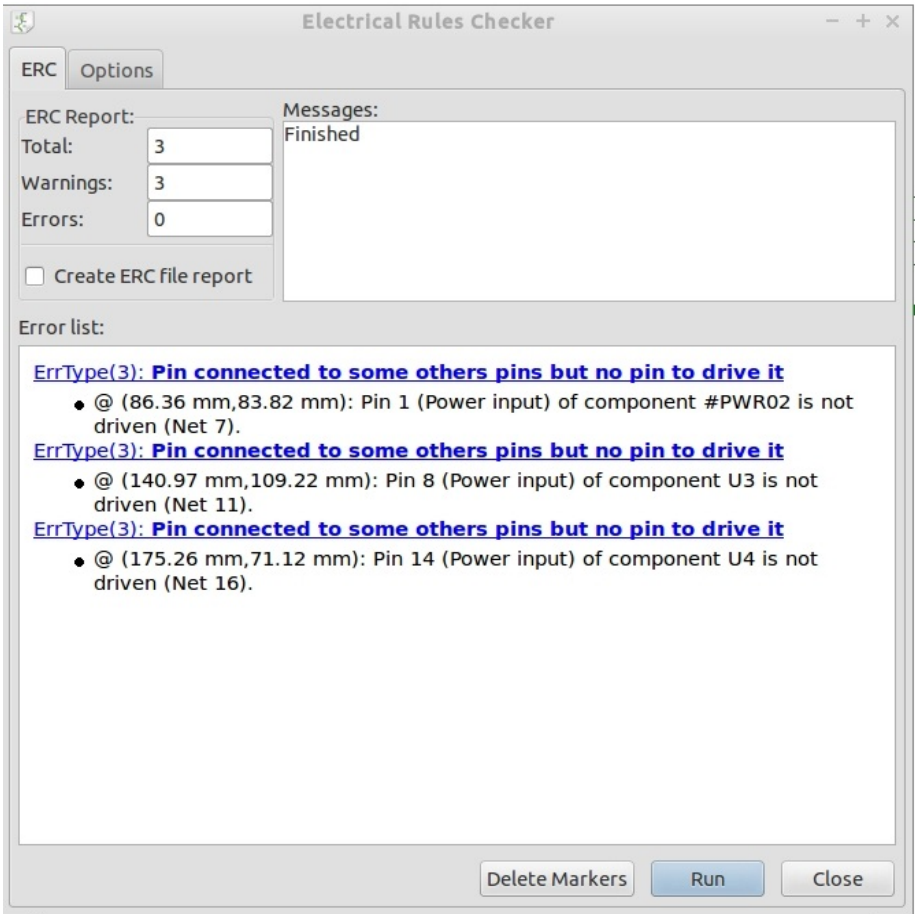
Running the electrical rule checker and solving reported errors

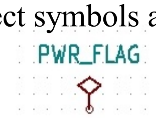
Start the rule checker from pulldown menu (Tools -> Electrical Rule Checker) or  icon from the upper tool bar. A blank dialog box should open:



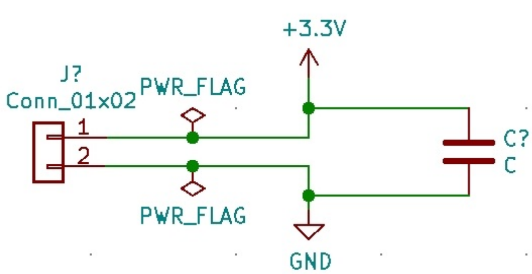
Then click Run to do a rule check. Note that the annotate schematic step needs to be done before rule checking works.

Here is an example result that shows errors involving power connections:



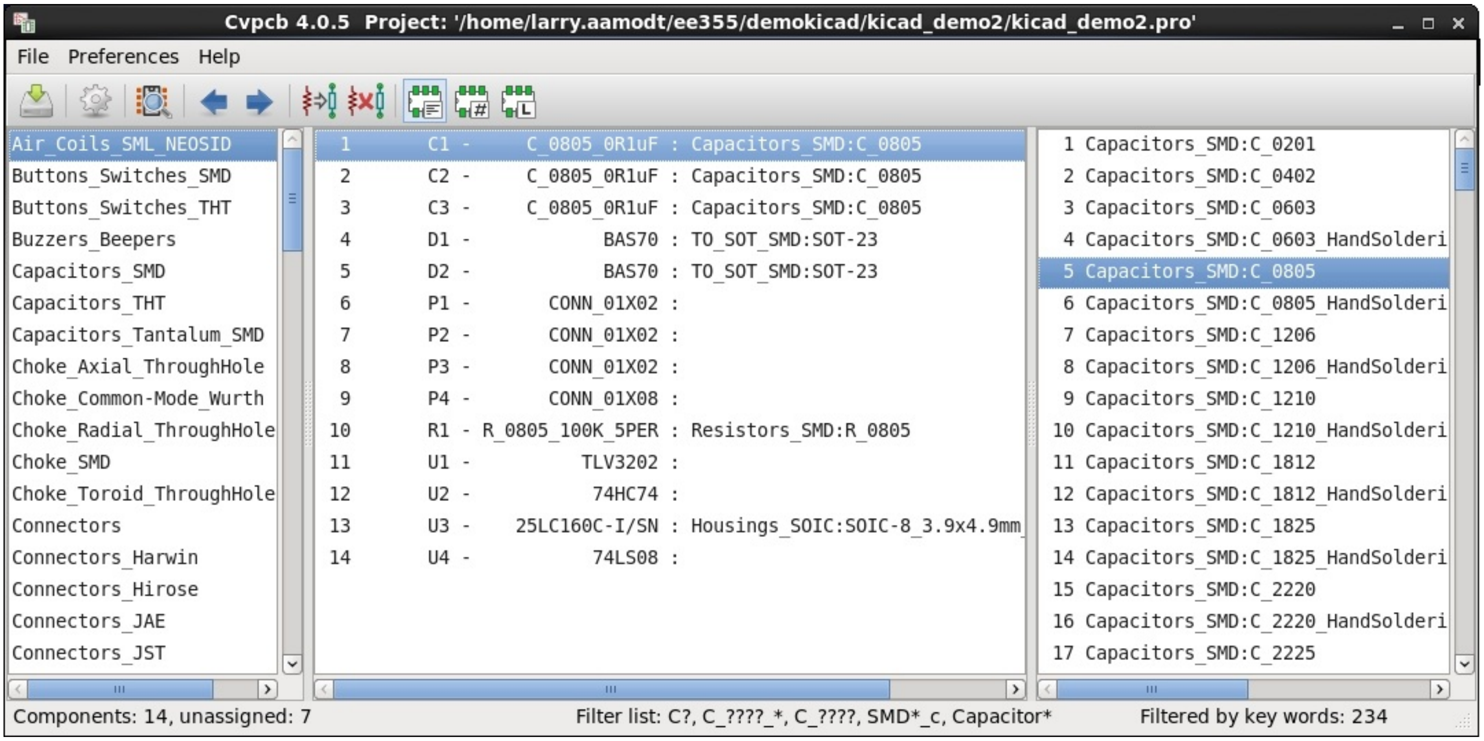
A common error is with power distribution. In the power symbols list there are many choices such as Vdd, Vcc, +3.3V, +3V3, GND, etc. and you likely have used one or more of these. Some components such as logic parts, memory parts, op-amps, etc. have power pins that are not initially visible on the schematic. If the name on a power symbol the schematic doesn't match the name used by a component (which may be hidden initially) errors may be reported. By default, for components such as 74HCxxx or 74LSxxx parts the power pins don't appear on the schematic symbol. You can make them visible using the hidden pints icon:  on the left tool bar. Add visible symbols to your power net to match the hidden ones on your component symbols. Also, even though the correct symbols are used the power net(s) need to be identified as such with the following symbol: 

Place this symbol on a ground net and your power net. A sample circuit:



Associate schematic symbols with board layout foot prints
Using (Tools -> Assign Component Footprints) or a top tool bar icon open the tool.

Click OK. A window similar to this will open:

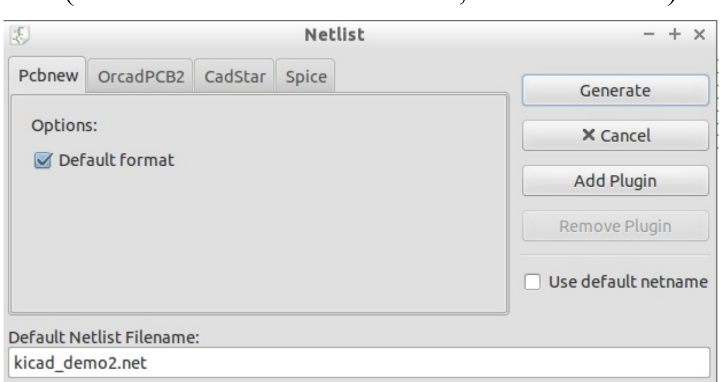


The left panel is a list of libraries. The middle panel is a list of components in your design for which foot prints need to be assigned. The right panel is a list of foot prints found in the selected library.

Some of your components have foot prints associated with them using information in the symbol and those foot prints are shown. Click on a component that doesn't have a foot print yet associated (i.e. blank to the right of the colon). Then click on the library with the desired foot print. Scroll through the foot prints and find the correct one. Double click on it to associate with the highlighted component in the center panel. Do this for all components as needed.

After all components have been associated with a foot print do a save.

Write the netlist
(Tools -> Generate Netlist File; Click Generate) Use defaults as shown in figure below.



A file called a netlist is used to transfer the design described by the schematic to the circuit board layout program. This netlist contains a list of the components in the design, a list of the connections (nets) between components, and information giving the physical geometry of the footprint for each type of component.

(The netlist file written by Kicad is an ASCII file meaning that you can use a text editor to open it and read the contents if you are curious about what is in it.)

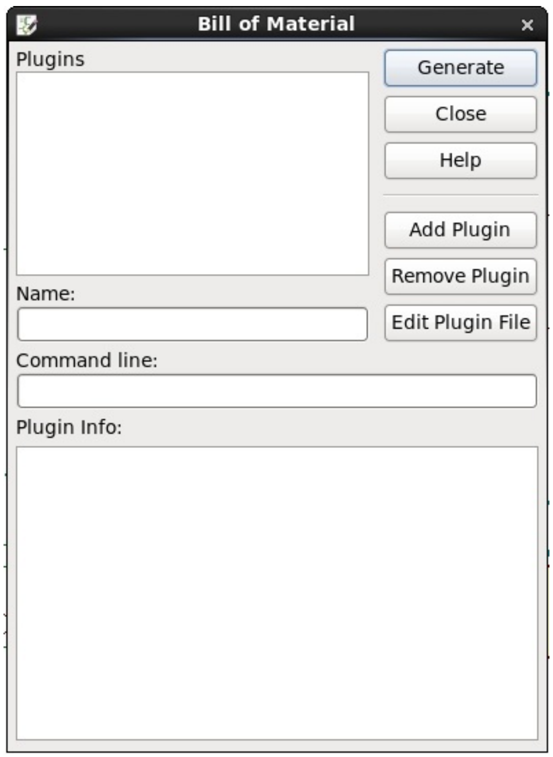
(You don't need to create the bill of materials today)

Write the Bill of Materials File (BOM file)

(Tools -> Generate Bill of Materials)

The bill of materials file lists the components used, quantity, possibly a supplier part number, etc.

When this tool is invoked it will display a window like this:



A plugin that will write the BOM file in a particular format needs to be added.

Click: Add Plugin and navigate to the folder shown below. Select bom2csv.xsl