rev 022421

Getting Started with Kicad

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

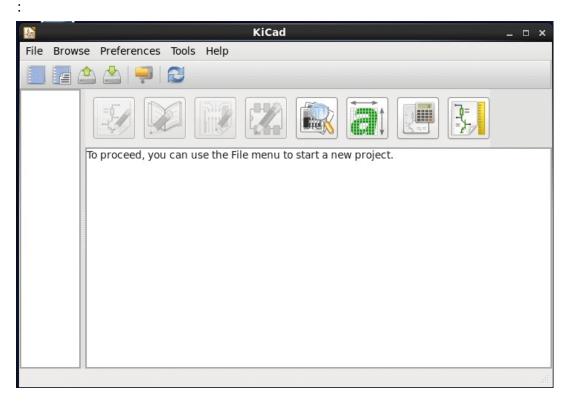
FIRST stepCreate a directory where you wish to place your designSecond stepInstall a WWU library of components to be used for this classDownload file aawwu\_lib21.tar from the class webpage into your new project<br/>directory.Open a terminal window and navigate to your project directory.<br/>Type in this command: tar -xf aawwu\_lib21.tar<br/>This will unpack the library files into your project directory.

#### Start Kicad

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.

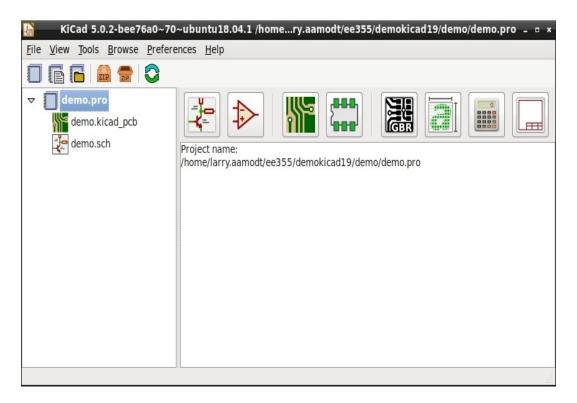
File > New Project > New Project

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

	t/ee355/kicad5demo3/s prary Path Plugin Typ		Description			
Active Nickname Li	brary Path Plugin Typ	pe Options	Description			
File browser	button					
Substitutions:						
me				Value		
	/usr/share/kicad/library					
KIPRJMOD}	/home/larry.aamodt/ee3	355/kicad5dem	103			
kii njinob)					😮 Cancel	Ok

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.

			Fo	otprint Li	braries		-
oraries l	by Scope						
Global	Libraries	Project Specific Librarie	es				
ile: /ho	ome/larry.aa	modt/ee355/wtr20/project/pcb3	1/fp-lib-table				
Active	Nicknam	e Library Path	Plugin Type	Options	Description		
	ı.						
	$\uparrow$ $\downarrow$	T					
- 2							
	titutions:						
h Subs	titutions:				Value		
h Subs ame		ome/larry.aamodt/ee355/wtr20	/project/pcb1		Value		
h Subs ame KIPRJN	MOD} /h	ome/larry.aamodt/ee355/wtr20 sr/share/kicad/modules/packag			Value		
h Subs ame {KIPRJN {KISYS:	MOD} //				Value		
h Subs ame {KIPRJN	MOD} //	sr/share/kicad/modules/packag			Value	Cancel	

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

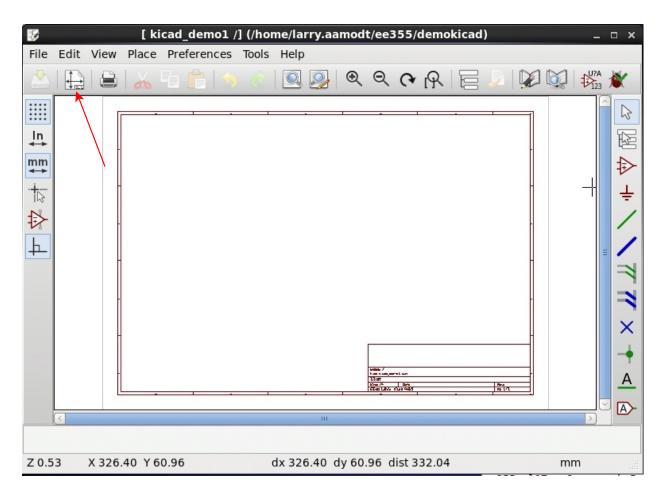
A **Select Library** window will open. It should show your project directory, but if needed, navigate to your project directory. You should see the directory (folder) aawwu.pretty

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,  $8.5 \times 11$  inches. That may be too small, so change the sheet size to B which is  $11 \times 17$  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:

8	Page Settings x
Paper	Title Block Parameters
Size:	Number of sheets: 1 Sheet number: 1
A4 210x297mm 🗘	Issue Date
Orientation:	<pre>&lt;&lt; 01/25/2017</pre>
Landscape 🗘	Revision
Custom Size:	Export to other sheets Title
Height: Width:	Export to other sheets
279.40 431.80	Company
Layout Preview	Export to other sheets
	Comment1
	Export to other sheets
	Comment2
	Export to other sheets
	Comment3
	Comment4
	Export to other sheets
	Page layout description file
	Browse
	Cancel

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):

8	Page Settings	×
Paper	Title Block Parameters	
Size:	Number of sheets: 1 Sheet number: 1	
B 11x17in	Sissue Date	
Orientation:	2017-01-25	Export to other sheets
Landscape	Revision	
· · ·	1.0	Export to other sheets
Custom Size:	Title	
Height: Width:	ENGR-355 Project	<ul> <li>Export to other sheets</li> </ul>
279.40 431.80	Company	
Layout Preview	Walla Walla University	Export to other sheets
	Comment1	
	L.Aamodt	Export to other sheets
-	Comment2	
		Export to other sheets
	Comment3	
		<ul> <li>Export to other sheets</li> </ul>
	Comment4	
		Export to other sheets
	Page layout description file	
		Browse
		Cancel OK

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.

*	Choose Symbol (484 i	tems loaded) – +	×
- Recent         - aawwu         Device	ly Used –	No symbol selected	
Select wit	h Browser	X Cancel V OK	

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):

ż.	Choose Symbol (72 items loaded)		- • ×
S Filter			
Recently Used	1		
▼ aawwu			
+3.3V	Power symbol creates a global label		
+3V3	Power symbol creates a global label		
+5V	Power symbol creates a global label		
25LCxxx	SPI Serial EEPROM, DIP-8/SOIC-8/TS	No symbol a	alastad
AD8001AN		No symbol s	selected
AT25xxx			
BR25Sxxx			
С	Unpolarized capacitor		
CAT250xxx			
CP1	Polarised capacitor		
Conn_01x01	Generic connector, single row, 01x0		
			1
Select with Browser		😮 Cancel	ОК

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).

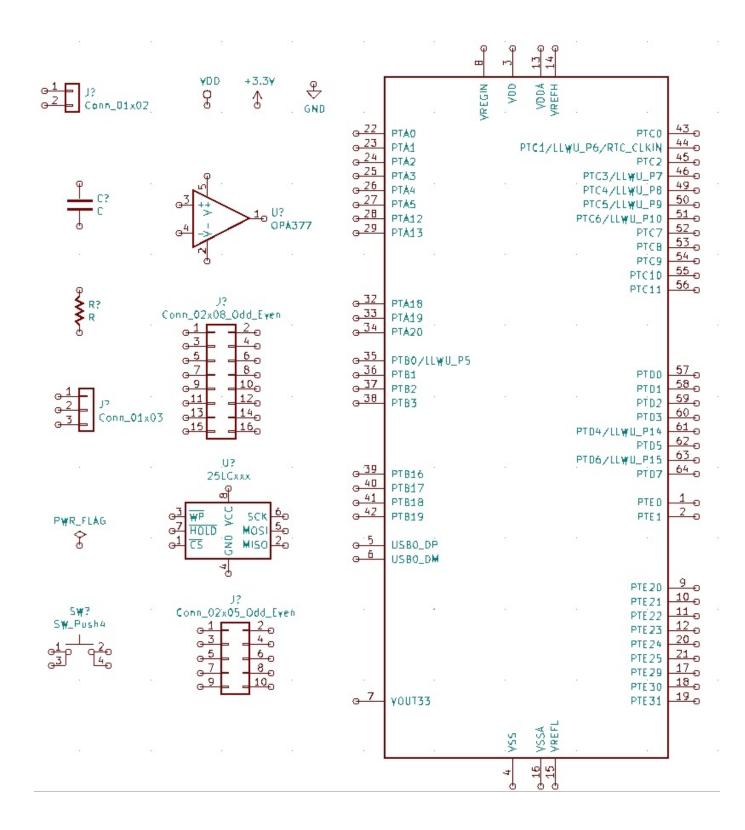
There are many hot keys. Get a list by pressing? A partial list is:

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 Jndo Ctrl+Z	S.	Hotkey	s List	×
Zoom InF1Zoom OutF2Zoom RedrawF3Zoom CenterF4Fit on ScreenHomeReset Local CoordinatesSpaceEdit ItemEDelete ItemDelRotate ItemRDrag ItemGJndoCtrl+ZRedoCtrl+YMouse Left ClickReturnMouse Left Double ClickEndSave SchematicCtrl+SLoad SchematicCtrl+FFind Next ItemF5Find Next ItemF5Find and ReplaceCtrl+Alt+FRepeat Last ItemInsMove Block -> Drag BlockTabSave Schematic ItemMCopy Component or LabelCAdd ComponentAAdd PowerPMirror XXMirror YYOrient Normal ComponentNEdit Component ReferenceU	Hotkey	's List		
Zoom InF1Zoom OutF2Zoom RedrawF3Zoom CenterF4Fit on ScreenHomeReset Local CoordinatesSpaceEdit ItemEDelete ItemDelRotate ItemRDrag ItemGJndoCtrl+ZRedoCtrl+YMouse Left ClickReturnMouse Left Double ClickEndSave SchematicCtrl+SLoad SchematicCtrl+FFind Next ItemF5Find Next ItemF5Find and ReplaceCtrl+Alt+FRepeat Last ItemInsMove Block -> Drag BlockTabSave Schematic ItemMCopy Component or LabelCAdd ComponentAAdd PowerPMirror XXMirror YYOrient Normal ComponentNEdit Component ReferenceU	Help (this v	window)	?	
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 Jndo Ctrl+Z Redo Ctrl+Y Mouse Left Click Return Mouse Left Double Click End Save Schematic Ctrl+S Load Schematic Ctrl+F Find Next Item F5 Find Next Item F5 Find and Replace Ctrl+Alt+F Repeat Last Item Ins Move Block -> Drag Block Tab Save Schematic Item M Copy Component or Label C Add Component A Add Power P Mirror X X Mirror Y Y Drient Normal Component N Edit Component Reference II	Zoom In		F1	
Zoom Center F4 Fit on Screen Home Reset Local Coordinates Space Edit Item E Delete Item Del Rotate Item G Jndo Ctrl+Z Redo Ctrl+Y Mouse Left Click Return Mouse Left Double Click End Save Schematic Ctrl+S Load Schematic Ctrl+F Find Next Item F5 Find Next Item F5 Find and Replace Ctrl+Alt+F Repeat Last Item Ins Move Block -> Drag Block Tab Save Schematic Item M Copy Component or Label C Add Component A Add Power P Mirror X X Mirror Y Y Drient Normal Component N Edit Component Reference U	Zoom Out		F2	
Fit on Screen Home Reset Local Coordinates Space Edit Item E Delete Item Del Rotate Item G Jndo Ctrl+Z Redo Ctrl+Y Mouse Left Click Return Mouse Left Double Click End Save Schematic Ctrl+S Load Schematic Ctrl+F Find Next Item F5 Find Next Item F5 Find and Replace Ctrl+Alt+F Repeat Last Item Ins Move Block -> Drag Block Tab Save Schematic Item M Copy Component or Label C Add Component A Add Power P Mirror X X Mirror Y Y Drient Normal Component N Edit Component Reference U	Zoom Redr	aw	F3	
Reset Local Coordinates       Space         Edit Item       E         Delete Item       Del         Rotate Item       R         Drag Item       G         Jndo       Ctrl+Z         Redo       Ctrl+Y         Mouse Left Click       Return         Mouse Left Double Click       End         Save Schematic       Ctrl+S         Load Schematic       Ctrl+F         Find Item       F5         Find Next Item       F5         Find and Replace       Ctrl+Alt+F         Repeat Last Item       Ins         Move Block -> Drag Block       Tab         Save Schematic Item       M         Copy Component or Label       C         Add Component       A         Add Power       P         Mirror X       X         Mirror Y       Y         Drient Normal Component       N         Edit Component Reference       U	Zoom Cent	ter	F4	
Edit Item       E         Delete Item       Del         Rotate Item       R         Drag Item       G         Jndo       Ctrl+Z         Redo       Ctrl+Y         Mouse Left Click       Return         Mouse Left Double Click       End         Save Schematic       Ctrl+S         Load Schematic       Ctrl+F         Find Item       F5         Find Next Item       F5         Find and Replace       Ctrl+Alt+F         Repeat Last Item       Ins         Move Block       Ctrl+C         Move Schematic Item       M         Copy Component or Label       C         Add Component       A         Add Power       P         Mirror X       X         Mirror Y       Y         Drient Normal Component       N         Edit Component Value       V	Fit on Scre	en	Home	
Delete Item Del Rotate Item R Rotate Item G Jndo Ctrl+Z Redo Ctrl+Y Mouse Left Click Return Mouse Left Double Click End Save Schematic Ctrl+S Load Schematic Ctrl+L Find Item F5 Find Next Item F5 Find Next Item F5 Find Next 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 Drient Normal Component N Edit Component Reference U	Reset Loca	l Coordinates	Space	
Rotate Item       R         Rotate Item       G         Orag Item       G         Jndo       Ctrl+Z         Redo       Ctrl+Y         Mouse Left Click       Return         Mouse Left Double Click       End         Save Schematic       Ctrl+S         Load Schematic       Ctrl+L         Sind Item       Ctrl+F         Find Item       F5         Find Next Item       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 Reference       U	Edit Item		E	
Rotate Item       R         Orag 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         Eind Item       Ctrl+F         Find Next Item       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         Drient Normal Component       N         Edit Component Value       V	Delete Iten	n	Del	=
Jndo       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 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         Drient Normal Component       N         Edit Component Reference       U	Rotate Iten	n	R	
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 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         Drient Normal Component       N         Edit Component Reference       U	Drag Item		G	
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 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         Drient Normal Component       N         Edit Component Beference       U	Undo		Ctrl+Z	
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         Drient Normal Component       N         Edit Component Reference       U	Redo		Ctrl+Y	
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         Drient Normal Component       N         Edit Component Reference       U			Return	
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         Drient Normal Component       N         Edit Component Reference       U			End	
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         Drient Normal Component       N         Edit Component Reference       U			Ctrl+S	
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 Drient Normal Component N Edit Component Reference U		matic		
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 Drient Normal Component N Edit Component Reference U				
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 Drient Normal Component N Edit Component Reference U				
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       Drient Normal Component     N       Edit Component Reference     U				
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 Drient Normal Component N Edit Component Reference U				
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 Drient Normal Component N Edit Component Reference U				
Move Schematic Item M Copy Component or Label C Add Component A Add Power P Mirror X X Mirror Y Y Drient Normal Component N Edit Component Value V		_		
Copy Component or Label C Add Component A Add Power P Mirror X X Mirror Y Y Drient Normal Component N Edit Component Value V Edit Component Reference U				
Add Component A Add Power P Mirror X X Mirror Y Y Drient Normal Component N Edit Component Value V				
Add Power P Mirror X X Mirror Y Y Drient Normal Component N Edit Component Value V Edit Component Reference U		-	-	
Mirror X X Mirror Y Y Drient Normal Component N Edit Component Value V Edit Component Reference				
Mirror Y Y Drient Normal Component N Edit Component Value V Edit Component Reference U				
Drient Normal Component N Edit Component Value V Edit Component Reference U				
Edit Component Reference		mal Component		
Edit Component Reference II				
			-	~
Close				Class
				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)
- 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.
- 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.

- 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:

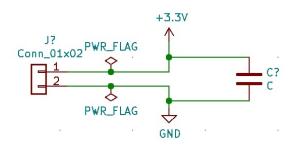
£)		Electrical Rules	Checker	- + >
ERC Option	IS			
ERC Report:		Messages:		
Total:	0	]		
Warnings:	0			
Errors:	0			
Create ER	C file report			
Error list:				

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:

£		Electrical Rules Checker - + ×
ERC Option	าร	
ERC Report:		Messages:
Total:	3	Finished
Warnings:	3	
Errors:	0	
Create EF	C file report	
Error list:		
• @ (8 drive <u>ErrType(3)</u> • @ (1 drive <u>ErrType(3)</u> • @ (1	6.36 mm,83. en (Net 7). : <b>Pin connec</b> :40.97 mm,10 en (Net 11). : <b>Pin connec</b>	ted to some others pins but no pin to drive it 82 mm): Pin 1 (Power input) of component #PWR02 is not ted to some others pins but no pin to drive it 19.22 mm): Pin 8 (Power input) of component U3 is not ted to some others pins but no pin to drive it 12 mm): Pin 14 (Power input) of component U4 is not
		Delete Markers Run Close

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: PWR\_FLAG\_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.

Cvpcb 4	1.0.5 Pi	oject: '/home/larry.aamodt/ee355/demokicad/kicad_	demo2/kicad_demo2.pro' _ 🗆 ×
File Preferences Help			
🖄   🕸   🕵   🜩 🗭	ŧ⇒į <b>ŧ×</b> į		
Air_Coils_SML_NEOSID	1	C1 - C_0805_0R1uF : Capacitors_SMD:C_0805	1 Capacitors_SMD:C_0201
Buttons_Switches_SMD	2	C2 - C_0805_0R1uF : Capacitors_SMD:C_0805	2 Capacitors_SMD:C_0402
Buttons_Switches_THT	3	C3 - C_0805_0R1uF : Capacitors_SMD:C_0805	3 Capacitors_SMD:C_0603
Buzzers_Beepers	4	D1 - BAS70 : T0_SOT_SMD:SOT-23	4 Capacitors_SMD:C_0603_HandSolderi
Capacitors_SMD	5	D2 - BAS70 : T0_SOT_SMD:SOT-23	5 Capacitors_SMD:C_0805
Capacitors_THT	6	P1 - CONN_01X02 :	6 Capacitors_SMD:C_0805_HandSolderi
Capacitors_Tantalum_SMD	7	P2 - CONN_01X02 :	7 Capacitors_SMD:C_1206
Choke_Axial_ThroughHole	8	P3 - CONN_01X02 :	<pre>8 Capacitors_SMD:C_1206_HandSolderi</pre>
Choke_Common-Mode_Wurth	9	P4 - CONN_01X08 :	9 Capacitors_SMD:C_1210
Choke_Radial_ThroughHole	10	R1 - R_0805_100K_5PER : Resistors_SMD:R_0805	10 Capacitors_SMD:C_1210_HandSolderi
Choke_SMD	11	U1 - TLV3202 :	11 Capacitors_SMD:C_1812
Choke_Toroid_ThroughHole	12	U2 - 74HC74 :	12 Capacitors_SMD:C_1812_HandSolderi
Connectors	13	U3 - 25LC160C-I/SN : Housings_SOIC:SOIC-8_3	.9x4.9mm 13 Capacitors_SMD:C_1825
Connectors_Harwin	14	U4 - 74LS08 :	14 Capacitors_SMD:C_1825_HandSolderi
Connectors_Hirose			15 Capacitors_SMD:C_2220
Connectors_JAE			16 Capacitors_SMD:C_2220_HandSolderi
Connectors_JST			17 Capacitors_SMD:C_2225
	<	III	
Components: 14, unassigned:	7	Filter list: C?, C_????_*, C_????, SMD*_c	c, Capacitor* Filtered by key words: 234

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.

£	Netlist	- + ×
Pcbnew Option	OrcadPCB2 CadStar Spice s: ault format	Generate × Cancel Add Plugin
		Remove Plugin
		🗌 Use default netname
Default N	etlist Filename:	
kicad_de	mo2.net	

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:

×
Generate
Close
Help
Add Plugin
Remove Plugin
Edit Plugin File

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