

KiCad Step by Step Tutorial

Copyright © 2006 David Jahshan: kicad at iridec.com.au

Copyright: Please freely copy and distribute (sell or give away) this document in any format. Send any corrections and comments to the document maintainer. You may create a derivative work and distribute it provided that:

1. If it's not a translation: Email a copy of your derivative work to the author.
2. License the derivative work in the spirit of the GPL. Include a copyright notice and at least a pointer to the license used.
3. Give due credit to previous authors and major contributors.

If you're considering making a derived work other than a translation, it's requested that you discuss your plans with the current maintainer.

Disclaimer: While care was taken in preparing this document, there are likely a number of errors in this document. Please let the author know about them. Since this is free documentation, the author will not be held legally responsible for any errors.

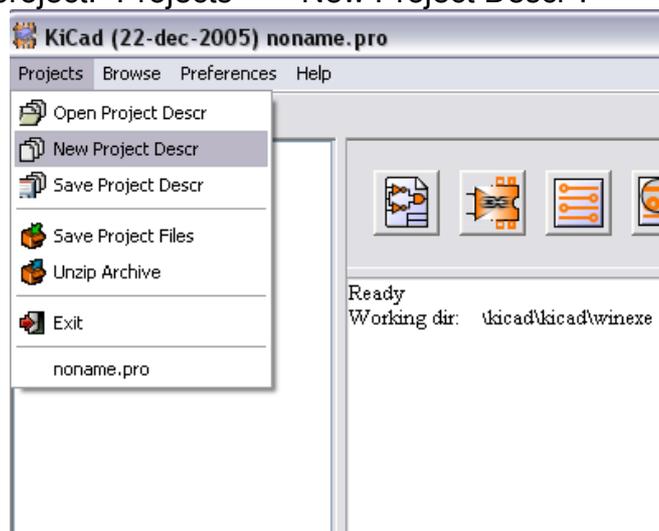
Trademarks: Any brand names should be assumed to be a trademark. Such trademarks belong to their respective owners.

KiCad is a open source (GPL) integrated package for schematic circuit capture and PCB layout.

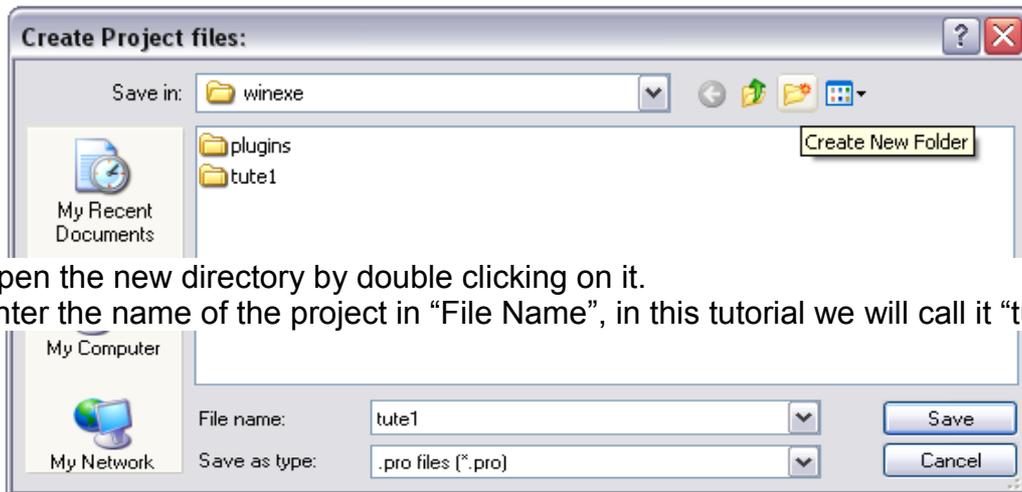
Before you start, you will need an installed copy of KiCad. This tutorial assumes that KiCad is installed at <C:\Kicad>. You can download a copy from <http://iut-tice.ujf-grenoble.fr/cao/>

Installation instructions are available on the web site under Infos:Install

1. Run "KiCad.exe".
2. You are now in the Main Window.
3. Create a new project: "Projects" -> "New Project Descr".



4. Click on "Create New Folder" button, and name the new folder "tute1".

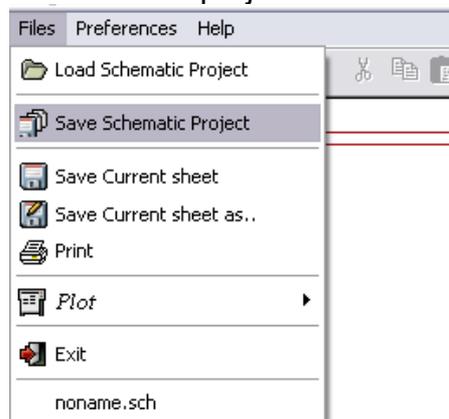


5. Open the new directory by double clicking on it.
6. Enter the name of the project in "File Name", in this tutorial we will call it "tute1".

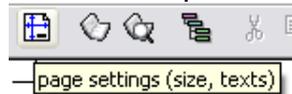
7. Click on "Save". You should notice the project name should change to "tute1".



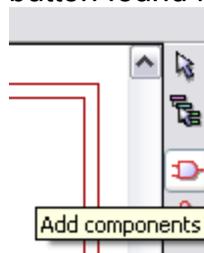
8. Double click on "tute1.sch".
9. An "Infos" window will appear informing you it is a new project. Click "OK".
10. You are now in the "EESchema" window. This window is used for entering schematics.
11. You should first save the schematic project: "Files" -> "Save Schematic Project".



12. Click the "page settings" button on the top toolbar.



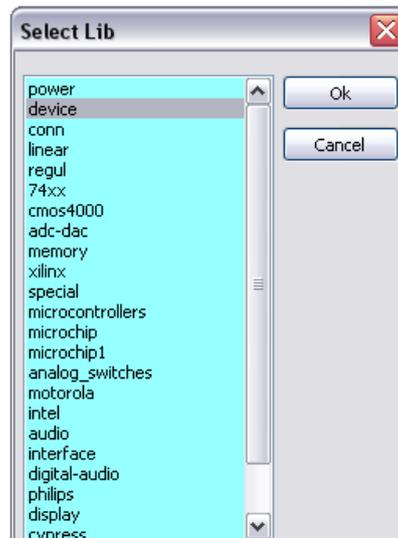
13. Select the "page size" as "A4" and the "Title" as "Tute 1".
14. Click on the "Add components" button found in the right toolbar.



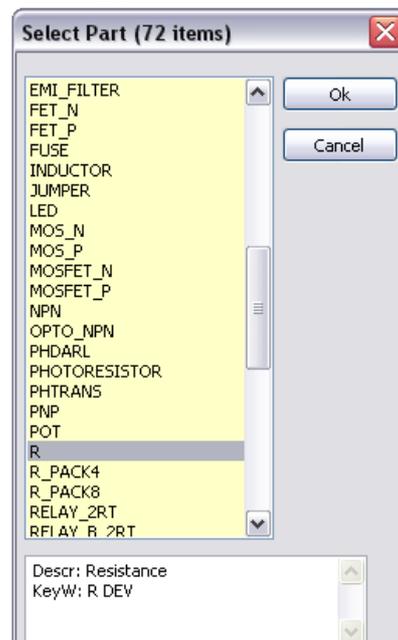
15. Click in the middle of the screen where you want to place your first component.
16. A "Component selection" window will appear.



17. Click on "List All". The "Select Lib" window will appear.



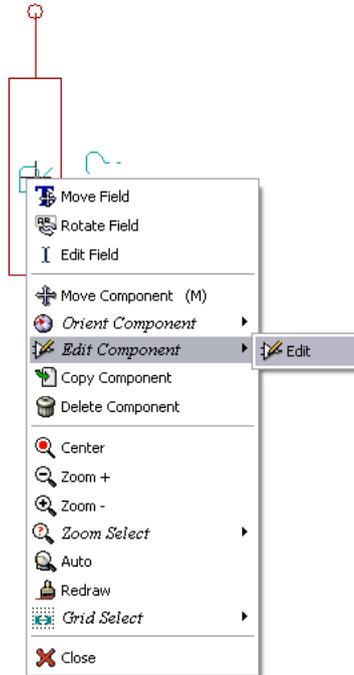
18. Double click on "device".
19. The "Select Part" window will appear.



20. Scroll down and double click on "R".
21. Press 'r' on the keyboard. Notice how the component rotates
22. Place the component in the middle of the sheet by left clicking where you want it.
23. Click on the magnifier twice to zoom in on the component.

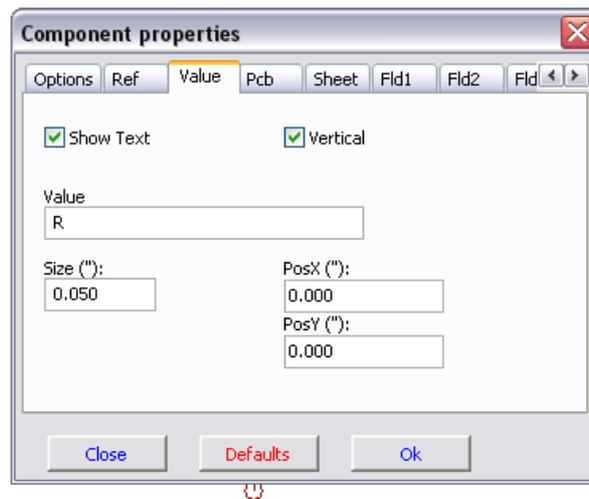


24. Right click in the middle of the component.



25. Select: “Edit Component” -> “Edit”.

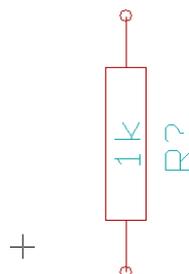
26. The “Component properties” window will appear.



27. Select the “Value” tab.

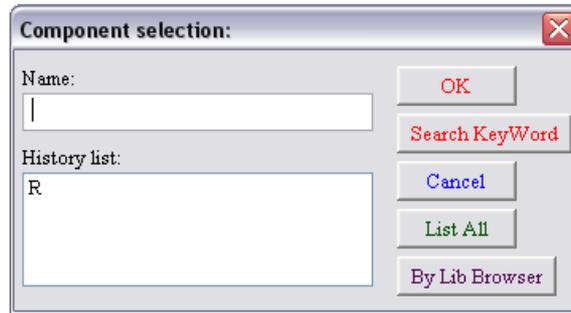
28. Replace the current “Value” “R” with “1k”.

29. Click “OK”.

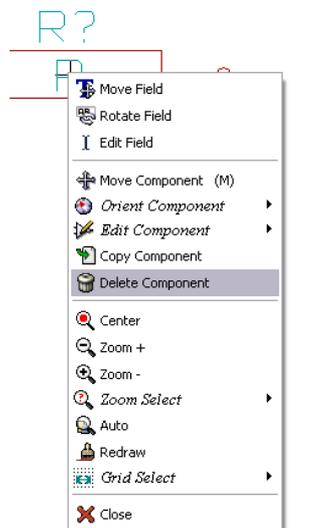


30. The value inside the resistor should now be “1k” .

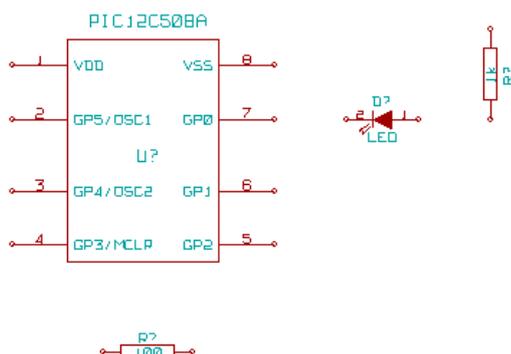
31. Place another resistor by clicking on the place you want the resistor to appear.
32. The “Component selection:” window will appear.
33. The resistor you previously chose is now in your history list appearing as “R”.



34. Click on “R”.
35. Place the resistor on the page
36. Repeat and place a third resistor on the page.
37. Right click on the second resistor

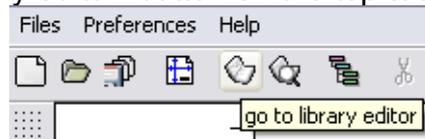


38. Click on “Delete Component”. This should remove the component.
39. Right click on the third resistor. Select “Move Component”.
40. Reposition the component left click to drop.
41. Repeat steps 24 to 27 on the third resistor and replace “R” with “100”
42. Repeat steps 14 to 20, however this time select “microcontrollers” instead of “device” and “PIC12C508A” instead of “R”.
43. Press 'y' and 'x' on the keyboard. Notice how the component is mirrored on it's x and y axis. Press 'y' and 'x' again to return to it's original orientation.
44. Place the component on the page.
45. Repeat steps 14 to 20, however this time choose “device” and “LED”.
46. Organise the components on the page in the following way:

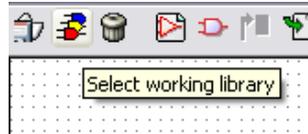


47. We will now add a component to the library.

48. Click on the “go to library editor” button on the top toolbar.



49. This will open the “Libedit” window.



50. Click on the “Select working library” button.

51. In the “select lib” window click on “conn”.

52. Click on the “New part” button.



53. Name the new part “MYCONN3”.

54. Enter the prefix as “J”, and number of parts as “1”.

55. If the warning “has a convert drawing” appears click “yes”.

56. In the middle of the screen the name of the component should appear.

57. Click on the magnifier twice to zoom in on it.

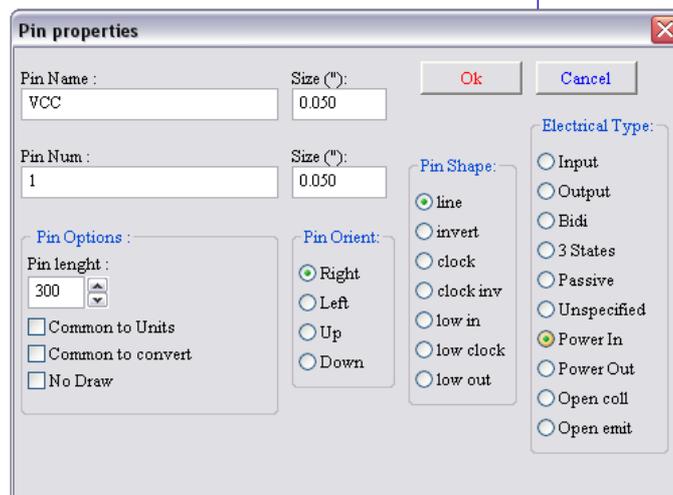
58. Click on the “Add Pins” button on the right toolbar.



59. Left click on the screen where you want the pin.

60. In the “Pin Properties” box enter pin name as “VCC”, enter pin number as “1”.

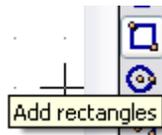
61. Select “Electrical Type” as “Power Out” then click “OK”. Then place the pin by clicking at the location you would like it to appear.



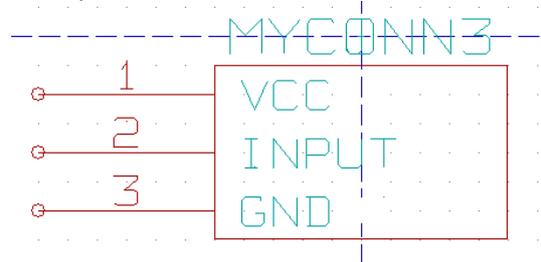
62. Repeat steps 59 to 61, this time “Pin Name” should be “INPUT” and 'Pin Number’ should be “2”. “Electrical Type” should be “Input”.

63. Repeat steps 59 to 61, this time “Pin Name” should be “GND” and “Pin Number” should be “3”. “Electrical Type” should be “Power Out”.

64. Arrange the pins and label as shown below step 65.



65. Click on the “Add rectangle” button. By left clicking and release the button to start a rectangle around the pin names. A second click ends the rectangle (or right click to access the pull up menu)



66. Click on “Save current part into current loaded library (in memory)” on the top toolbar.



67. Click on “Save current loaded library on disk (file update)” on the top tool bar.



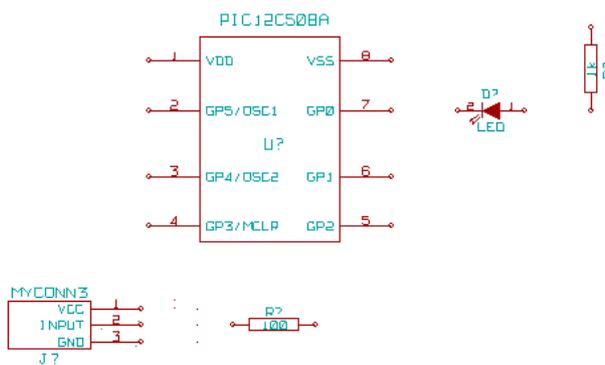
68. Click “yes” on the confirmation message.

69. You can now close the “Libedit” window.

70. Return back to the “EeSchema” window.

71. Repeat steps 14 to 20, however this time choose “conn” and “MYCONN3”.

72. Your newly created part will appear. Choose a location near the second resistor to place this component. Press the 'y' key to mirror it on the y axis.



73. The component identifier “J?” will appear under the “MYCONN3” label. Right click on “J?” and click on “move field”. Reposition “J?” to under the pins.

74. Click on the “Add powers” button on the right toolbar.



75. Click above the pin of the 1k resistor.

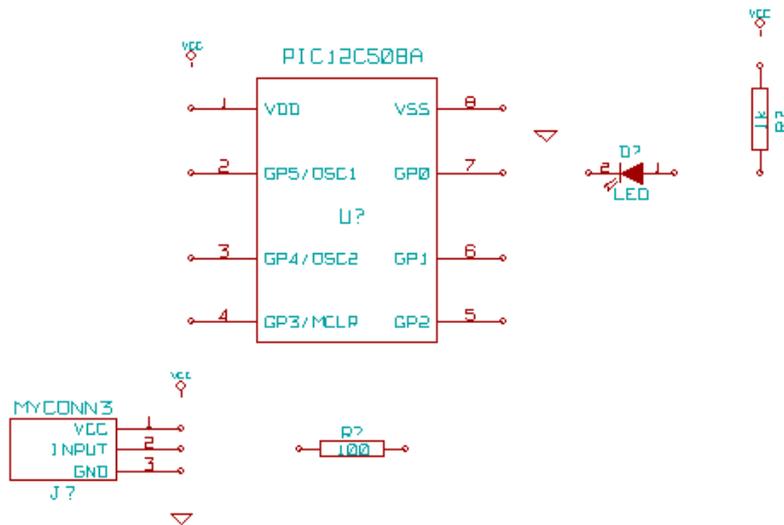
76. In the “Component Selection” click on list all.

77. Scroll down and select “VCC” in the “Select Part” window.

78. Click above the pin of the 1k resistor to place the part.

79. Click above the VDD pin near the microcontroller.

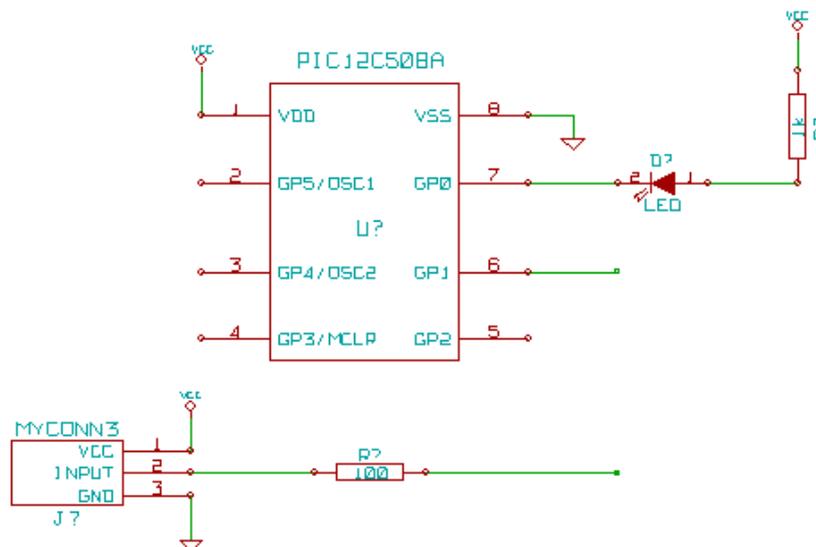
80. In the “Component Selection history” select “VCC” and click again next to the VDD pin.
81. Repeat again and place the VCC pin above the VCC pin of “MYCONN3”.
82. Repeat steps 74 to 76 but select GND this time.
83. Place the GND pin under the GND pin of “MYCONN3”.
84. Place the GND symbol little to the right and below the VSS pin of the microcontroller.



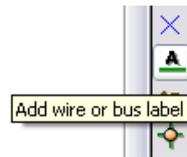
85. Click on the “Add wires” on the right toolbar **Careful not to pick “Add bus” which appears directly beneath but has thicker lines**.



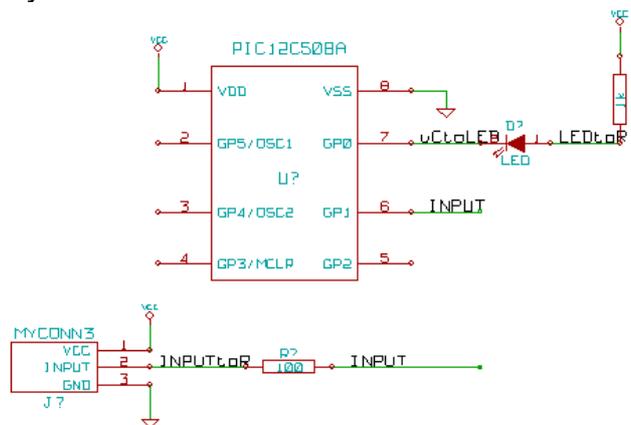
86. Left click on the little circle on the end of pin 7 of the microcontroller and then on the little circle on pin two of the LED.
87. Repeat process to wire up the other components as below.
88. When wiring up the VCC and GND symbols, the wire should touch at the bottom of the VCC symbol, and in the middle top of the GND symbol.



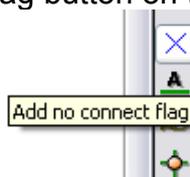
89. Label the nets by clicking on the “Add wire or bus label” button on the right toolbar.



90. Click in the middle of the wire between the microcontroller and the LED.
91. Enter the name "uCtoLED".
92. Click near the circle (little to the right) of pin 7 to place the net name.
93. Name the wire between the resistor and the LED to "LEDtoR".
94. Name the wire between "MYCONN3" to the resistor as "INPUTtoR".
95. Name the line on the right of the 100 ohm resistor as "INPUT".
96. Name the line from pin 6 as "INPUT". This creates an invisible connection between the two pins labelled "INPUT". This is a useful technique when connecting wires in a complex design where drawing the lines would make the drawing very messy.
97. You do not have to label the VCC and GND lines, the labels are implied from the power objects they are connected to.



98. The program automatically checks for errors therefore any wires that are not connected may generate a warning. To avoid these warnings you can instruct the program that the unconnected wires are deliberate.
99. Click on the "Add no connect flag" button on the right toolbar.



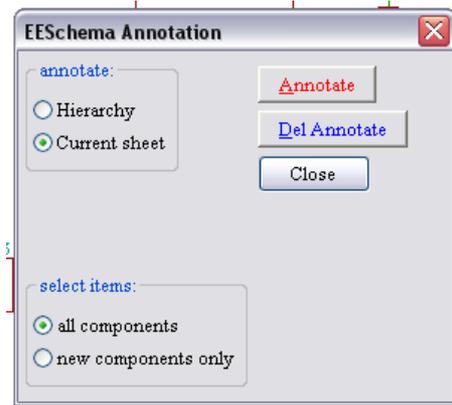
100. Click on the little circle at the end of lines 2, 3, 4 and 5.
101. To add comments on the schematic use the "Add graphics text (comment)" on the right toolbar .



102. The components now need to be given unique identifiers. To do this click on "Schematic Annotation" button.



103. In "EESchema Annotation" select "Current Sheet" and "all components".



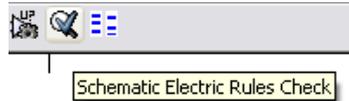
104. Click on “Annotate”.

105. Click “yes” for the confirmation message.

106. Notice how all the “?” on the components have been replaced with a number.

Each identifier is unique. In our example “R1”, “R2”, “U1”, “D1” and “J1”.

107. Click on the “Schematic Electric Rules Check” button. Push the “Test ERC” button.



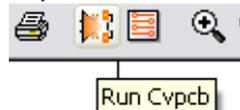
108. This will generate a report to inform you of any errors or warnings such as wires being disconnected. You should have 0 Errors and 0 Warnings. A small green arrow will appear in the location of the error if you have made a mistake. Check “Write erc report” and press the “Test ERC” button again to receive more information about the errors.

109. Click on “Netlist generation” on the top toolbar.



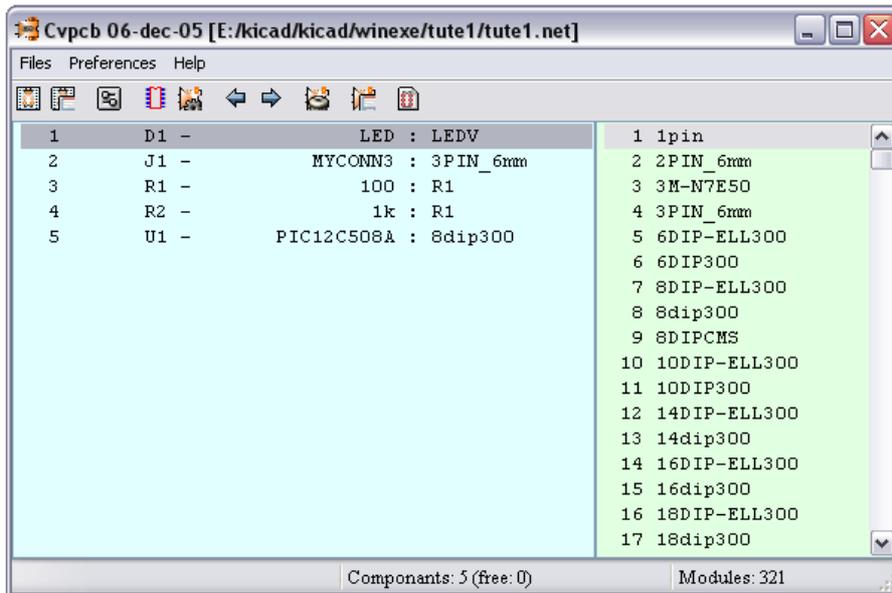
110. Click “Netlist” then on “save” to the default file name.

111. Click on “Run Cvp pcb” on the top toolbar.



112. Cvp pcb permits you to link footprints to components.

113. In the light blue screen select “D1” and scroll down in the light green screen to “LEDV” and double click on it.



114. For "J1" select the "3PIN_6mm" footprint.

115. For "R1" and "R2" select the "R1" footprint from the light green screen.

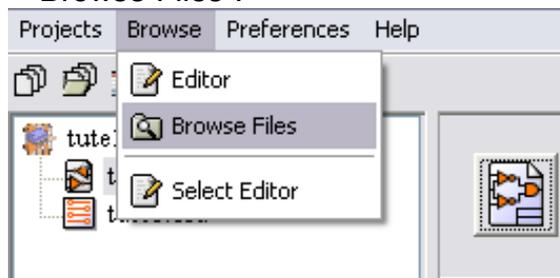
116. Select 8dip300 for "U1".

117. Click on "files" -> "Save netlist". The default "tute1.net" is fine therefore click save.

118. Save the project by clicking on "files" -> "Save Schematic Project".

119. Switch to KiCad main window.

120. Select "Browse" -> "Browse Files".



121. If an error message appears, choose your text browser. Most computers have one at "c:\windows\notepad.exe".

122. Select the "tute1.net" file. This will open your netlist file. It describes which components and which pins are connected to which pins.

123. Now return back to the "EeSchema" window.

124. To create a bill of materials click on the "Bill of materials" button on the top toolbar.



125. Click on "Create List" and then on "Save".

126. To view the file repeat step 120 and select "tute1.lst".

127. Now click on the "Run Pcbnew" button on the top toolbar.

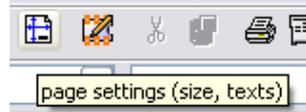


128. The "Pcbnew" window will open.

129. Click "OK" on the error message for the file not existing.

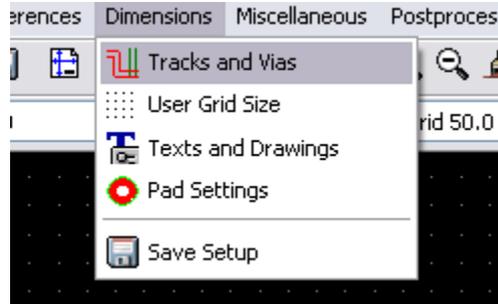
130. Click on "files" -> "Save board".

131. Click on “page settings” button on the top toolbar.



132. Select “paper size” as “A4” and enter “title” as “Tute 1”.

133. Click on “Dimensions” -> “Tracks and Vias”.



134. Set the settings so that they correspond to your PCB manufacturing capabilities. (consult your PCB manufacturer for this information.) For our example increase clearance to 0.0150”.

135. Click on the “Read Netlist” button on the top toolbar.



136. Click the “Select” button select “tute1.net” and click on “open” and then click the “Read” button. Then click the “Close” button.

137. The components will be placed in the top left hand corner just above the page, scroll up to see the components.

138. Right click on a component select “move component” and position it in the middle of the page.

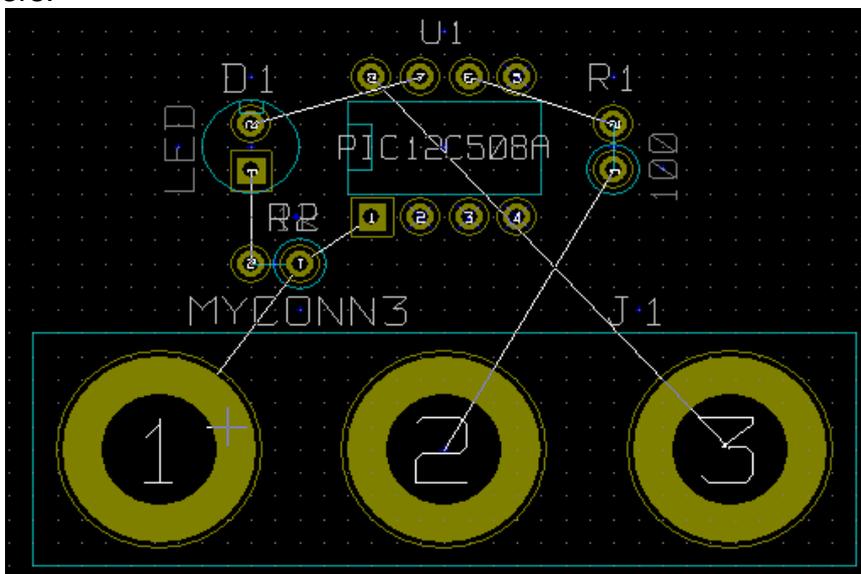
139. Repeat previous step till all the components are in the middle of the page.

140. Make sure that the “General ratsnest not show” button is on.



141. This will display the ratsnest, which is a set of lines showing which pin connects to which other pin.

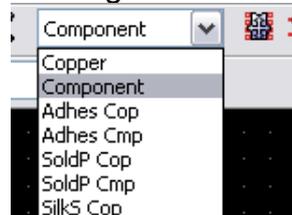
142. Move the components around till the ratsnest until you minimise the number of crossovers.



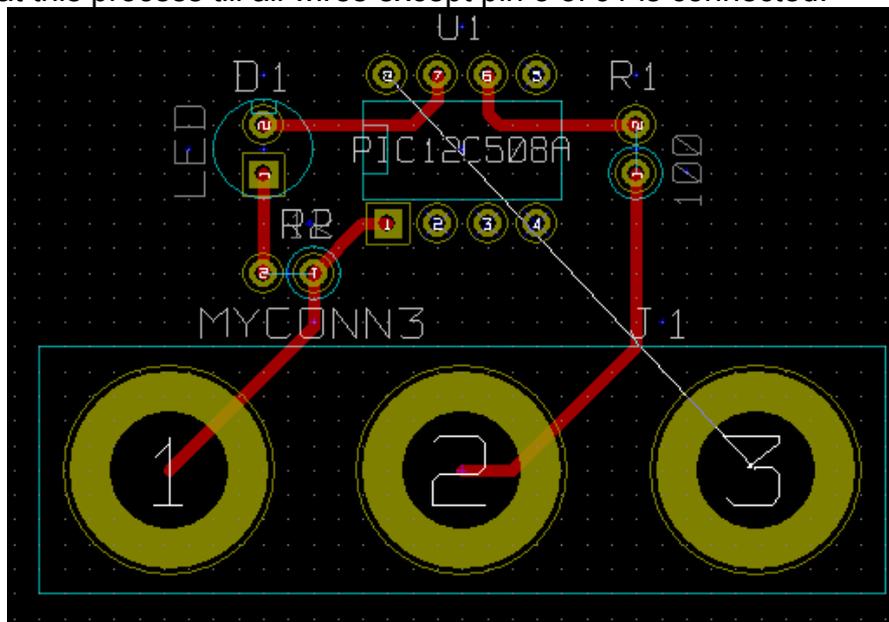
- 143.If the ratsnest disappears or the screen gets messy right click and click “redraw”.
- 144.Now we will connect up all except the ground wires on the “component side” (top layer).
- 145.Click on the “Add Tracks an vias” button on the right toolbar.



- 146.Select “Component” out of the drag down menu on the top toolbar.



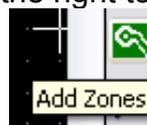
- 147.Click in the middle of pin 1 of “J1” and run a track to the pad “R2”.
- 148.Repeat this process till all wires except pin 3 of J1 is connected.



- 149.In the drag down menu on the top toolbar select Copper (bottom layer).
- 150.Click on “Add tracks and vias” button (step 145).
- 151.Draw a track between pin 3 of J1 and pin 8 of U1.
- 152.Click on “Net highlight” button on the right toolbar.

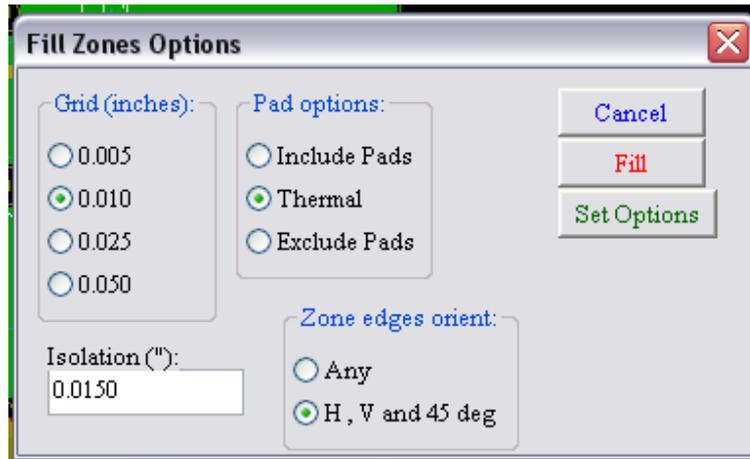


- 153.Click on pin 3 of J1. It should turn yellow.
- 154.Click on “Add Zones” button on the right toolbar.

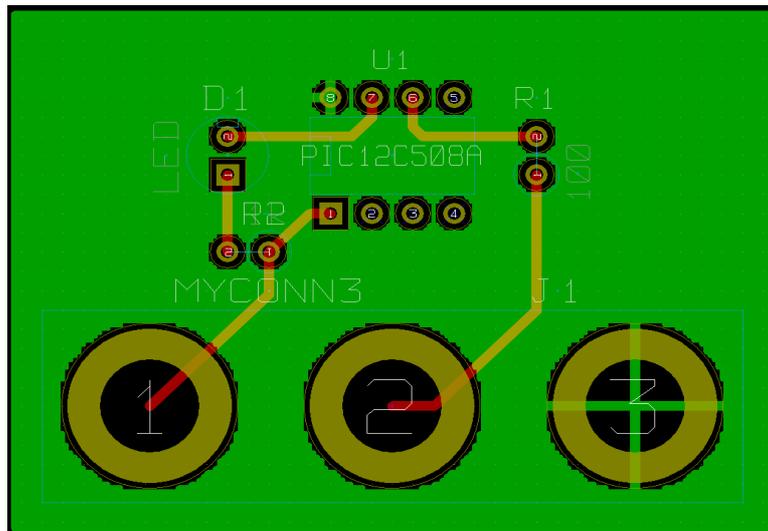


- 155.Trace around the outline of the board.
- 156.Right click inside the area you have just traced.
- 157.Click on “Fill Zones”.
- 158.Select “Grid” “0.010”, “Pad options:” “Thermal”, “Zone edges orient:” “H,V” and

then click on "Fill".

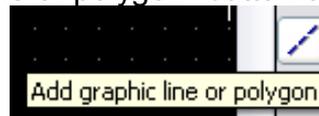


159. Your board should look like this.



160. Now select "Edges Pcb" from the drag down menu in the top toolbar.

161. Select the "Add graphic line or polygon" button on the right toolbar.



162. Trace around the edge of the board but remember to leave a small gap between the edge of the green and the edge of the PCB.

163. Run a design rules check by clicking on "Pcb Design Rules Check".



164. Click on "Test DRC". There should be no errors.

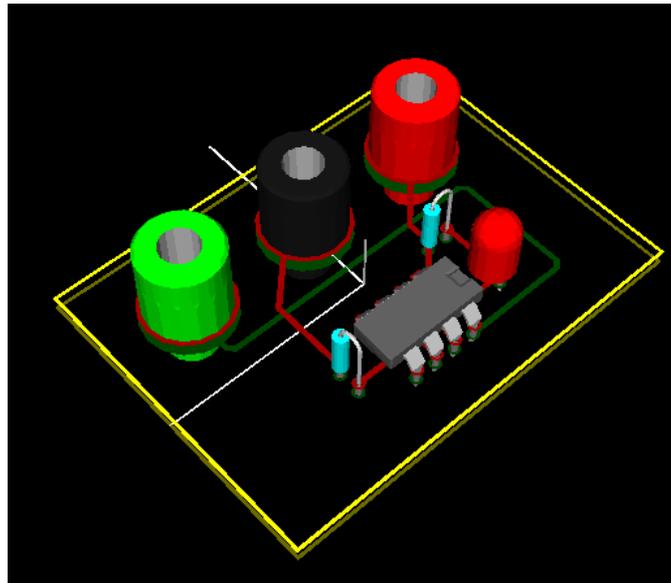
165. Click on "List Unconn". There should be no unconnected.

166. Save your file by clicking on "files" -> "Save board".

167. To view your board in 3d click on "3D Display" -> "3D Display".



168. You can drag your mouse around to rotate the PCB.



169. Your board is complete, to send it off to a manufacturer you will need to generate a GERBER file.

170. Click on “files” -> “plot”.

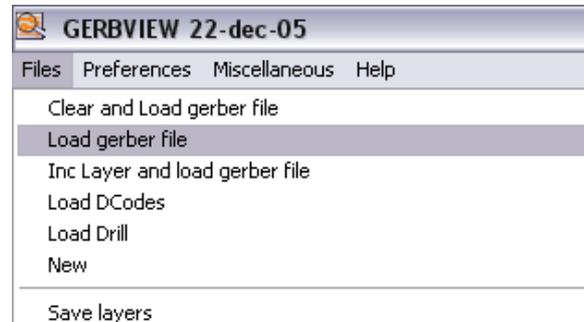
171. Select GERBER as the “plot format” and click on plot.

172. To view GERBER files go to the main KiCad window.

173. Click on the “GerbView” button.



174. Click on “files” -> “Load GERBER file”.

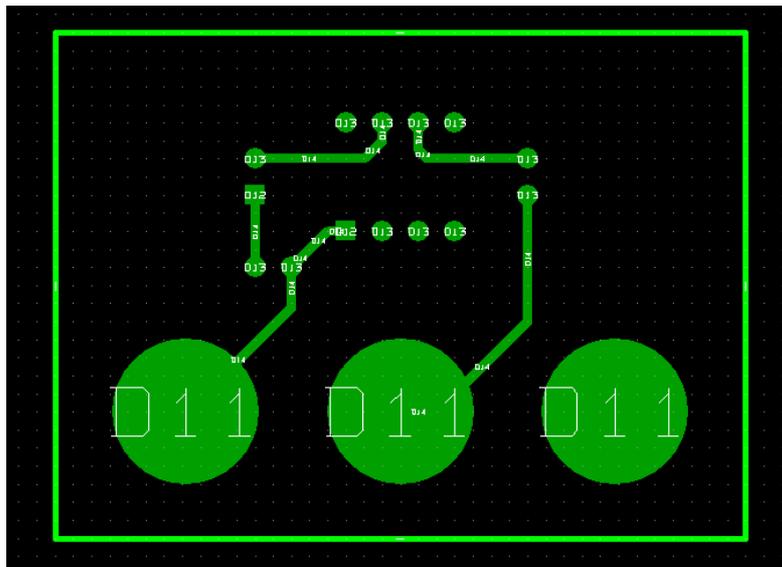


175. Select the file named “tute1_Copper.pho” and then on “open”.

176. On the drag down menu select “Layer2”.

177. Repeat steps 174 and 175 but this time load “tute1_component.pho”.

178. Repeat steps 176 but choose “Layer3” then steps 174 and 175 but this time load “tute1_SlkSCmp.pho”.

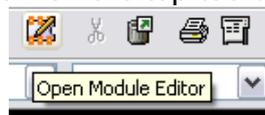


179. This way you can examine the layers that will be sent to production.

There is an extensive footprint library with KiCad, however on occasion you might find that the footprint that you need is not in a KiCad library. Follows are some steps for creating a surface mount footprint in KiCad.

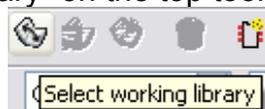
180. To create a new PCB footprint switch back to “PCBnew”.

181. Click on “Open Module Editor” on the top toolbar.



182. This will open the “Module Editor”.

183. Click on “select working library” on the top toolbar.



184. For this exercise select the “connect” library.

185. Click the “New Module” button on the top toolbar.



186. Enter “MYCONN3” as the “module reference”.

187. In the middle of the screen a “MYCONN3” label will appear.

188. Under the label will be “VAL**”.

189. Right click on “MYCONN3” and move above “VAL**”.

190. Right click on “VAL**”, select “Edit Text Mod” and rename it to “SMD”.

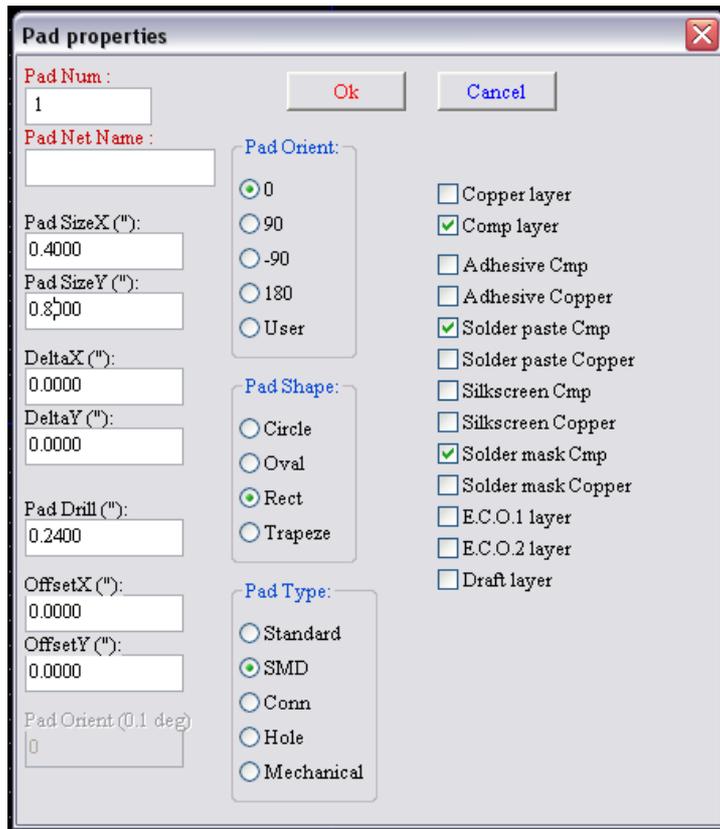
191. Check the “no display” box.

192. Select the “Add Pads” on the right toolbar.

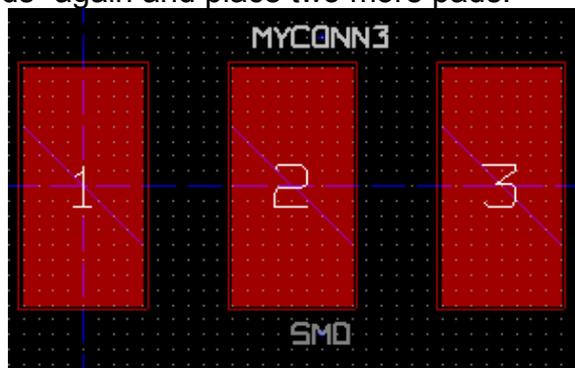


193. Click on the screen to place the pad.

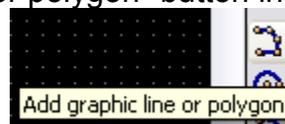
194. Right click on the new pad and click “edit”.



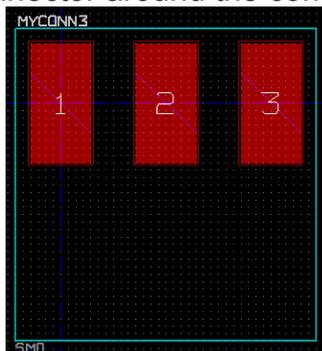
195. Set the "Pad Num" to "1", "Pad Size X" to "0.4", "Pad Size Y" to "0.8", "Pad Shape" to "Rect" and "Pad Type" to "SMD". Click "Ok".
196. Click on "Add Pads" again and place two more pads.



197. Move the "MYCONN3" and "SMD" labels out of the way so it looks like above.
198. Click on "Add graphic line or polygon" button in the right toolbar.



199. Draw an outline of the connector around the component.



200. Click on "Save Module in working directory" on the top toolbar.



201. You can now return to PCB new and click on “Add modules” button on the right toolbar.



202. Click in the screen, and the module name window will pop-up.

203. Select the module “MYCONN3” and place it on your PCB design.

This has been a quick tutorial on most of the features in KiCad. For more detailed instructions there is a detailed help file which can be accessed through all KiCad modules. Click on help -> help.