BASIC ELECTRONICS LAB MANUAL

ELECTRONICS ENGINEERING PRACTICES

Prepared By

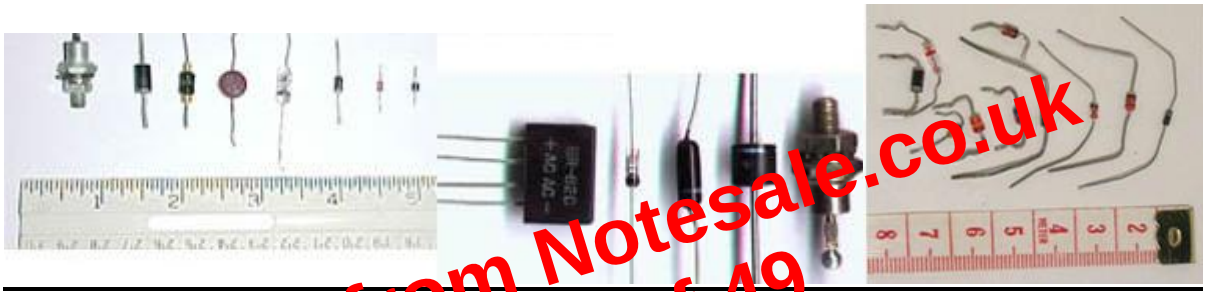
Denmark Y. Gabawa

SEMICONDUCTORS

DIODES

In electronics, a diode is a two-terminal electronic component with asymmetric conductance, it has low (ideally zero) resistance to current flow in one direction, and high (ideally infinite) resistance in the other.

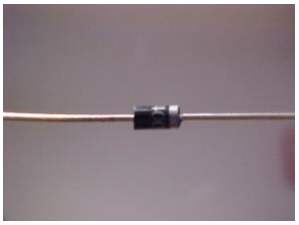
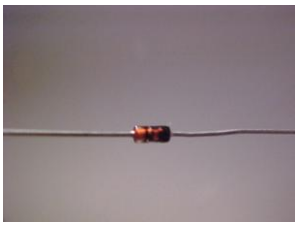
Semiconductors, such as Diodes (typically marked with an “D” on a circuit board).



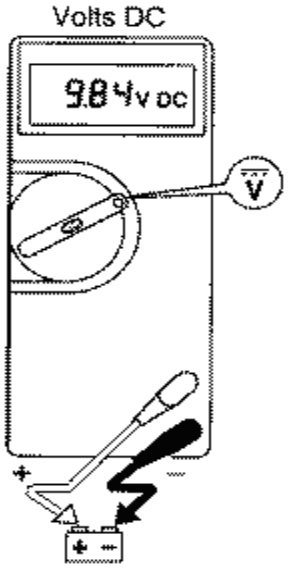
TRANSISTORS

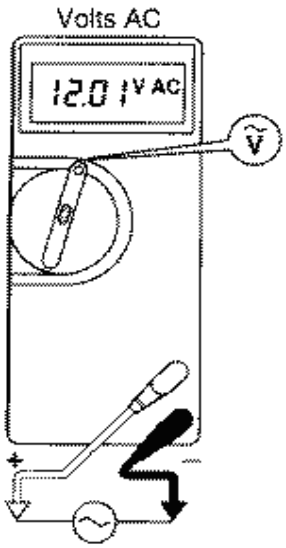
A transistor is a semiconductor device used to amplify and switch electronic signals and electrical power. It is composed of semiconductor material with at least three terminals for connection to an external circuit. A voltage or current applied to one pair of the transistor's terminals changes the current through another pair of terminals. Because the controlled (output) power can be higher than the controlling (input) power, a transistor can amplify a signal. Today, some transistors are packaged individually, but many more are found embedded in integrated circuits.

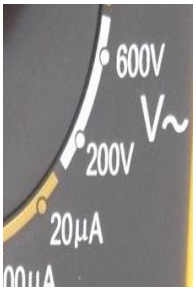
Transistors (typically marked with an “Q” on a circuit board).

No.	Diode Number	Forward Bias Voltage	Reverse Bias Voltage
1	1N4001 		
2	1N914 		

Result:

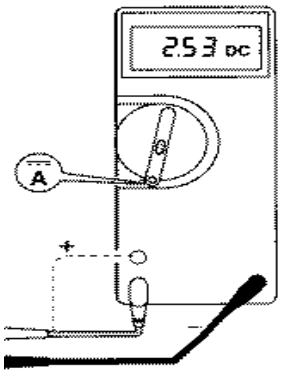






CURRENT MEASUREMENT

1. Connect the positive(red) test lead to the 'V/mA' jack socket and the negative(black) lead to the 'COM' jack socket(for measurements up to 200mA). For measurements between 200mA and 10A connect the red test lead to the '10A' socket.
2. Set the selector switch to the desired uA/mA/A range.
3. Open the circuit to be measured and connect the test leads in **SERIES** with the load in which current is to be measured.
4. To avoid blowing an input fuse, use the 10A jack until you are sure that the current is less than 300 mA. Turn off power to the circuit. Break the circuit. (For circuits of more than 10 amps, use a current clamp.) Put the meter in series with the circuit and turn power on.



 *(File.Close)* Closes the actual circuit or an open subcircuit on the screen. It is very useful for closing an open subcircuit.

 *(Edit.Copy)* Copy a selected part of the circuit or text to the clipboard.

 *(Edit.Paste)* Paste Clipboard contents into the schematic editor. Note that the content may come from the schematic editor itself, TINA's diagram window, or any other Windows program.

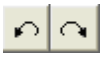
 *Selection mode.* If this button is pressed you can select and drag components with the cursor. To select a component (part), wire or text, just click on it with the cursor. You can also select several objects by holding down the Ctrl key and clicking on the objects one-by-one; or by clicking at one corner of the area, holding down the left mouse button, moving to the opposite corner and then releasing the mouse button. Selected objects will turn red. You can drag the selected objects by dragging one of them. Click and hold the left mouse button when the cursor is over one of the selected objects and move them with the mouse. You can unselect all selected object(s) by clicking on an empty area. One or more selected objects can be deleted while leaving the others still selected by holding the Ctrl key down and left-clicking the mouse.You can select all the wire segments, connected by the same ID on Jumpers, by holding down the Shift Key and clicking one of the wire segments.

 *(Insert.Last component)* Retrieves the last component inserted, for a new insertion of another copy, with the same parameters as the previous insertion.

 *(Insert.Wire)* Use this icon for inserting (adding) wires to the schematic design.

 *(Insert.Text)* Add comments into schematics and analysis results.

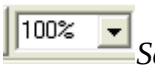
 *(Edit.Hide/Reconnect)* Use Hide/Reconnect to place or remove a connecting dot between crossing wire or wire-component connection.

 *(Edit.Rotate Left (Ctrl L), (Edit.Rotate Right (Ctrl R),* Rotates the selected component.

 *(Edit.Mirror)* Mirrors the selected component. *Hotkeys* Ctrl L, Ctrl H

 *Switches* On/Off the grid. i.e.makes the grid visible or invisible.

 *Explicitly zoom* in on a selected portion of the current view. It will also zoom out a portion if you just click with the icon into the middle of the area you want to zoom out.



Select zoom ratio from a list from 10% to 200%. You can also select Zoom All which will zoom the effective drawing to full screen. *Interactive modes,* see also on the Interactive menu:

rotate the component by pressing the + or - keys (on your computer's numeric keypad) or mirror it by pressing the asterisk (*) key (also on your computer's numeric keypad). Once you have selected the part's position and orientation, press the left button of the mouse to lock the symbol in place.

❻ **Find component tool:** This tool helps you find by name anycomponent in the TINA catalog. See more details above at the toolbar description.

❼ **Open files tab:** You can have several different circuit files or different parts (macros) of a circuit open in the schematic editor at the same time. Clicking on a tab brings that circuit page up in the editor.

❽ **The TINA Task bar:** *TINA's* Task bar appears at the bottom of the screen and provides speed buttons for the various tools or T&M instruments currently in use. Each tool or instrument operates in its own window and can be made active by clicking on its speed button (icon of the tool). Once the cursor is over the speed button, a brief hint appears. Note that the first button (furthest to the left), the Lock schematic button, has a special function. When the Lock schematic button is pressed, the schematic window is locked in place as a background behind other windows, so that it can never cover a diagram or virtual instruments. When the schematic window is not locked and it is currently selected, you will always see the entire schematic window with any other windows hidden behind.

❾ **The Help line:** The Help line, at the bottom of the screen, provides short explanations of items pointed to by the cursor.

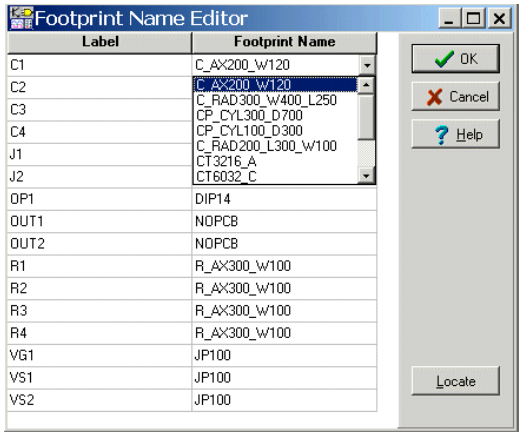


Figure 6:Component Footprint name editor

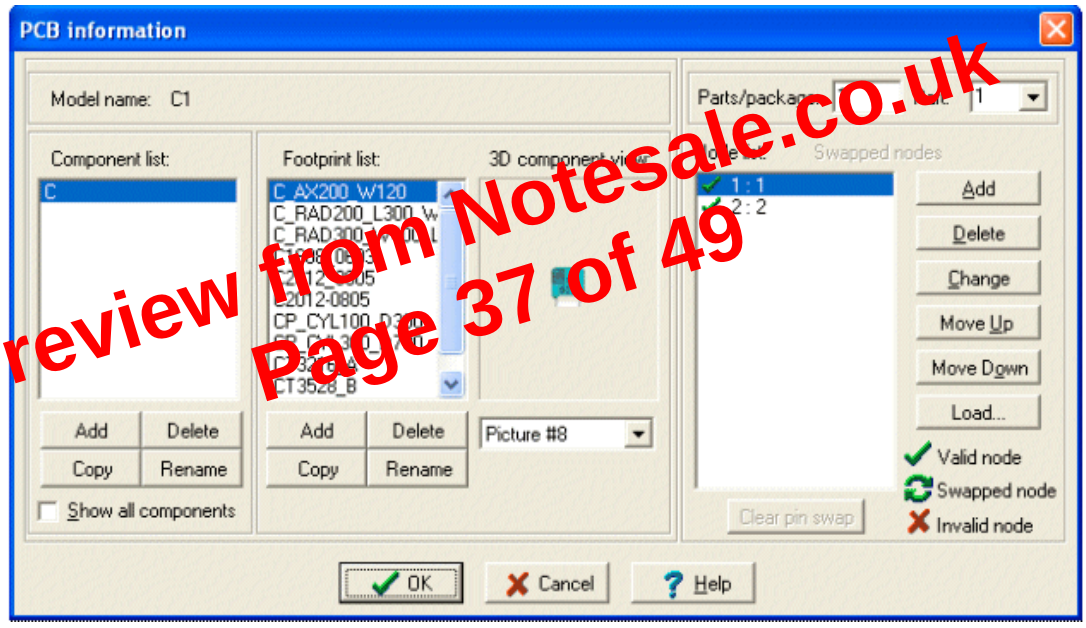
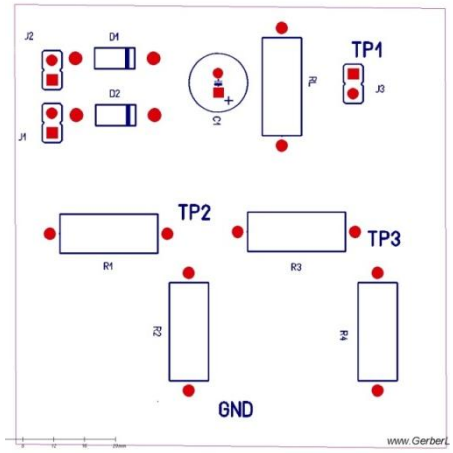


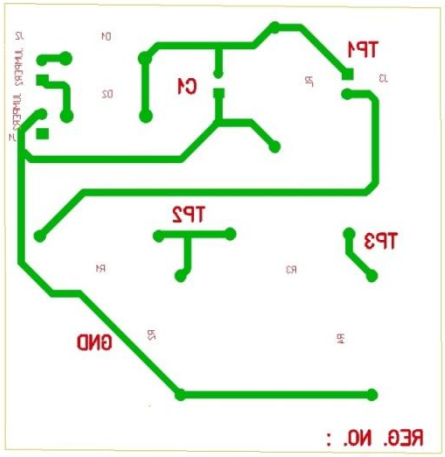
Figure 7:Component information

If you find the footprint name you want on the list, click on it and press OK: you will be returned to the component property dialog with the selected footprint name in the Footprint name line. To confirm the change, press OK on the component property dialog again .If you do not find the footprint name you want, you can add a new footprint using the Add buttons of the "PCB information" dialog. Press the Help button for more information. When everything looks good, you can make a final check by clicking the 2D/3D view button. The 3D view of those components for which a physical representation has already been added will appear

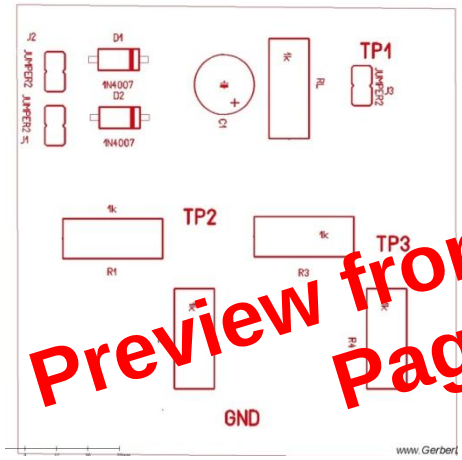
Generate the following output files



Assembly Drawing

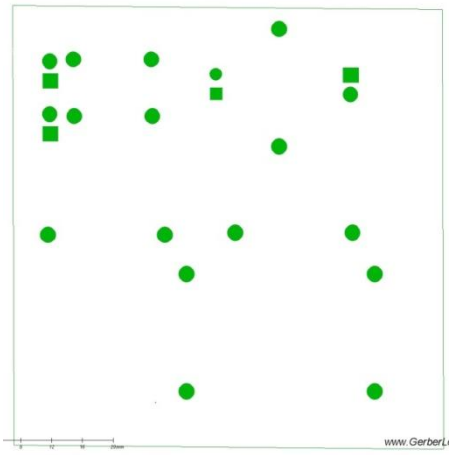


Routing (Bottom)

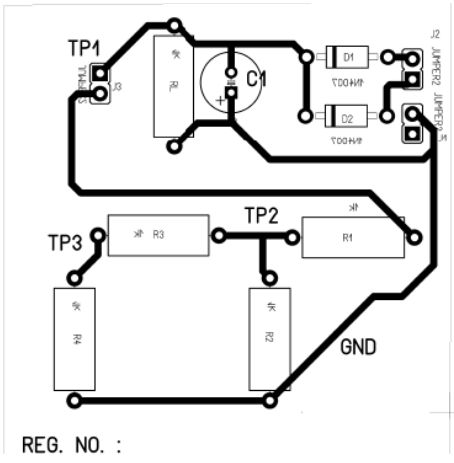


Silk screen

NC Drill Drawing



Solder mask (Bottom)



Top and Bottom layer