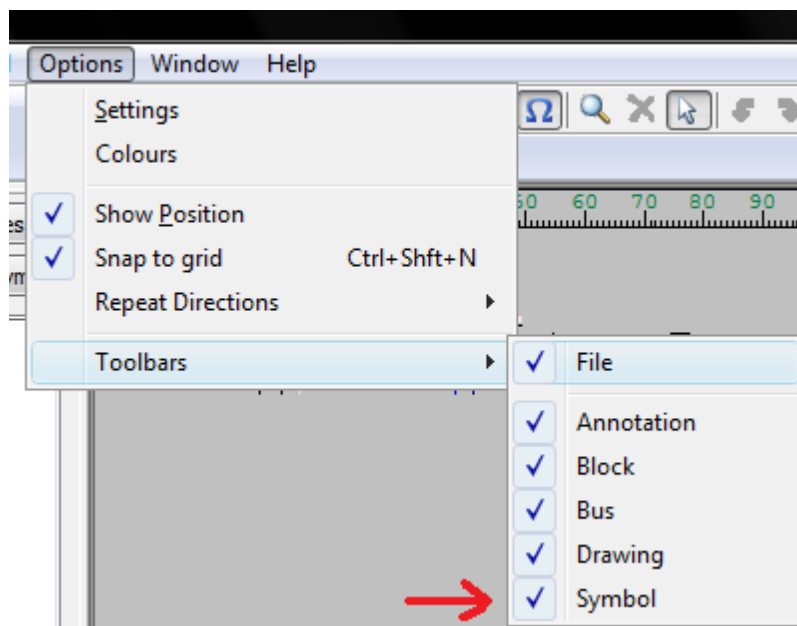
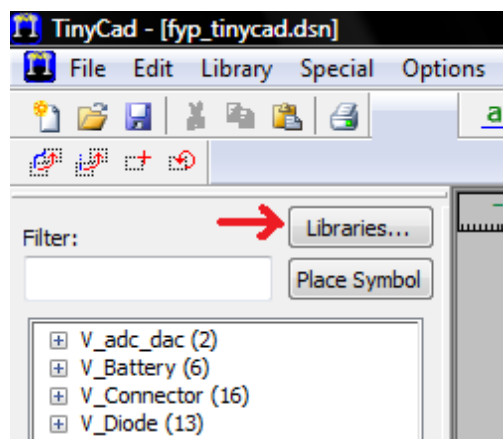


Using TinyCAD with VeeCAD

Show TinyCAD Symbol Window

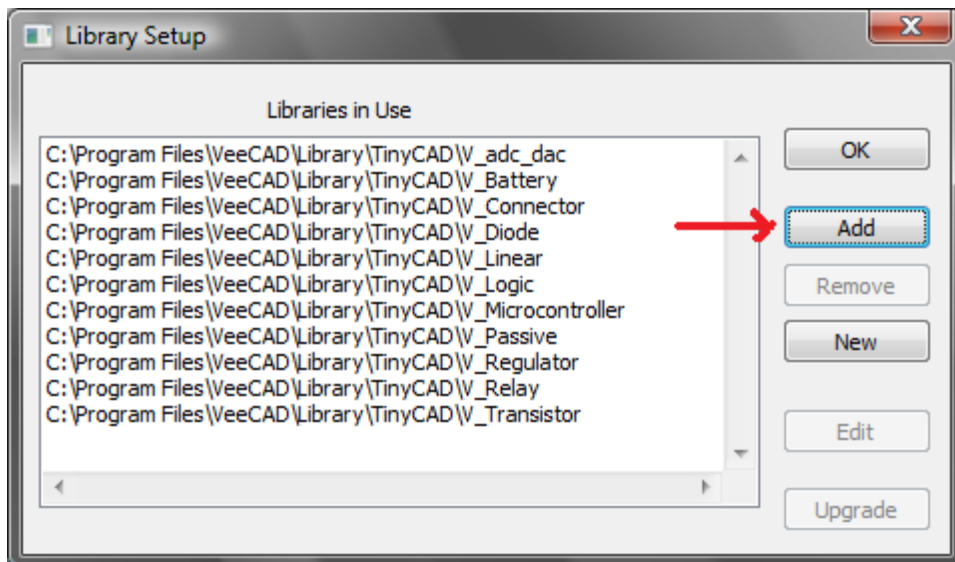


Click the Libraries... Button

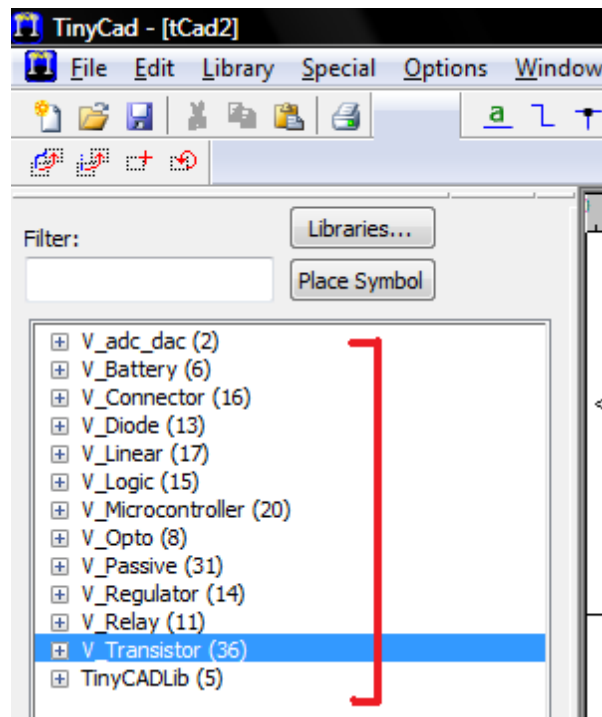


Add the V_xxx Libraries to TinyCAD

Click Add and select one of the TinyCAD *.TCLib symbol library files supplied with VeeCAD. This is typically installed at c:\program files\VeeCAD\Libraries\TinyCAD. Repeat for all symbol libraries.

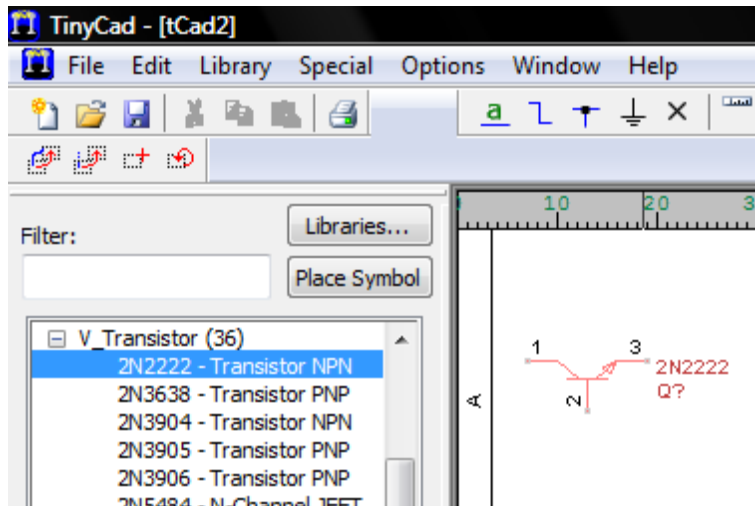


View the V_xxx Libraries in the Symbol Window



Place a Component from the V_Transistor Library

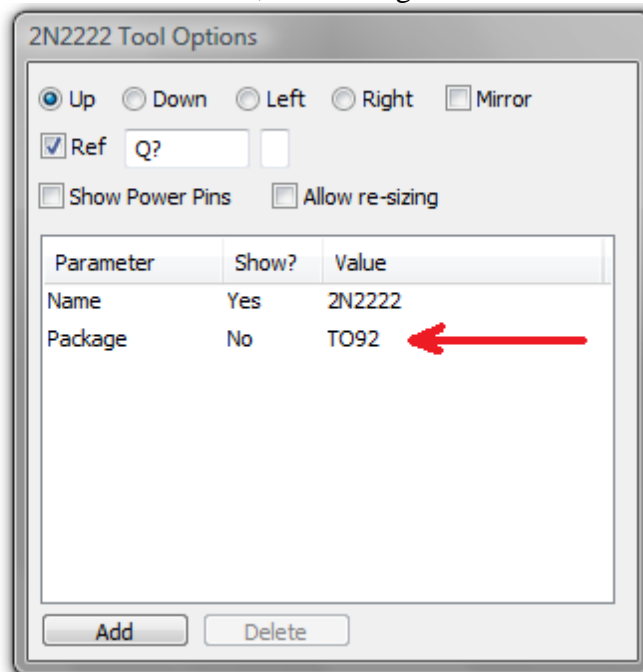
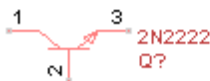
Click the '+' beside the V_Transistor library, then double-click the 2N2222 transistor.



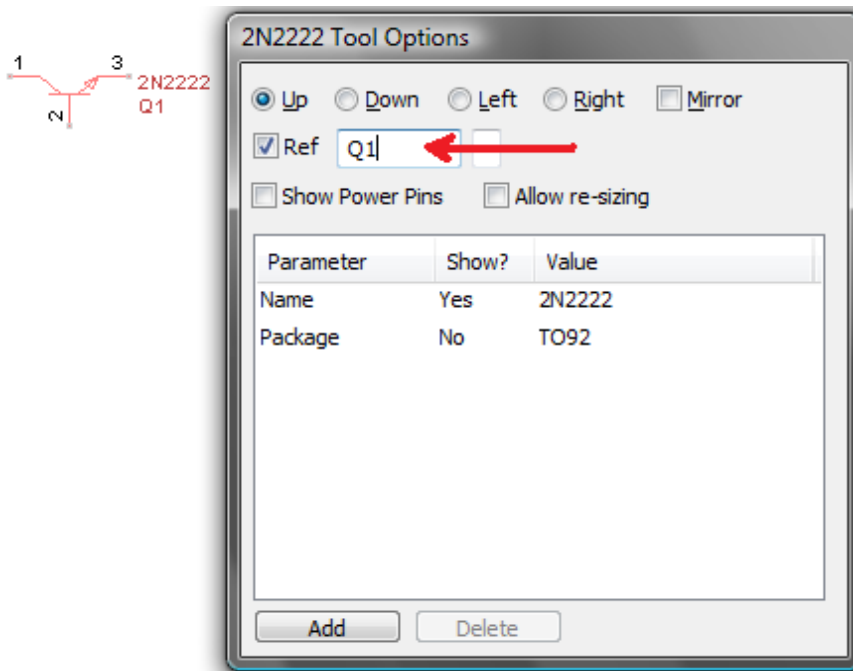
Inspect the Package Attribute

Click on the component you have placed.

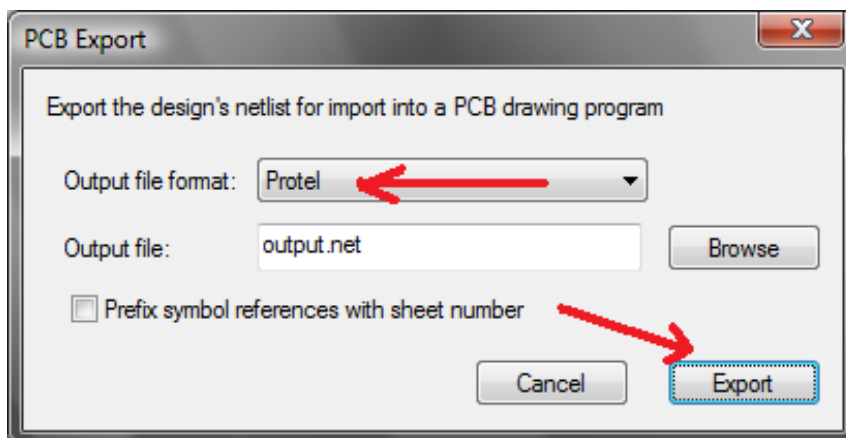
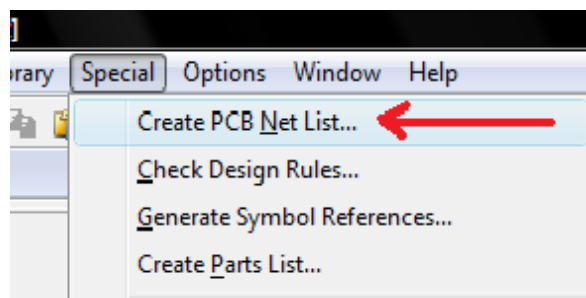
In the Tool window, inspect the Package Attribute. This value will show VeeCAD what shape to display from its library of outlines. For this transistor, the Package is “TO92”.



Set the Reference for Your Component



Export Your Netlist

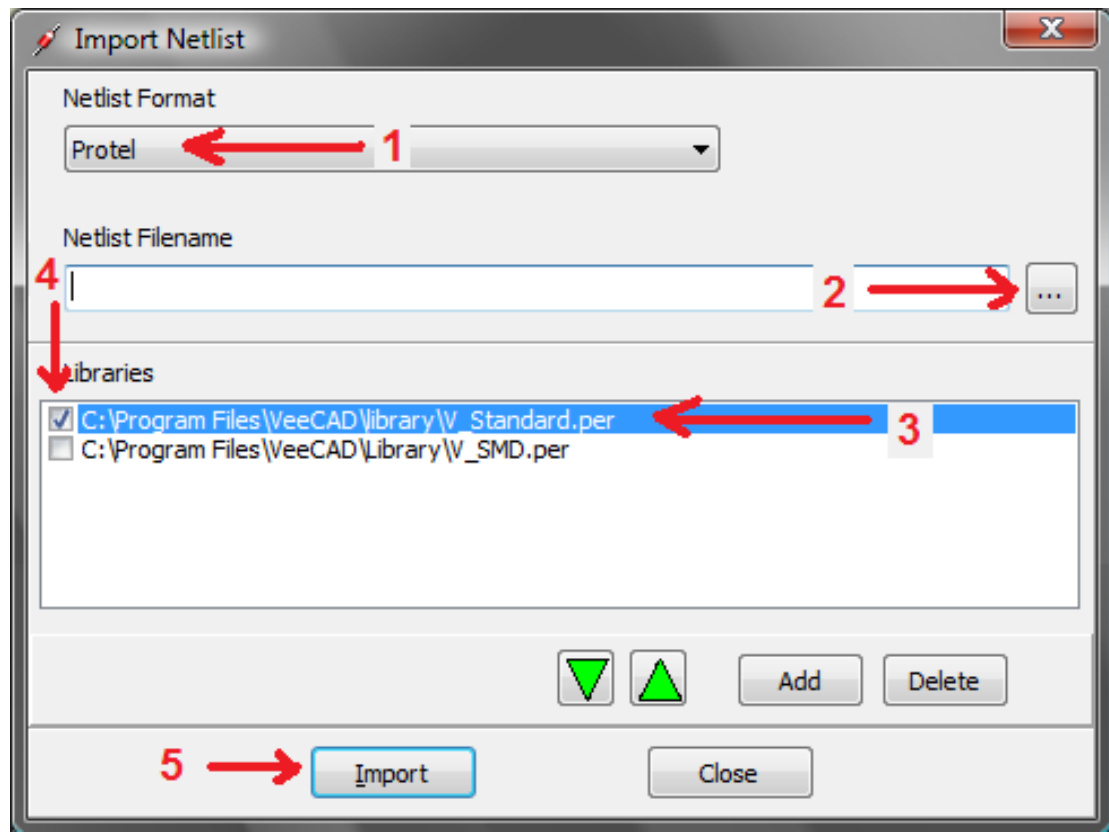


Import Netlist into VeeCAD

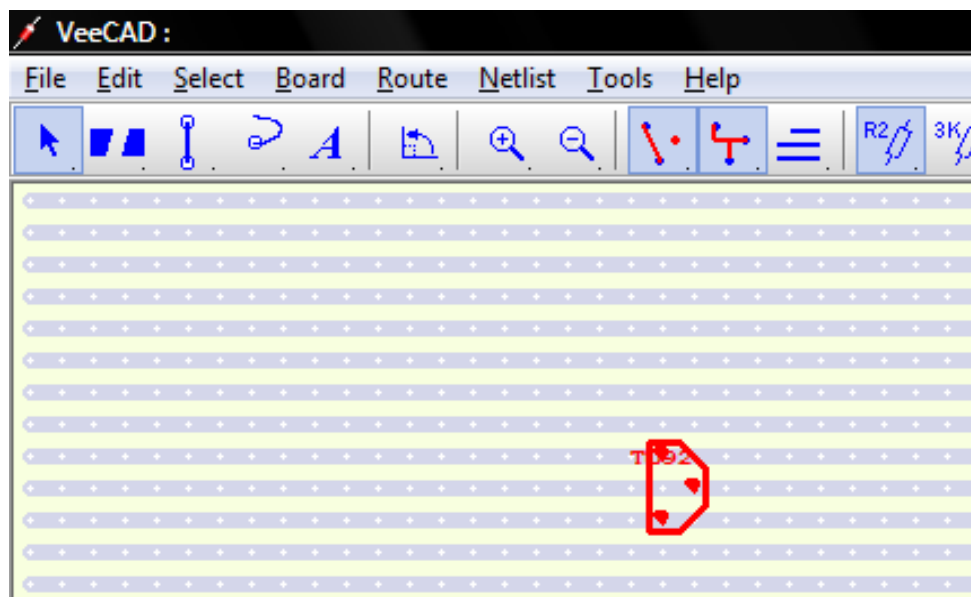
Start VeeCAD and on the menu select File -> New. On the VeeCAD menu select Netlist -> Import.

1. Ensure Protel format is selected.
2. Click the "...” button and select the netlist file you exported from TinyCAD.

3. Ensure that the V_Standard library is available.
4. Ensure that the V_Standard library is checked.
5. Click Import.



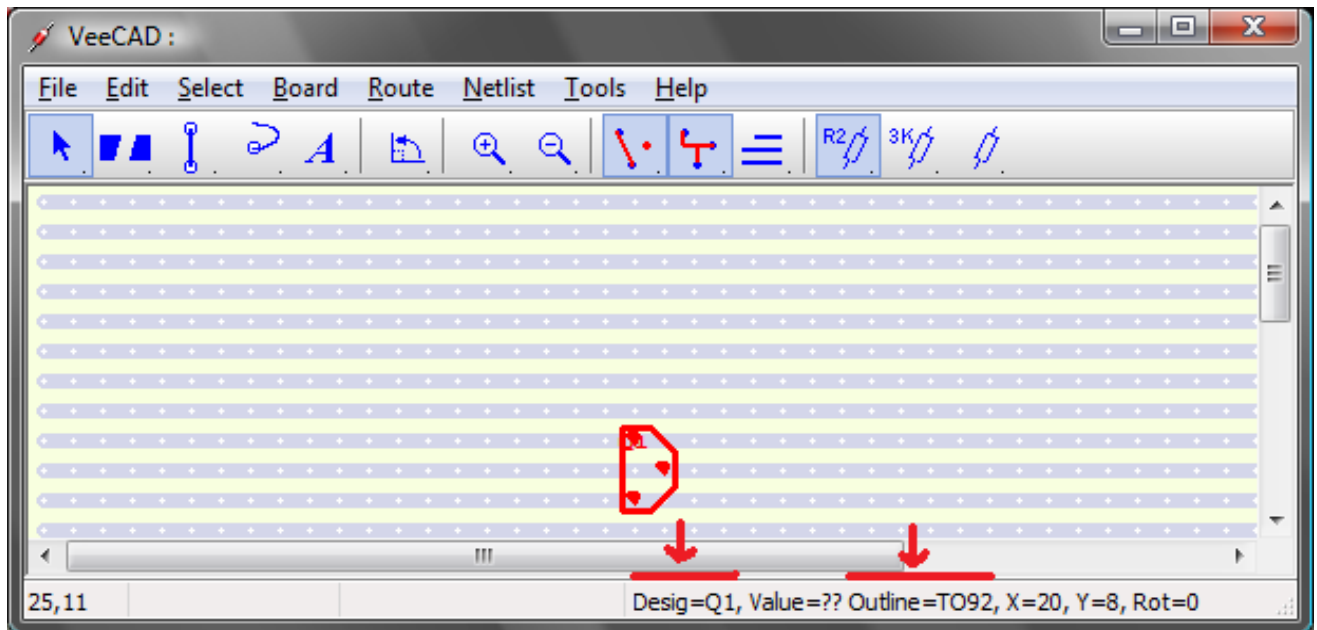
VeeCAD Will Show Your Component



Inspect the Transistor Information

1. Click on the transistor
2. At the bottom of the screen is the Status Bar.

3. Notice that Desig=Q1. This is the same as the “Ref” attribute in TinyCAD.
4. Notice that Outline=TO92. This is the same as the “Package” attribute in TinyCAD.



Now Extend Your Schematic in TinyCAD

1. Add more symbols from the V_xx libraries.
2. Add wires between the symbol pins.
3. Export your netlist.
4. Import your netlist into VeeCAD and complete the layout.