

Micro-Cap 11

Electronic Circuit Analysis Program User's Guide

Copyright 1982-2016 by Spectrum Software

1021 South Wolfe Road

Sunnyvale, CA 94086

Phone: (408) 738-4387

FAX: (408) 738-4702

Internet: www.spectrum-soft.com

Support: support@spectrum-soft.com

Sales: sales@spectrum-soft.com

A quick tour

As a quick introduction to MC11, we'll take a brief tour of the program. Begin with a double click on the MC11 icon.

To begin the tour, we'll load one of the sample circuit files supplied with MC11. Choose **Open** from the **File** menu. When the file prompt comes up, type in the file name MIXED4. Click Open. The program loads the circuit and displays it.

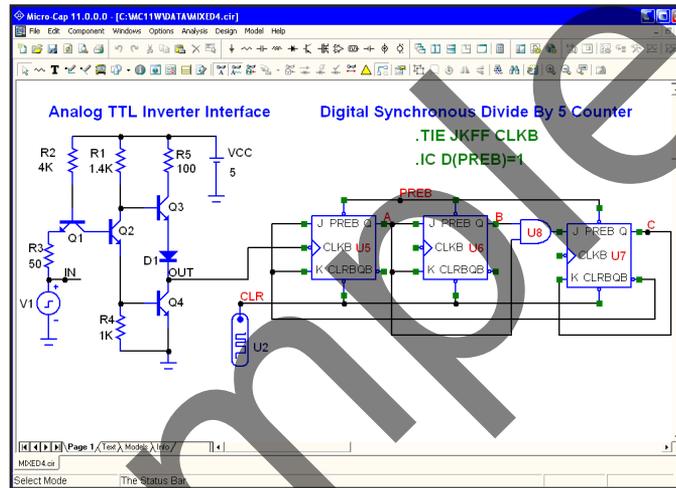


Figure 2-1 The MIXED4 sample circuit

The circuit uses an analog TTL inverter to drive a digital three-stage synchronous divide-by-five counter. The digital clock input to the first JK flip-flop is driven by the analog TTL output, while the CLR initializing pulse for the flip-flops comes from a digital source, U2. The CLK inputs are connected together with a .TIE command. The flip-flop preset nodes are connected together by a wire and the node is labeled PREB with grid text. This node is initialized to a '1' state with an .IC command. The digital source states are specified in a .DEFINE command statement located in the schematic text area. The text area is a private cache in each schematic for storing text. You can toggle the display between the drawing and text areas by pressing CTRL + G or by clicking on the Text tab in the lower page scroll bar. This combination of analog and digital circuitry is easily handled by the program, since it contains a native, event-driven digital simulation engine, internally synchronized with its SPICE-based analog simulation engine.

Transient analysis display

Select **Transient** from the **Analysis** menu. MC11 presents the Analysis Limits dialog box where you specify the simulation time range, choose plot and simulation options, and select waveforms to be plotted during the run.

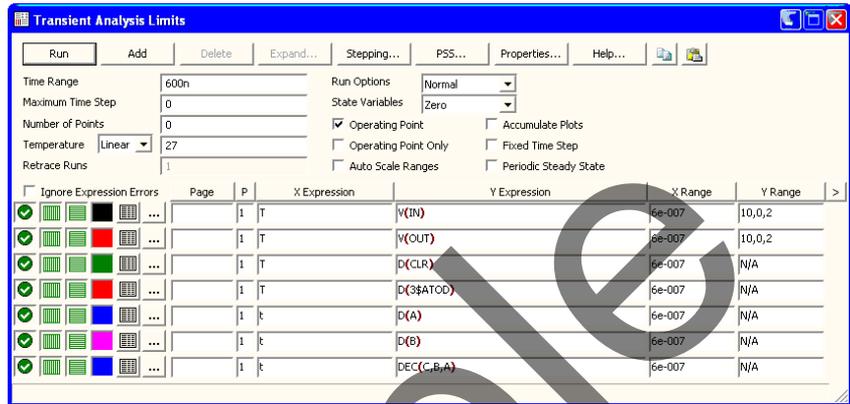


Figure 2-2 The Analysis Limits dialog box

Here we have chosen to plot the analog voltage waveforms at the input pulse source, the output of the analog section, and several of the digital waveforms. In general, MC11 can plot any expression using as variables any node voltage or state or any device terminal voltage or current. Here are a few other expressions that could have been plotted in this circuit.

d(CLR)	Digital state waveform on node CLR.
hex(CLR,PB,1,2)	Hex waveform at nodes CLR,PB,1,2.
vbe(q3)	Base-emitter voltage of transistor q3.
I(VCC)	Current through the VCC source.
PG(VCC)	Power generated by the VCC source.
qbe(q1)	Charge stored in the base-emitter cap of q1.
cbc(q4)	Base-collector capacitance of q4.

There are a great many other variables available for plotting. Click the *right* mouse button in any of the Y Expression fields to see a list of variables available for plotting or printing. Click outside of the menu to remove it.

Now let's run the actual simulation. Click on the Run button  or press F2 to start the simulation. MC11 plots the results during the run. The run can be stopped at any time by clicking the Stop button  or by pressing ESC.

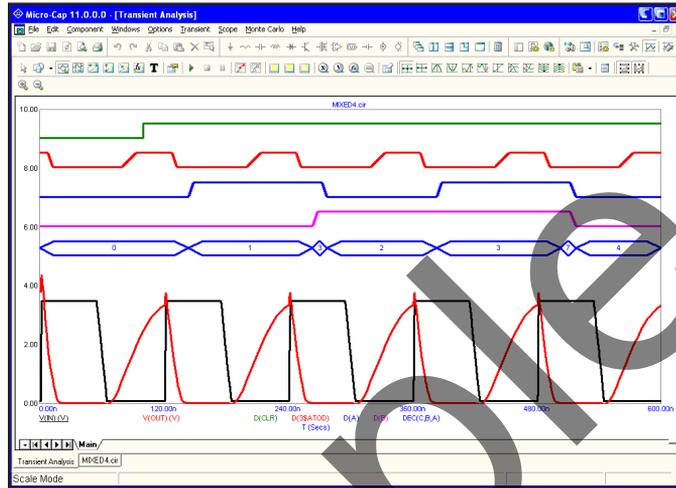


Figure 2-3 Transient analysis of the circuit

The final waveform plots look like Figure 2-3. Each waveform can have a unique token, width, pattern, and color. These options are set from **Options / Default Properties for New Circuits** for new circuits and from the Properties (F10) dialog box for the current circuit. Waveforms can be grouped into one or more plots. Grouping is controlled by the plot number assigned to each waveform. The plot number is the number in the P column of the Analysis Limits dialog box. You can assign all the waveforms to a single plot, or group them into as many plots as will fit on the screen. To assign several waveforms to the same plot, give them all the same plot number. To create several plots, use different plot numbers. To disable plotting a waveform, enter a blank for the plot number.

In this example, all waveforms in the plot group share a common vertical scale. To use different scales disable the **Plot Properties (F10) / Scales and Formats / Same Y Scales for Each Plot Group** option and enter the scales you want in the Y Range field. All waveforms in a plot group share the horizontal or X scale.

Digital and analog waveforms may be mixed or separated. Separated digital waveforms show the expression on the left, adjacent to the plot. Mixed digital waveforms display the expression at the bottom of the plot along with the analog expressions in the order of their occurrence in the Analysis Limits dialog box.

The same rich set of expressions available in transient analysis is also available in AC analysis. In addition, these operators are very useful:

db(c)	Finds the decibel value of the complex expression c.
re(c)	Finds the real value of the complex expression c.
im(c)	Finds the imaginary value of the complex expression c.
ph(c)	Finds the phase in degrees of the complex expression c.
gd(c)	Finds the group delay of the complex expression c.
inoise	Finds the input noise reflected to the input node.
onoise	Finds the output noise at the output node.

Scope in Cursor mode

Click on the Cursor mode button  in the Tool bar or press F8. In this mode, two cursors are placed on the graph and may be moved about. The table below the plots show the waveform values at each of the two cursors, the difference between them, and the slope. The display should look like this:

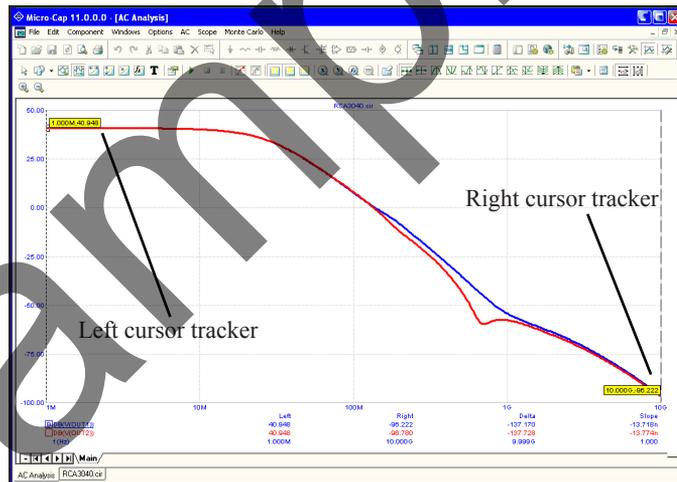


Figure 2-12 Cursor mode

The left mouse button controls the left cursor and the right button controls the right cursor. The cursor keys LEFT ARROW and RIGHT ARROW also control the cursors. As the cursors move around, the numeric display continuously updates to show the value of the waveforms, the change in waveforms between the two cursors, and the waveform slope between the two cursors. The optional cursor trackers also track the X and Y values as the cursors are moved.

The Optimizer

Note from Fig 2-10 that the load resistors R1 and R3 are set to RL, a symbolic variable defined as 1.32K. We'll optimize the value of RL to maximize the gain at 1MHz. Press CTRL + F11 to bring up the Optimizer. It looks like this:

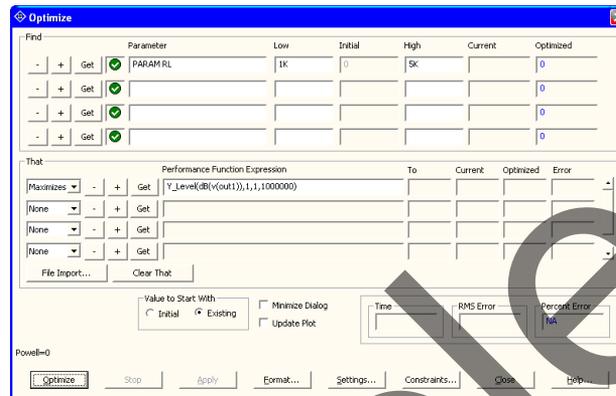


Figure 2-13 The Optimizer dialog box

These settings seek the value of RL that maximizes the performance function $Y_Level(DB(V(OUT1)),1,1,1E6)$. From Fig 2-12 the value of $DB(V(OUT1))$ at 1.0MHz before optimization is 40.9dB. Click on the Optimize button. The optimizer whirs and chugs and eventually spits out an optimal value of 2.3K. Click on the Apply and Close buttons. Press F2 to rerun the analysis, then F8 and you'll see that with $RL=2.3K$, $DB(V(OUT1))$ at 1MHz is now 45.4dB, a net gain of 4.5dB. Press F3 to exit the AC analysis routine. Choose **Close** from the **File** menu to unload the circuit.

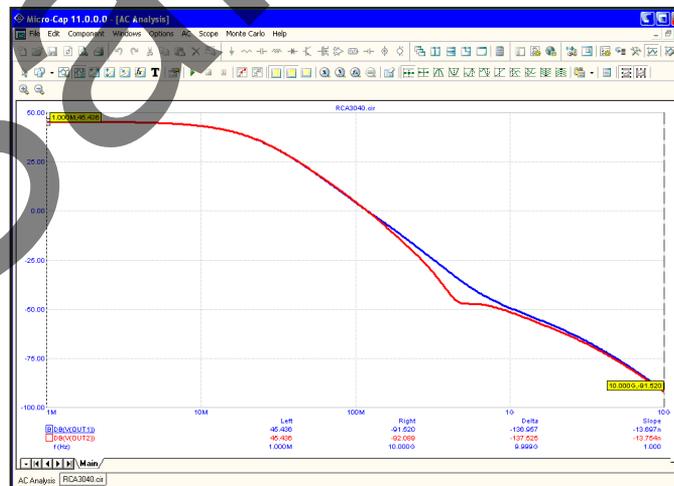


Figure 2-14 The optimized gain curve

DC analysis

Choose **Open** from the **File** menu to load a new circuit, then type "CURVES" and click the Open button.

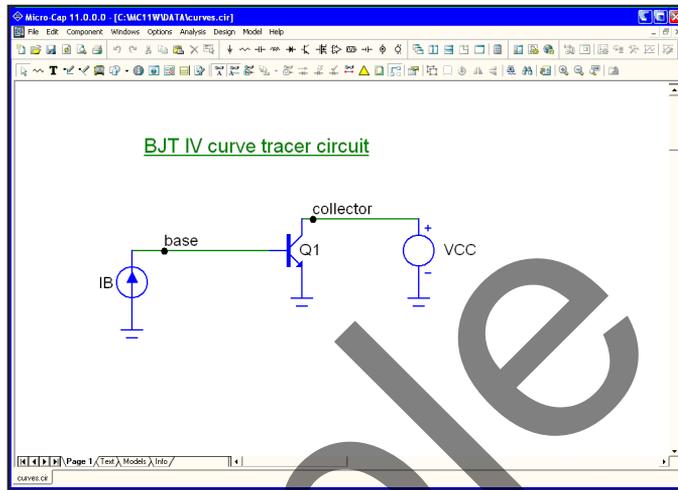


Figure 2-15 The CURVES circuit

This creates a set of bipolar transistor IV curves. Choose **DC** from the **Analysis** menu. Click on the Run button. The plot looks like this:

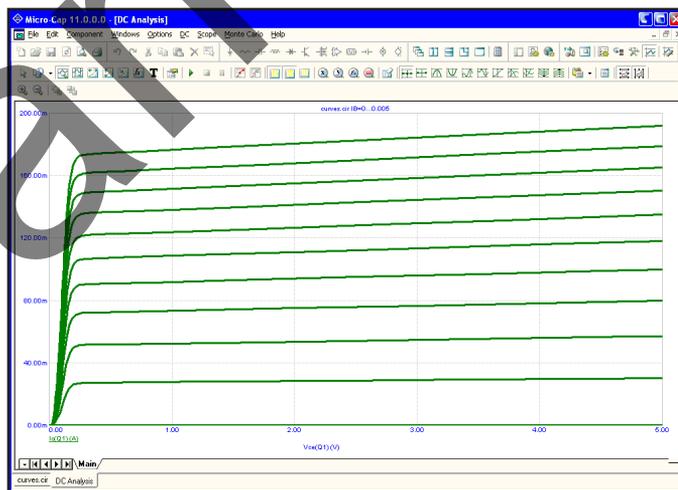


Figure 2-16 The IV curves of a bipolar transistor

Press F3 to exit the DC analysis, then unload the circuit with CTRL + F4.