This manual is written for the Micro-Cap 10 Electronic Circuit Analysis Program for PC Computers. A free PC version of the program is available at: www.spectrum-soft.com. Contributions made in 2012 by Vahid Bahrami Yekta, Joel Niebergal (tutorial), Zhang Dong (simulations) and Robert Fichtner are acknowledged.

1. Drawing schematics with Micro-Cap 10

The work space of the software is shown in Fig.1. The white part of the page is where you put your components. If you roll the cursor over different menu parts, a comment will appear that help you to find out the action. For the beginning, you need only some of the menus:

Fig.1 Work Space
1. Cursor menu: You can change your cursor according to your need on this menu.

   ![Cursor Menu](image)

   **Fig.2 Cursors**

   From left to right on Fig.2: select cursor (To edit a component), Component cursor (to place a components on the screen), Text cursor (to add a text box to the page), and wiring cursor (to wire between components)

2. Component pallet: it consists of three tabs.

   ![Component Pallet](image)

   **Fig.3 Component Pallet**

   - Browse tab: you can find any needed component in this tab; they are categorized based on their type, for example: analog parts, passive parts, active parts. To put a component on the screen, just click on its name and you can see the mouse changes to the component shape.
- Search tab: If you don’t know a component category, don’t worry, just search its name in the search tab and find it swiftly.

- Favourites tab: Recently used items are shown on this tab, very easy to find common components like resistance on this tab instead of finding it any time.

3. Editing tools:

Fig.4 Editing tools

You can rotate your components to the desired position by these tools. Play around with them, they are easy to work.

When you place a component on the screen, a Model Attributes Dialog Box appears:

Fig.5 Model Attributes Dialog Box

In the window you can define all of the properties of a component. Some components like resistance need a value that you can enter here. We discuss this further in the following.
Exercise: Create the schematics shown in Fig.6.

![Circuit Diagram](image)

**Fig.6 Drawing circuit exercise**

The battery and the sine wave source are in Waveform Sources in the Component pallet. To place the sine source V2, select the Sine Source and then click on the sheet where the Sine Source is to be placed. The attributes box as shown in Fig.7 will appear.

![Sine Source Attributes Box](image)

**Fig.7 Sin source attributes box**

From the menu, as in Fig.7, choose "GENERAL", as the value, this is a default model for this part. All the parameters are valued by default when you choose general model.
2. Copying and Printing

You can choose, File->print, from the menu to print your schematic. Or, even better, create a selection box by dragging the cursor over the circuit (click and hold the mouse button). From the Edit menu, select “copy to Clipboard” then “Copy the Select Part Box in BMP Format”, then paste the image in to word or paint programs to edit.

Exercise: Copy the schematics created in the previous exercise to Open Office or MS Paint.

3. Diode Characteristics

Let’s create the following circuit shown in Fig.8:

![Fig.8 A diode circuit](image)

The default model (GENERIC) is selected as the value for diode using the same procedure in section 1. For the battery, the value is set to zero. A diode is a two-terminal device with a certain I-V characteristic. To obtain V-I characteristics of this diode we will use the DC analysis. For DC-analysis, go to analysis->DC in the menu. The “DC analysis limit” window appears, set the values just like Fig.9:

![Fig.9 DC analysis window](image)

In this analysis the applied voltage V1 (defined as Variable 1) will be varied from maximum of 1 volt to minimum 0.5 volts at steps of 0.05. The corresponding diode current I(D1) is calculated
and plotted on Y-axes for 0 to 100mA range vs. voltage applied to the diode on X-axes. Here is the result:

![Diode I-V Characteristic](image)

Fig.10 Diode I-V Characteristic

You can stop, run again and pause the simulation at any time, by the control buttons on the result screen. Fig.11 shows these buttons.

![Simulation control buttons](image)

Fig.11 Simulation control buttons

4. Bipolar Junction Transistor (BJT) characteristics.

A bipolar junction transistor (BJT) is a three-terminal device. One terminal is called the base (B) and is a controlling terminal. The other terminals, the Collector (C) and the Emitter (E), form a two-terminal device where the I-V characteristic is controlled by base current. The transistor is therefore characterized by family of I-V characteristics with the base current IB as the controlling parameter. DC analysis is used to generate such a family of transistor characteristics. The transistor is an active device that requires a supply voltage Vcc to be applied between the collector and the emitter. Let’s construct the following circuit using the default model for Q1:
For Q1 use the generic model as value, it assigns all of the parameters by a default value.
We have to define base current IB generated by a current source and set it to 0 mA. Similarly, the voltage source Vcc is defined and set to 0 volts.

![IB source attributes](image)

Fig.14 IB source attributes

Now, select from top menu, analysis->DC… and adjust the simulation parameters as in Fig15:
Fig. 15 DC analysis window for BJT

Input 2 is defined as base current IB. It assumes values from 0 to 20uA in steps of 4uA - in a total of 6 steps. For each base current step, the Input 1 defined as Vcc voltage is swept from 0 volts to 2 volts. For a numerical display only, we can select the number of points for each Vcc sweep, say 5 points. For each IB, VCC is swept from 0 volts to 2 volts and the corresponding current IC is calculated and its value displayed. Now, if you run the simulation, the DC characteristic of BJT is shown, as in Fig.16:

Note: The text “IB = ... uA” has been added after simulation run, by Ms-word.

Fig.16 Transistor Characteristics
If you want to see how the numerical values have changed compare to each other, on the plot page go to "DC->numeric output":

Limits

Variable 1 Method Auto
Variable 1 Name VCC
Variable 1 Range 2,0
Variable 2 Method Linear
Variable 2 Name IB
Variable 2 Range 20u,0,4u
Temperature Linear 27
Number of Points 5
Maximum Change % 10
Run Options Save

Temperature=27 IB=0

Waveform Values

VCE(Q1) IC(Q1)
(V) (A)
0.000 -0.003f
500.000m 17.845p
1.000 19.634p
1.500 21.449p
2.000 23.290p
Temperature=27 IB=4e-006

Waveform Values

<table>
<thead>
<tr>
<th>VCE(Q1) (V)</th>
<th>IC(Q1) (A)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>-11.146u</td>
</tr>
<tr>
<td>500.000m</td>
<td>239.816u</td>
</tr>
<tr>
<td>1.000</td>
<td>241.017u</td>
</tr>
<tr>
<td>1.500</td>
<td>242.217u</td>
</tr>
<tr>
<td>2.000</td>
<td>243.417u</td>
</tr>
</tbody>
</table>

And so on …

This Generic model represents a very idealized transistor. Such plots are similar to that obtained by a curve tracer.

5. Transient Analysis

Suppose we want find voltages at each node of Fig.17 First we need to display the node numbers. To display node numbers, go to, Options -> View -> Node numbers.

You can then run a TRANSIENT analysis. This type of analysis simulates a scope and will generate voltage at each node v1, v2 and v3 vs. time. This is indicated by selection of Time Range in Fig.18 limits box.
After simulation, the simulation result would be look like Fig.19:

You can hold the mouse at any desired location on the plots and read the corresponding voltages and time.
After running TRANSIENT analysis, you can see all node voltages displayed directly on the circuit if you select Options -> View -> Node Voltages/State from the top menu.

As an exercise calculate the node voltages and verify the simulation results.

6. Sinusoidal Sources

Put two Sinusoidal Voltage source on the screen. Choose general and 60Hz model for them, but change their frequencies to 100 KHz and 200 KHz and adjust other parameters like Fig.21.

![Fig.20. Example of sinusoidal sources](image)

![Fig.21 sin source attributes](image)
The entries in the Transient analysis window shown in Fig.22 are intended to produce three plots. The results are shown in Fig.23. First plot will display voltages waveform $v(1)$ at node 1 and waveform $v(2)$ at node 2 Vs. time. The second plot will show the sum of $v(1)+v(2)$ Vs. time. The third plot is $v(1)$ vs. $v(2)$. The last plot is similar to X-Y mode of the scope. In our case, it produces the so called Lissajous figure.

![Fig22. Transient Analysis window](image)

![Fig.23 various plots generated by sinusoidal sources](image)
7. Curve Tracer

Here we will demonstrate the simulation of a curve tracer. Draw the circuit shown in Fig.24. Instead of diode D1, you can insert any nonlinear two terminal devices to be investigated. For D1 choose GENERIC as its model and for the sin source set A = 10, freq = 10Hz.

A sinusoidal voltage is applied to the diode. The current through D1 is monitored as a voltage drop on a 40 ohms resistor. This resistor limits the maximum current through the diode (or any other nonlinear device). We will plot the value of the diode current vs. the voltage applied to the diode. In order to get the right magnitude and the direction of this current we have to convert the voltage at node 1 to a current through the diode as indicated in the Transient analysis window in Fig.25.

Fig. 24 Curve tracer circuit

Fig. 25 Transient analysis window for curve tracer
The simulation result is shown in Fig.26.

![Fig.26 Curve tracer simulation result](image)

Note: In order to avoid transients within the diode, we used a very low sweeping frequency of 10 Hz. You can compare this result with that obtained earlier in Fig.10, p. 1.8 for the same diode.