

# 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](http://www.spectrum-soft.com)

Support: [support@spectrum-soft.com](mailto:support@spectrum-soft.com)

Sales: [sales@spectrum-soft.com](mailto: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.

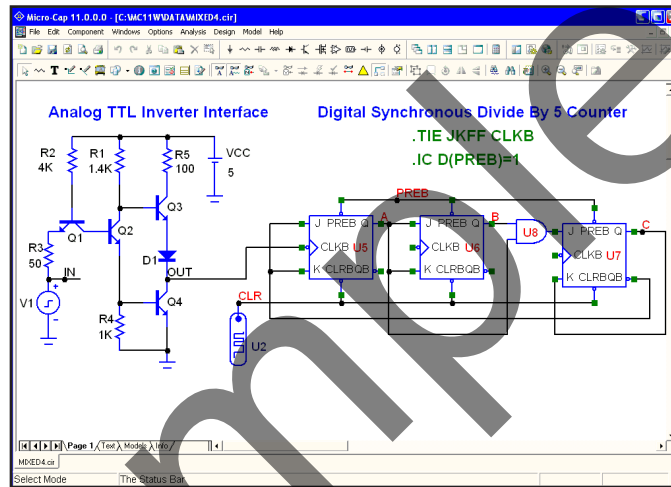
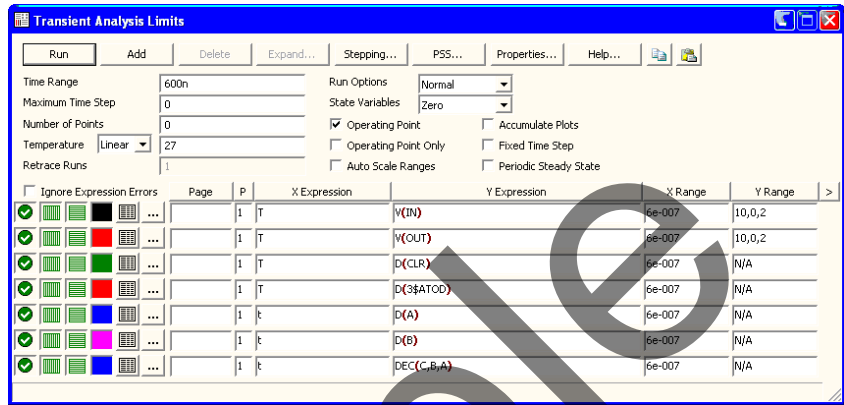


Figure 2-1 The MIXED4 sample circuit

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.





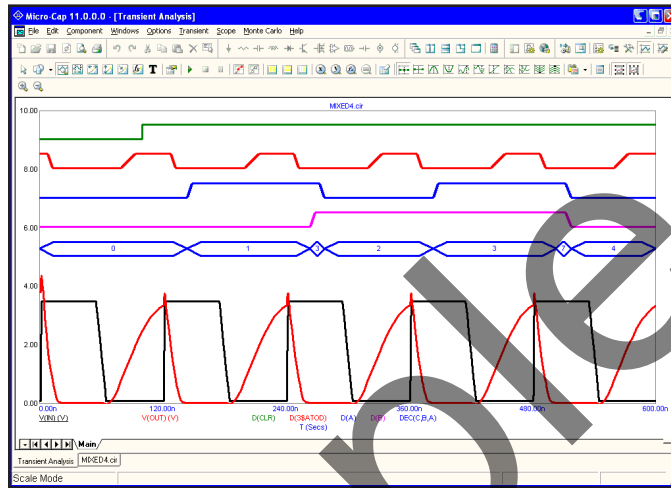
**Figure 2-2 The Analysis Limits dialog box**

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:

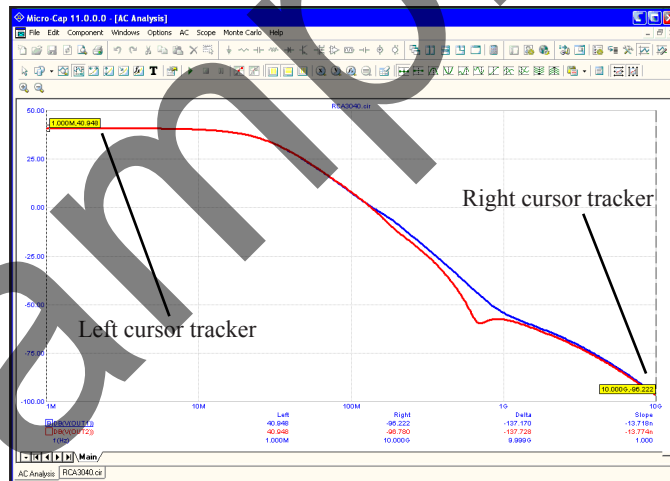


Figure 2-12 Cursor mode

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:

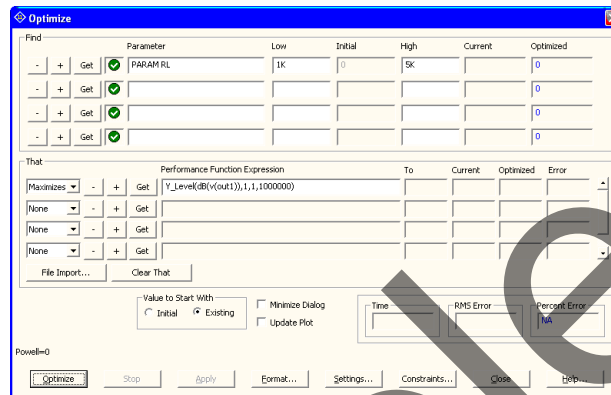


Figure 2-13 The Optimizer dialog box

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.

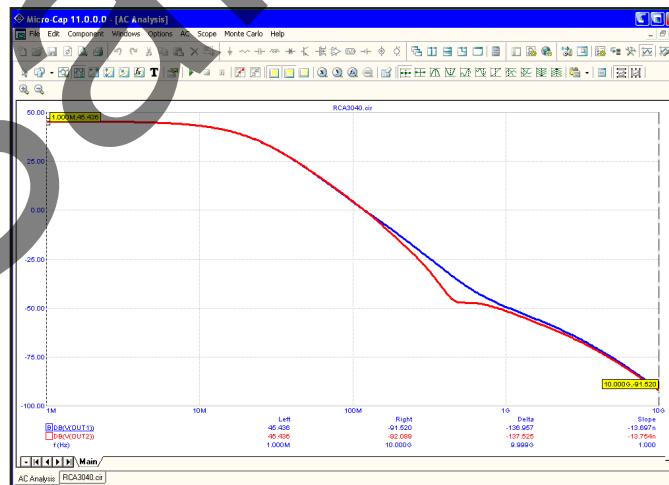


Figure 2-14 The optimized gain curve

## DC analysis

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

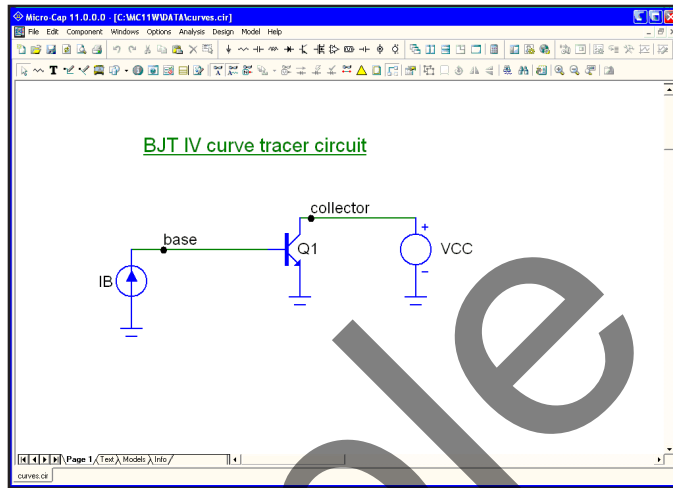


Figure 2-15 The CURVES circuit

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:

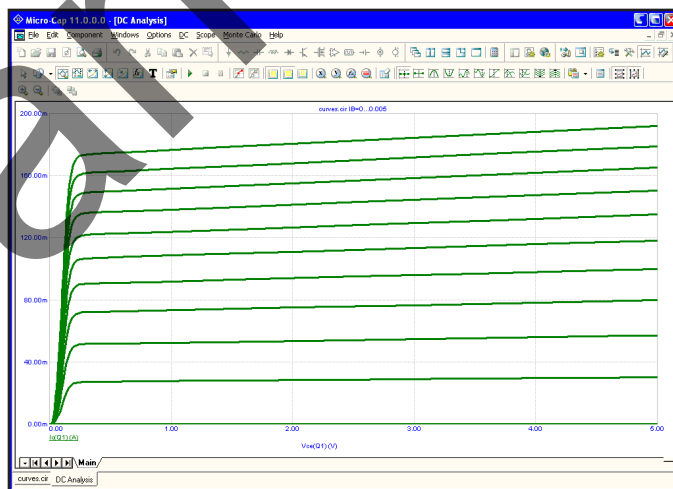


Figure 2-16 The IV curves of a bipolar transistor

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

## Probe

Press CTRL + O, then type "MIXED4". Click Open. Choose **Probe Transient** from the **Analysis** menu. If needed, MC11 runs the analysis, saves it to disk, and presents the Probe display. You scroll the schematic or pan it with a click and drag of the *right* mouse button and probe any node or component with a click of the *left* mouse button. Move the mouse to the pulse source and click the *left* button on the dot near the IN node, and its voltage waveform is fetched from disk and plotted. Click the mouse on the OUT node, CLKB pin, J pin, and CLR node of the first flip-flop. This plots the analog input and output, and the clock, J, and clear digital waveforms. Analog and digital waveforms are placed in the same plot when the **Separate Analog and Digital** option on the **Probe** menu is not checked.

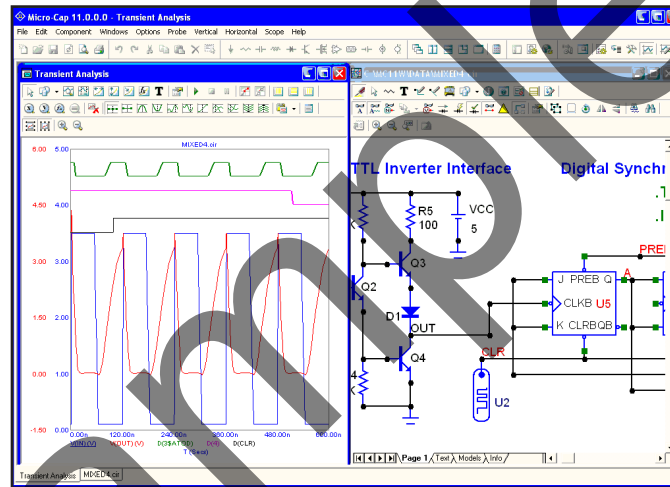
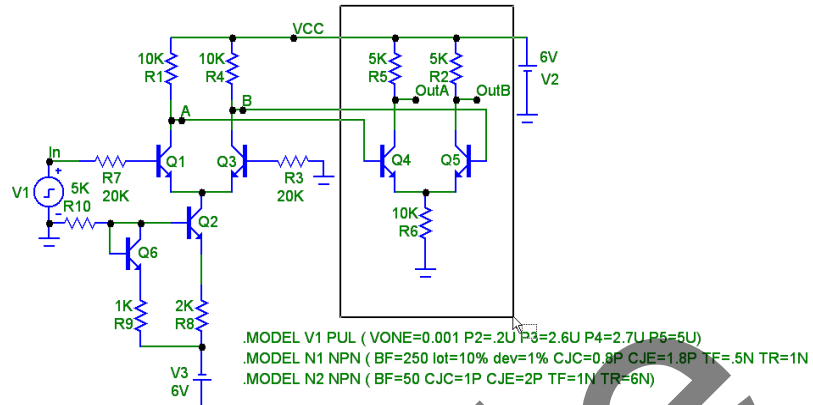


Figure 2-17 The Probe display

Probing on a node produces either analog node voltage waveforms or digital node state waveforms. Probing between the leads of an analog device produces pin to pin terminal voltage waveforms such as VBE(Q4). Probing on a digital device produces a list of the node names. Selecting a node name from this list plots the digital waveform on that node. For analog parts, current waveforms are also available, as are charge, capacitance, inductance, flux, and all the usual variables. In addition, you can enter expressions involving circuit variables.

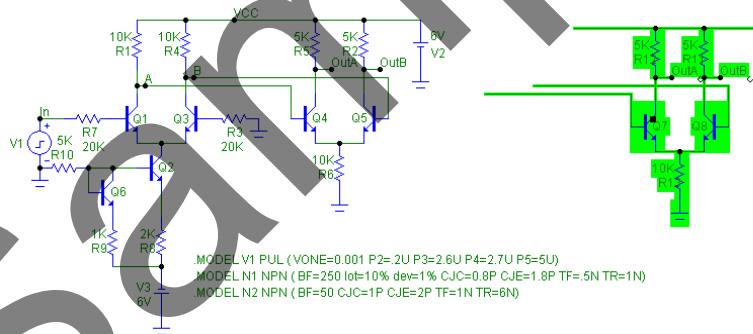
Press F3 to exit the Probe routines and return to the Schematic editor. Remove the circuit with CTRL + F4. This completes the quick tour. For more extensive tours of MC11, see the **Demo** options in the **Help** menu.





**Figure 3-13** Selecting a region for copying to the clipboard

Now press CTRL + C to copy the selected region to the clipboard. Move the mouse to the right of the battery and click the left button. This moves the insertion point. Press CTRL + V to paste the clipboard contents to the insertion point. Your circuit should now look like this:



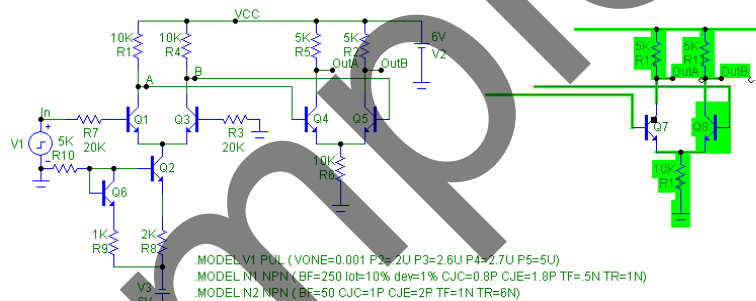
**Figure 3-14** After pasting the clipboard contents

The selected region has been copied, or pasted, from the clipboard to the circuit.

## Selection

Single objects or entire regions may be selected for editing, deleting, moving, rotating, mirroring, stepping, or copying. They may be selected or deselected individually or as a group. Clicking on an object or region while the SHIFT key is down toggles the selection state. If the object or region was formerly selected, it becomes deselected. If it was formerly deselected, it becomes selected.

To illustrate, press the SHIFT key and hold it down. Click on the Q7 transistor in the selected group. Be careful not to click on the name Q7 as that will select the PART attribute rather than the part itself. Release the SHIFT key and the circuit looks like this:



**Figure 3-15 Partial selection using SHIFT + mouse click**

The Q7 transistor is now deselected. If you press the Del key, Q7 will remain but the rest of the selected region will be deleted. Press the SHIFT key and click the same transistor and it is again selected. If you now press the Del key, the entire selected region will be deleted.

By using partial selection, virtually any subset of a schematic can be selected for editing, copying, deleting, or moving.

To complete this part of the tutorial, click on the **Revert** option from the **File** menu to restore the DIFFAMP circuit.

## The Plot Properties dialog box

Curve display and format can be changed with the Plot Properties dialog box. Press F10 to invoke the dialog box for the plot. It looks like this:

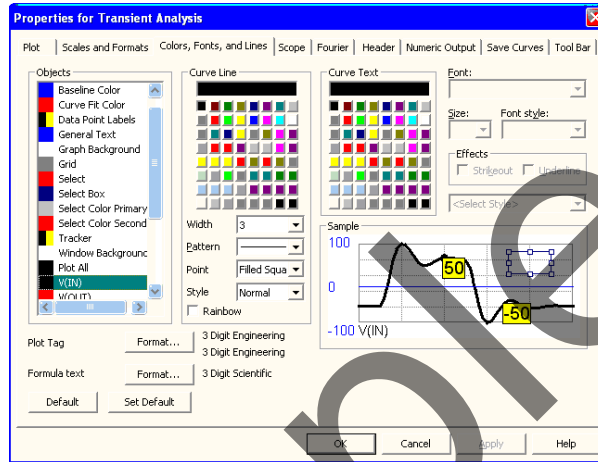



Figure 4-6 The Properties dialog box

Select V(IN) from the **Objects** section of the **Colors, Fonts, and Lines** panel, and from the **Curve Line** section select a width of 5. Click on V(IN) and select the third pattern in the list. Click the OK button. Disable tokens by clicking the  Tokens button in the Tool bar. The screen should look like this:

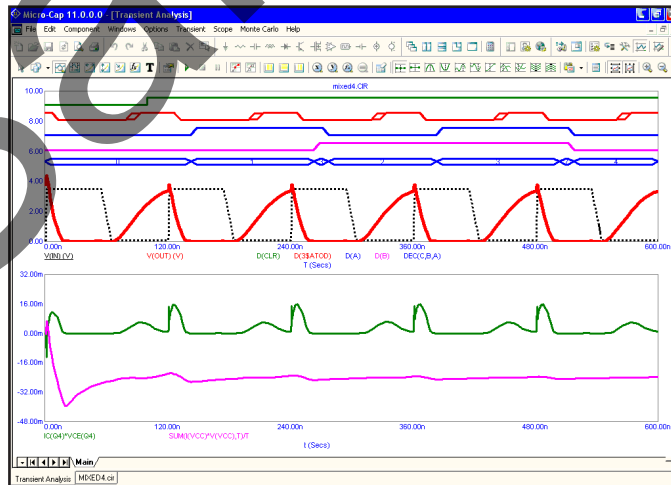

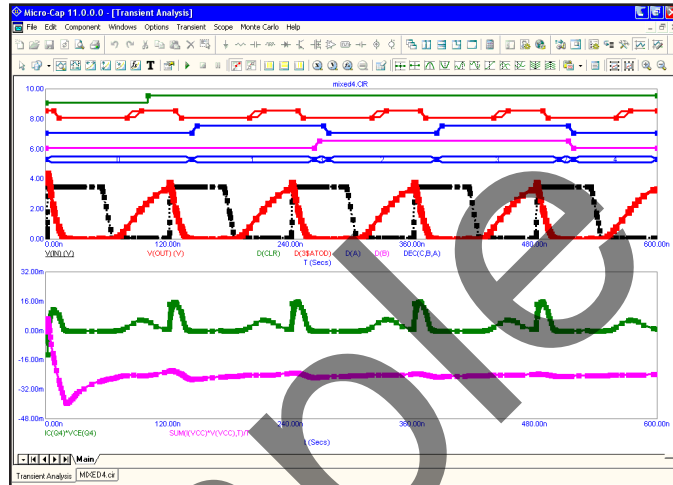


Figure 4-7 Using line width and pattern

## Data Points Option

Let's experiment with some of the other options. Click on the Data Points button . This toggles the display of marker icons for the calculated data points. Since the option was initially disabled, clicking the button enables it, and the data point marker icons are added producing a plot that looks like this:





**Figure 4-8** The plot with the Data Points option enabled

Since the simulator uses a variable time step algorithm to minimize errors and accelerate the run, the actual time points will not be regularly spaced. The data point markers are placed at the actual computed time points. There are at least two situations where you might want to see these:

**Inadequate curve sampling:** This can happen in transient analysis when the simulator flies past a switch threshold. It may also happen that you want more data points to "fill in" source curves that look choppy. Even though the charge error is strictly controlled by the internal truncation error routines, the data sampling may seem inadequate. Data point icons show where the data points are and whether more will help. In AC analysis, your plot may miss a steep spike on a narrow pass or reject band. Data point icons show whether the simulator actually made a calculation within the band.

**Identifying real curve spikes:** Sometimes you see spikes on curves. If the curve changes direction from one data point to the next, it's likely the spikes are simply artifacts of the integration routines. The only way to tell is to mark the data points. If they are integration artifacts, you can reduce them by decreasing RELTOL or the minimum time step or both.

## Experimenting with the options

To illustrate some other options, click on the horizontal axis grid  and vertical axis grid  buttons. The plot should now look like this:

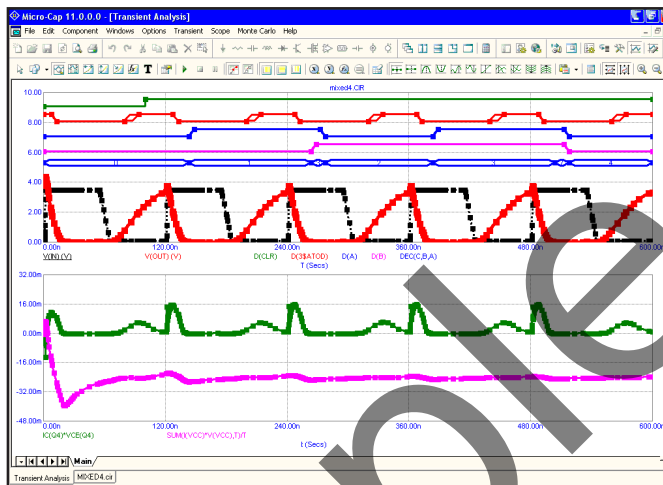



Figure 4-9 Adding grids to the plot

Press F9, then enable the Operating Point and Operating Point Only options. This tells the program to do an operating point calculation, and then stop. An operating point calculation sets all sources to their defined initial values, removes all capacitors, and replaces all inductors with short circuits. It then calculates the steady-state DC values of the state variables (node voltages, inductor currents and digital states) at the initial time point. This set of values is referred to as the DC operating point. Press F2 to do the operating point only run. Since we're doing only an operating point, no plot will appear. Exit the analysis by pressing F3. Click on the Node Voltage  button and MC11 displays the last calculated states and voltages. In this case, since we just completed an operating point only calculation, these are the operating point values.

## What happens in AC analysis

In AC analysis the optional DC operating point and optional Periodic Steady State are first computed. Digital nodes stabilize and potentially affect analog nodes to which they may be attached. Linearized, small signal AC models are then created for each component and are entered into a set of linear network equations. These are repeatedly solved at many frequency points to obtain the small signal response of the circuit to whatever AC excitation is present in the circuit. The linearized AC model of a digital component is an open circuit. This means that digital parts are ignored during small signal analysis.

The excitation is supplied by one or more independent sources in the circuit. Pulse and Sine sources provide a fixed, real, 1.0 volt AC signal. User sources provide a signal comprised of the real and imaginary parts specified in their files. The SPICE independent sources, V and I, provide user-specified real AC signal amplitudes. Function sources can create an AC signal only if they have a FREQ expression. These are the only sources that generate AC excitation.

AC analysis represents the excitation and system variables (node voltages and various branch currents) as complex quantities. Operators, such as RE (real), IM (imaginary), dB ( $20 * \text{Log}()$ ), MAG (magnitude), PH (phase), and GD (group delay) are provided to print and plot these complex quantities.

To explore AC analysis, load the file DIFFAMP. It looks like this:

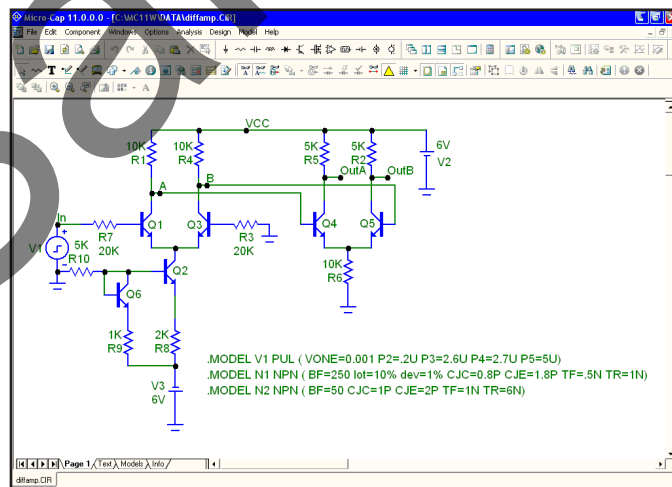
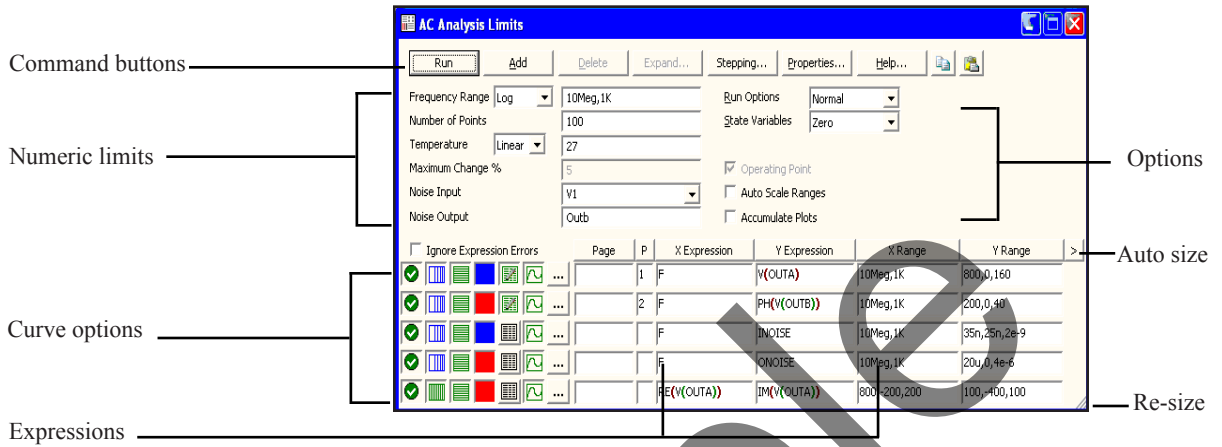


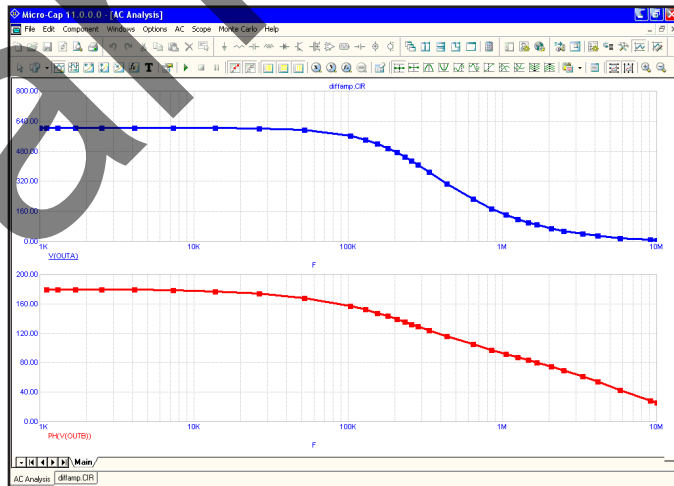
Figure 5-1 The DIFFAMP circuit

Choose **AC** from the **Analysis** menu. When the AC Analysis Limits dialog box comes up, the fields should look like this:



**Figure 5-2 The AC Analysis Limits dialog box**

In this circuit, we have a single source providing a 1.0 volt AC signal. The dialog box instructs MC11 to sweep the frequency from 1K to 10Meg and produce plots of the magnitude of the voltage at the node OUTA and the phase angle of the voltage at node OUTB. Press F2 and the result looks like this:



**Figure 5-3 The AC analysis plot**

## Experimenting with options and limits

The small squares on the plots in Figure 5-3 mark the actual points calculated by MC11. We can produce a smoother plot at the expense of more data points by reducing the *Maximum change %* value from 5% to 1%. Press F9 to display the AC Analysis Limits dialog box. Double-click the mouse in the Maximum Change % field and type "1". Press F2 to run the analysis. It looks like this:



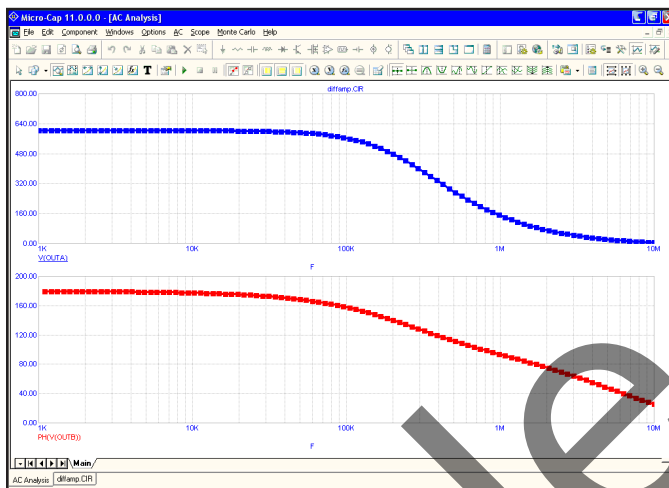
**Figure 5-4** The plot with Maximum change % set to 1%

The lower value of 1% produces a smoother plot by calculating more data points. The current Frequency Step method is Auto. In this mode, the frequency step is automatically adjusted to restrain the point-to-point change of both the vertical value and the angle of the plot itself to less than the specified maximum change. The Auto method produces the smoothest plot at the lowest computation price for most curves.

This analysis uses 1% and generates about three times the number of data points as the 5% analysis. Since the analysis time increases in direct proportion to the number of data points, it is wise to keep the *Maximum change %* as large as possible. A value of 1 to 5 is usually suitable. If the first curve is very flat, you may want to try a fixed frequency step method.

Let's try one of these methods. Press F9 to display the Analysis Limits dialog box. Click in the Frequency Range list box. When the list drops down, click on the Log option. Press F2 to start the run. The result looks like Figure 5-5.

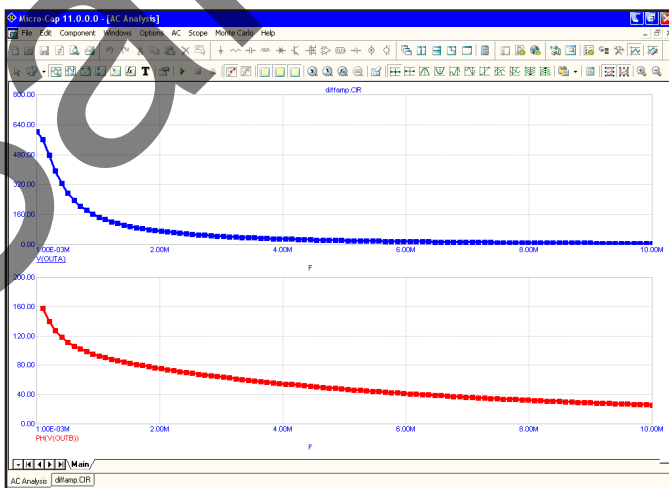




**Figure 5-5 The plot using the log option**


The constant horizontal distance between data points is achieved by making each new frequency a product of the prior frequency and a fixed multiplier. Since the lowest frequency forms the base that is multiplied, it must not be zero.

The Linear option is only used when the X axis is linear. To illustrate, press F9 and edit the Frequency Range field to 1E7,0. Change the Frequency Range method to Linear and change the X Log/Linear button states of the first two curves to linear. Press F2 to run. The result looks like Figure 5-6.



**Figure 5-6 The plot using the linear option**

## Numeric output

To obtain a tabulation of the numerical values simply click the Numeric Output button  for each curve for which numeric output is desired. Do that now for the first two curves. Press F9, then enable the buttons. Press F2 to run the analysis. All text or numeric output is directed to the Numeric Output window and to a disk file with the name DIFFFAMP.ANO. Press F5 to see the window. The first part shows the results of the DC operating point. Scroll down to the Waveform Values section. The display should look like this:

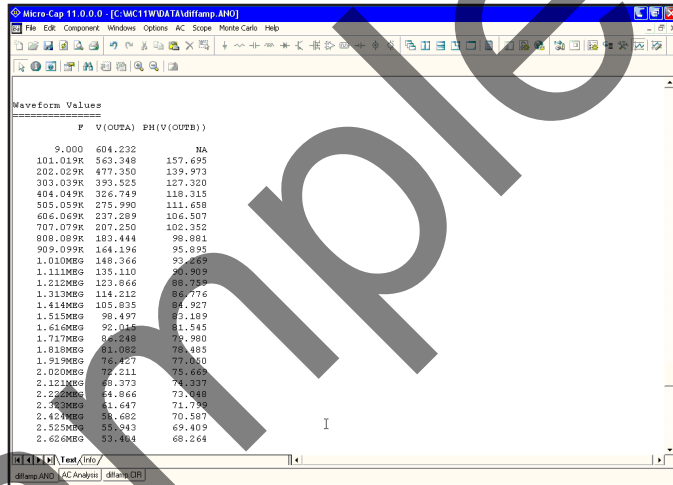




Figure 5-7 Numeric output

A printout of the data in the Numeric Output window may be obtained from the **Print** option of the **File** menu.

To display the Analysis Plot window, press F4, or click on the Analysis Plot button .

To display the Numeric Output window, press F5, or click on the Numeric Output button .

## Noise plots

MC11 models three types of noise; thermal, shot, and flicker noise.

Thermal noise is produced by the random thermal motion of electrons and is always associated with resistance. Discrete resistors and the parasitic resistance of active devices contribute thermal noise.

Shot noise is caused by a random variation in the production of current from a variety of sources including recombination and injection. It is usually associated with dependent current sources. All semiconductor devices generate shot noise.

Flicker noise comes from a variety of sources. In BJTs, the source is contamination traps and other crystal defects that randomly release their captured carriers.

Noise analysis computes the contribution from all these sources as seen at the input and output. Output noise is calculated at the output node specified in the Noise Output field. To plot it, specify ONOISE as the Y expression. Input noise is calculated at the same output node(s) specified in the Noise Output field, but is divided by the gain from the node(s) specified in the Noise Input field to the output node. To plot or print input noise, specify INOISE as the Y expression.

Press F9 and edit the analysis limits to match Figure 5-8. Be sure to disable numeric output for the first two curves and to select the Log option.

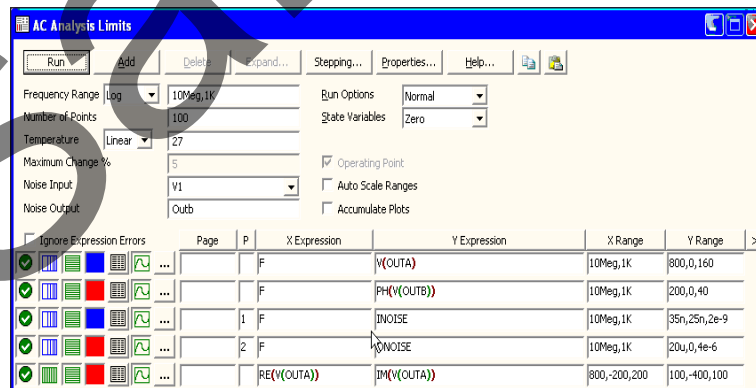
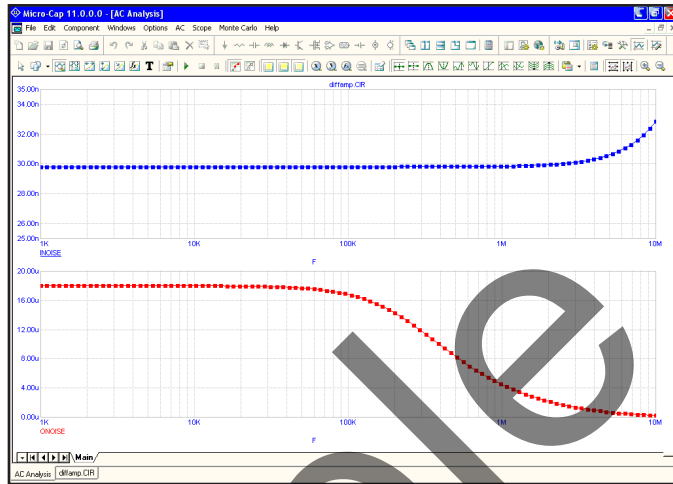


Figure 5-8 The Analysis Limits for Noise plots

Press F2 to run the analysis. The results look like this:



**Figure 5-9 Plots of input and output noise**

In this circuit, thermal noise is generated by the resistors. Shot and flicker noise are generated by the transistors.

Because the equations are structured differently for noise, it is not possible to simultaneously plot noise variables and other types of variables such as voltage and current. That is why we disabled the other curves by placing a blank in their P fields and disabling their Numeric Output field. If you attempt to plot or print both noise and other variables, the program will issue an error message.

Noise is an essentially random process and as a result carries no phase information. For this reason, noise is defined as an RMS quantity. Thus the phase (PH) and group delay (GD) operators will not produce meaningful results when plotting noise. With noise, only magnitudes are meaningful.

The units of noise are  $V / \sqrt{\text{Hz}}$ .

## Nyquist plots

A Nyquist plot is easy to generate. Edit the analysis limits to look like these. Here we have enabled the last curve and disabled the other curves and selected log stepping.

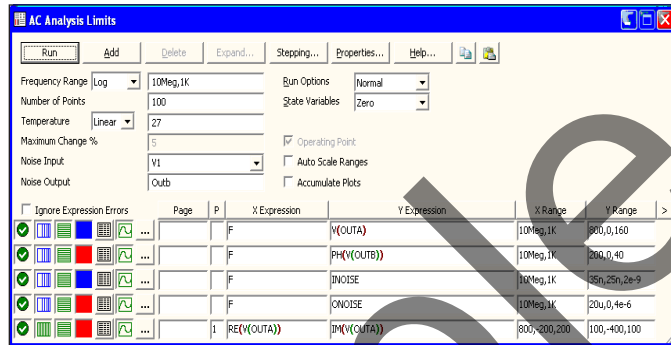


Figure 5-10 Analysis limits for the Nyquist plot

Press F2 to run the analysis. The result looks like this:

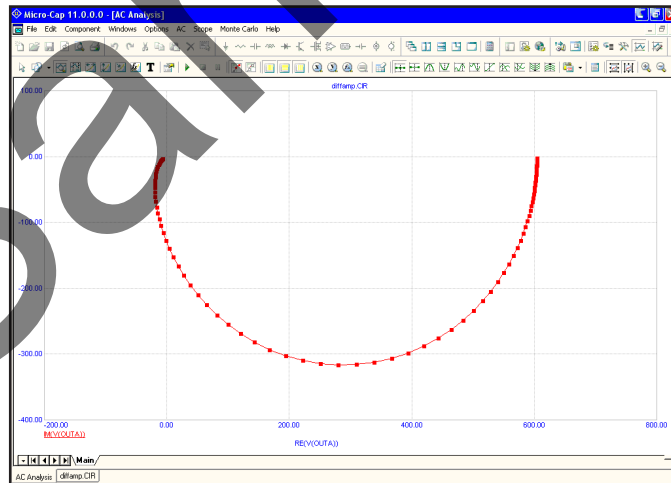



Figure 5-11 The Nyquist plot

Note that the X expression in this case was  $RE(V(OUTA))$ , not F as it usually is. In general, it can be any expression involving any AC variable.

## Smith charts

To generate a Smith chart plot, use the  option from the chart type selector, adjacent to the P column. Here is how the Analysis Limits dialog box looks when set up to create a Smith chart plot for S22, using the SMITH.CIR file.

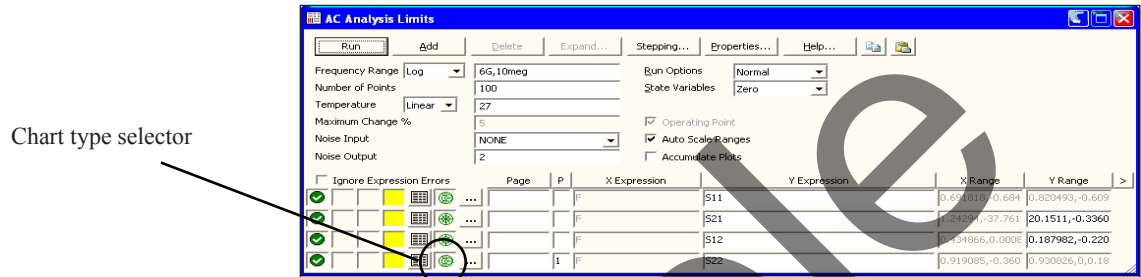


Figure 5-12 Analysis limits for the Smith chart

Note that when the Smith chart option is selected, the X expression must be F, so this field is not available for editing. Here is the resulting S22 plot.

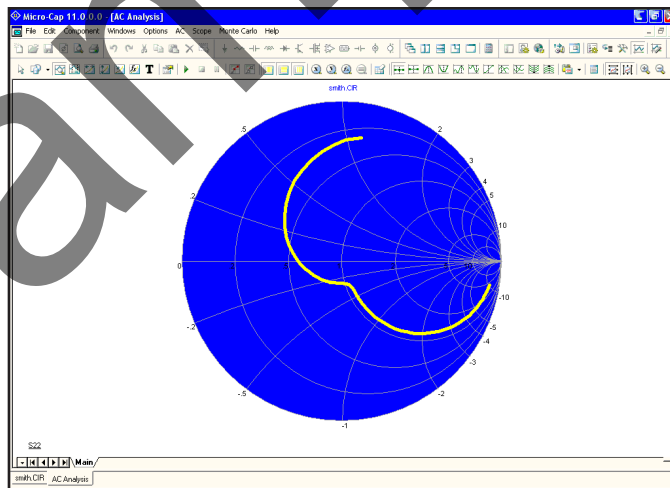



Figure 5-13 Smith chart plot of S22

## Polar plots

To create a polar plot, use the polar plot option, . Here is how the dialog box appears when set up to create a polar plot for S21 using the SMITH.CIR file.

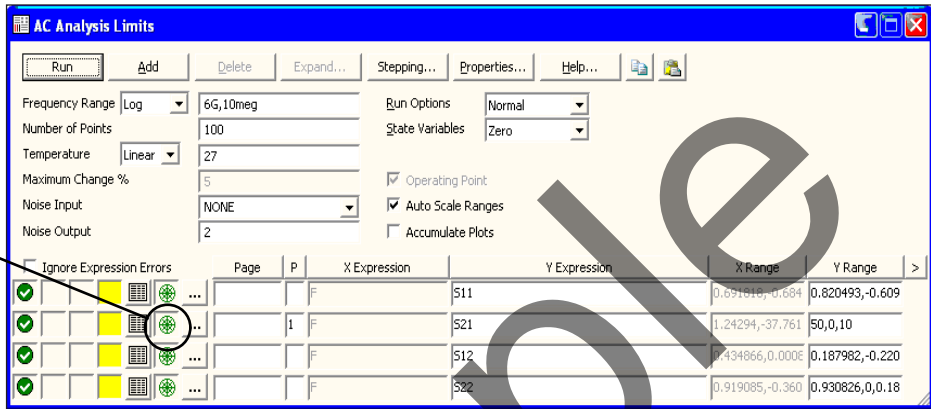


Figure 5-14 Analysis limits for a polar plot

When the polar plot option is selected, the X expression must be F, so this field is not available for editing. Here is the resulting polar plot.

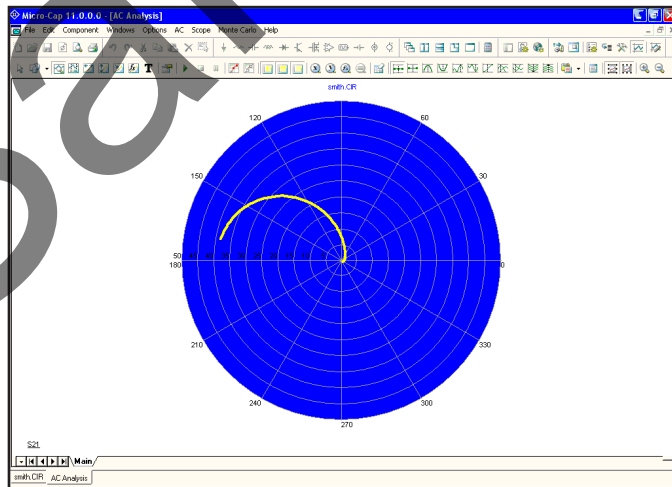
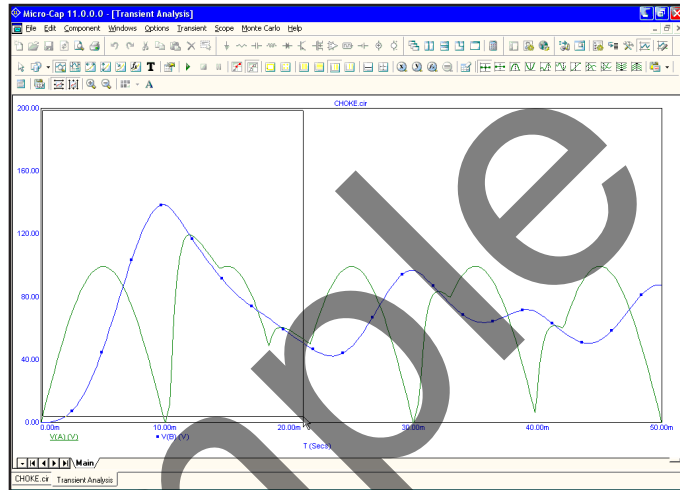


Figure 5-15 Polar plot of S21

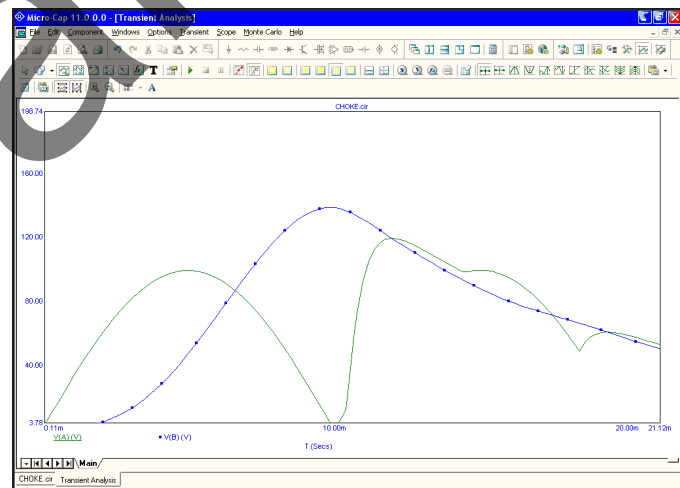
## Magnifying

Click the left mouse button just above and to the left of the large peak of the  $V(B)$  curve. While holding the left button down, drag the mouse down and to the right to create a magnification box like the one shown in Figure 7-3.



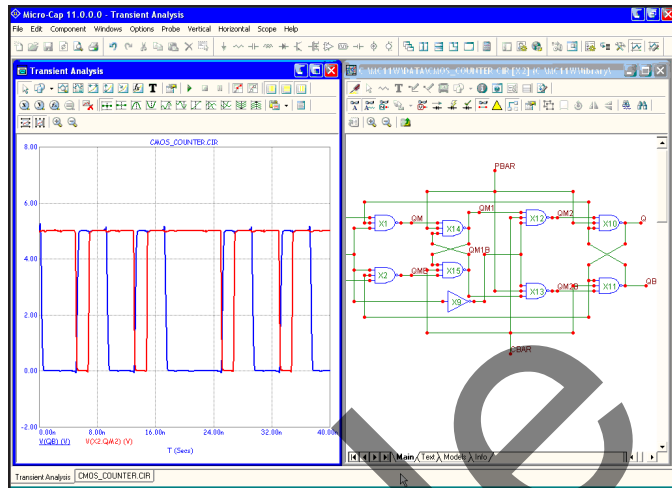
**Figure 7-3 Magnifying a plot region**

Release the button and MC11 scales the plot to match the outline of the box.



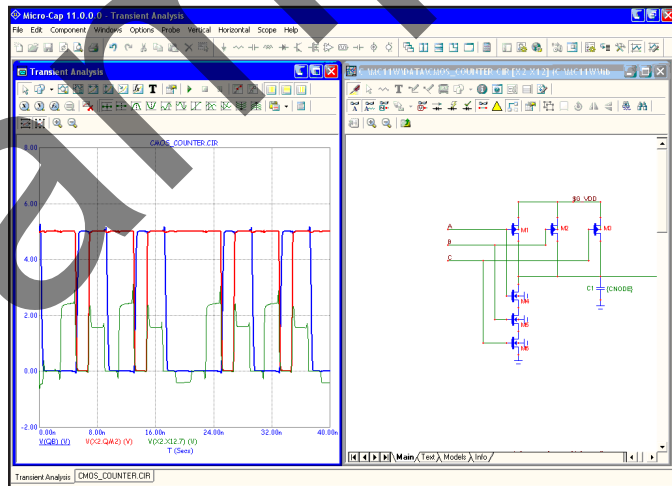
**Figure 7-4 The magnified region**






**Figure 8-13 Probing the second JK**

Click on the X12 NAND3. The system loads and displays its schematic. All of its nodes are available to the external pins, except the internal nodes of the 3-nand stack. Click on one of these, like the drain node of M6. The display should now look like this:

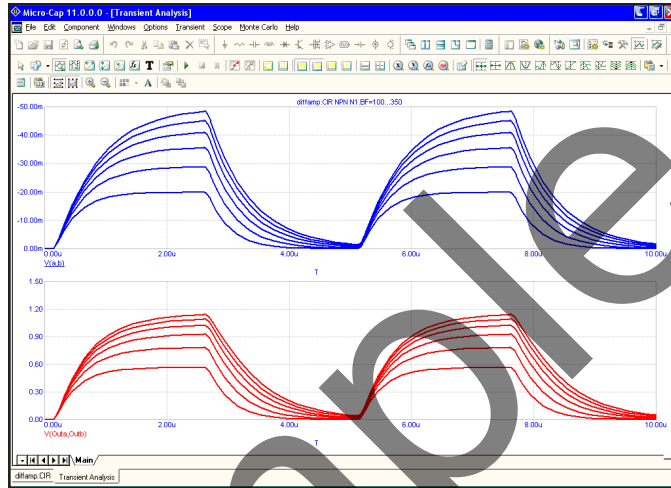


**Figure 8-14 Probing the NAND3**

Suppose you want to return to the JK level. Click on the Up Tool Bar button  or press ALT + Up Arrow. Click on it again and you are back at the top level schematic.

## Stepping in transient analysis

To see how stepping works in transient analysis, enable the Yes option in the Step It section of Parameter panel 1, click OK, and press F2 to start the run. It looks like this:

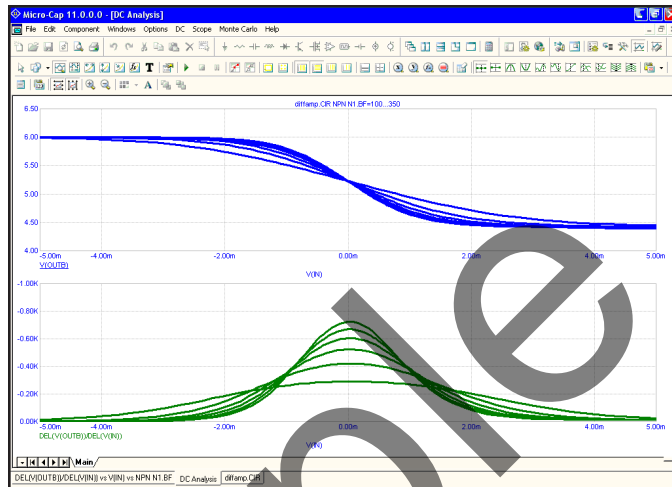


**Figure 9-2 Stepping BF in four transistors simultaneously**

In this run, each device which uses the N1 model, which includes the transistors Q1, Q3, Q4, and Q5, has its BF (forward beta) stepped through the values 100, 150, ... 350. To see which devices use N1, double-click on each device in the schematic to see its Attribute dialog box. From the results, it seems the offset voltage of V(OUTA,OUTB) is insensitive to large changes in BF, if all four devices have the same value.

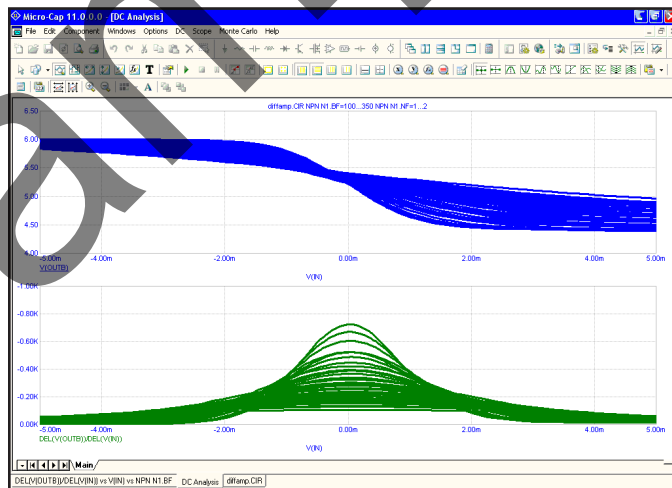
Now we'll step the BF of just Q1 to see how the circuit tolerates small relative differences in beta. Press F11 to access the Stepping dialog box. Click on the Component item in the Parameter Type group. From the Step What list, click on Q1 and from the parameter list select BF. Type "250" in the From field, "255" in the To field, and "1" in the Step Value field. Click on the OK button and press F2 to start the run. The plot, shown in Figure 9-3, demonstrates the circuit sensitivity to differential beta. When Q1's BF changes from 250 to 255, while all other transistors have constant BF, large shifts occur in the output offset of both curves V(A,B) and V(OUTA,OUTB).

To illustrate stepping in DC, press F3 to exit the analysis. Press ALT + 3 for DC analysis. Press F11, click on the Linear option, Click on the Yes option, click OK, then press F2 to start the DC run. It looks like this:



**Figure 9-7 DC analysis linear stepping**

To illustrate multiple parameter stepping, press F11, click in the Yes option in the Step It field of parameter 2. Press ENTER, then F2. The run looks like this:

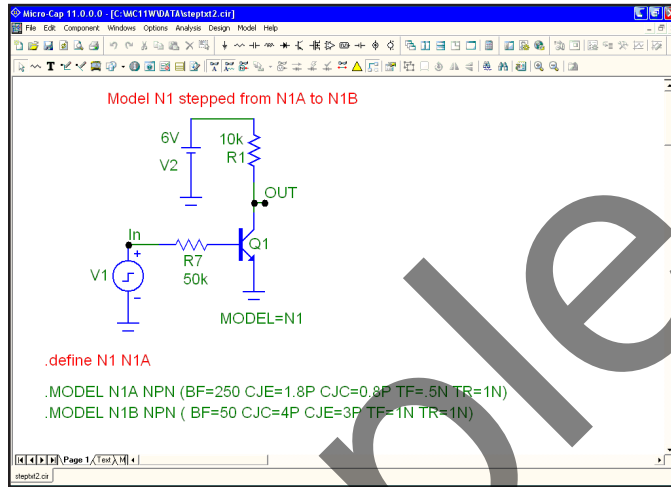


**Figure 9-8 Stepping multiple parameters**

Here we've simultaneously stepped BF and NF of all transistors using the N1 NPN model. Nested stepping was selected, and each parameter has six values, so  $6 \cdot 6 = 36$  runs were done.

## Text stepping

Text can also be stepped. Consider the circuit below:

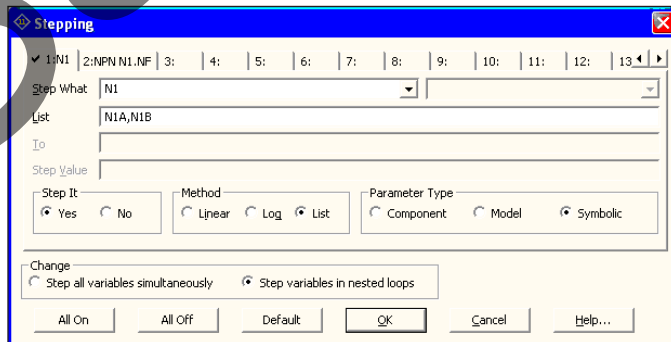


**Figure 9-9 Stepping the N1 variable**

Here Q1's model is set to N1, which is defined as N1A via the statement,

```
.DEFINE N1 N1A
```

If we step N1 through a list that includes N1A and N1B, we get two simulations. One is for the case when N1 is N1A and uses the N1A model parameter set. The second is for the case when N1 is N1B and uses the N1B model parameter set. The Stepping dialog box for this case is shown below.



**Figure 9-10 The Stepping dialog box for text stepping the N1 variable**

## An example

To illustrate the Monte Carlo features, load the file CARLO4.CIR. This file contains a simple pulse source driving an RLC network. The circuit looks like this:

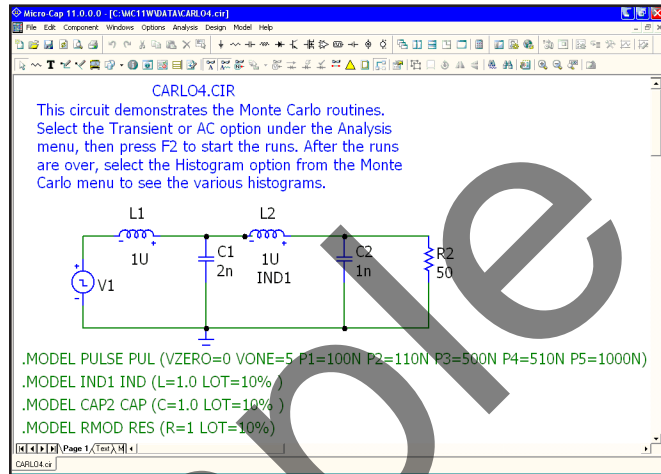


Figure 10-1 The CARLO4 circuit

Select transient analysis and then using the mouse, click the **Options** item from the **Monte Carlo** menu. It should look like this:

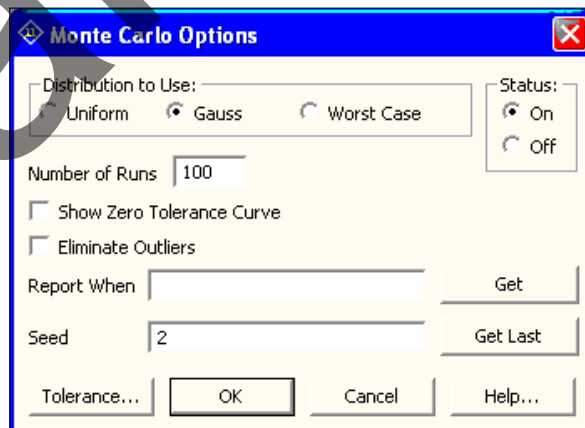


Figure 10-2 Monte Carlo options

## A transient analysis example

To see how the animation mode works in transient analysis, click on **Open** under the **File** menu and load the circuit ANIM3. The ANIM3 circuit uses the seven segment components to display the outputs of three 7448 seven segment decoders. The circuit should appear like this:

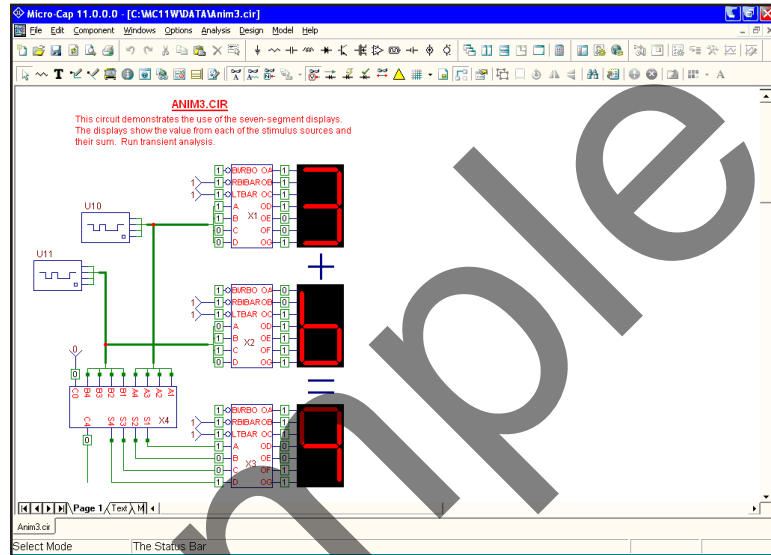


Figure 14-2 The ANIM3 circuit

Select **Transient** from the **Analysis** menu. Close the Analysis Limits dialog box. Click on the **Scope** menu and select **Animate Options**. The Wait setting for the ANIM3 circuit is currently set to Wait for Time Delay and the Time Delay has been defined as 0.2. This means that a single data point will be calculated every 0.2 seconds. Click OK. Hit the F2 key to run the simulation. The resulting values will be displayed in the plot and on the schematic. Notice that both the digital states on the schematic and the seven segment displays are updated with each data point. Figures 14-3 and 14-4 display the simulation results at two separate times in the circuit.

For animate mode, a split screen should be used to be able to view both the waveform plot and the schematic at the same time. This can be done by selecting **Tile Vertical** or **Tile Horizontal** under the **Windows** menu. The schematic can be moved within the window by dragging with the right mouse button.

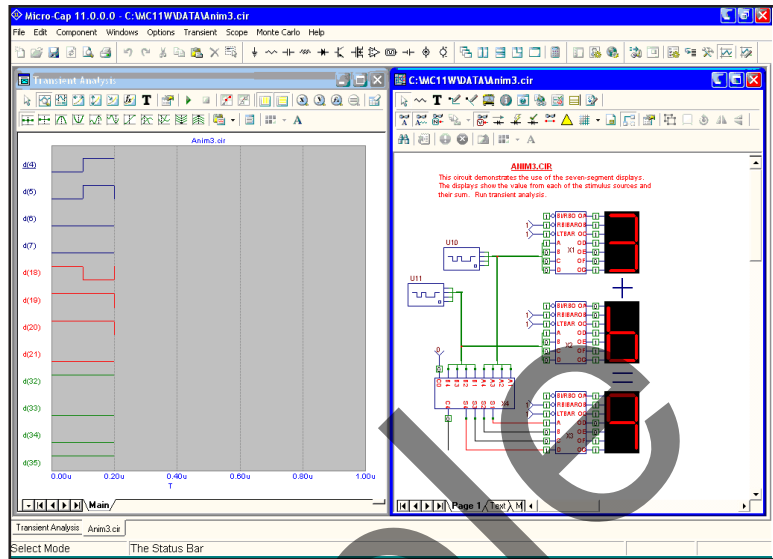


Figure 14-3 Animation mode results at 200ns

The waveforms in the plot also show the output values of the Ina stimulus (nodes 21-18), the Inb stimulus (nodes 28-25), and the resulting output (nodes 35-32) of the adder. As can be seen in the figures, it is much easier to view the basic output through the animation components than through the actual waveforms.

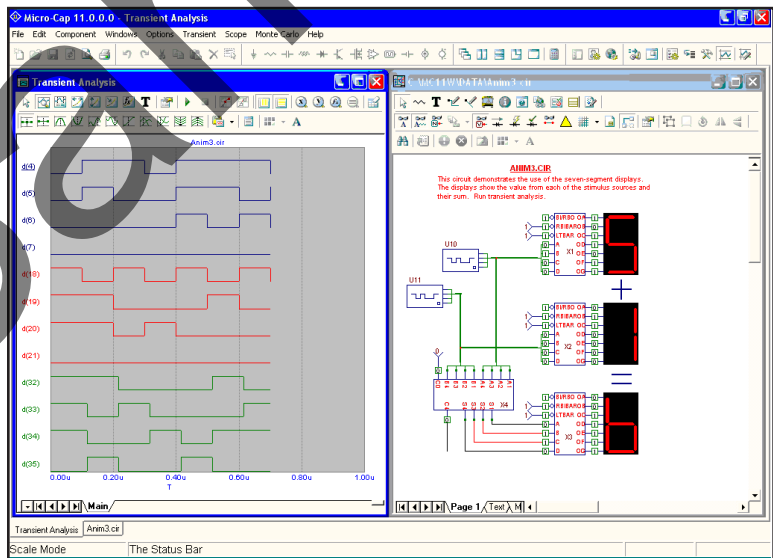


Figure 14-4 Animation mode results at 700ns

## A Dynamic DC example

To see how the animation mode works in Dynamic DC analysis, click on **Open** under the **File** menu and load the circuit ANIM5. The ANIM5 circuit shows how to use the voltage and current meters, analog Bar, DPST switch, and motor. Run Dynamic DC. The circuit should appear like this:

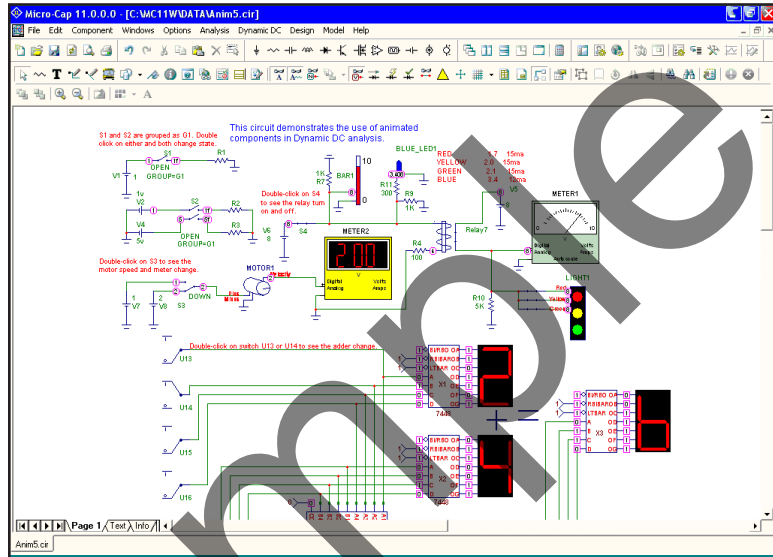


Figure 14-5 The ANIM5 circuit

In Dynamic DC MC11 does a DC operating point calculation every time a change is made to the circuit. It continually updates the display. In this circuit we have a 1.0 volt and a 2.0 volt battery connected through a SPDT switch to a motor that rotates at 1RPM per input volt. The rotational velocity (in volts) is measured by a digital voltmeter. There is also an analog bar, an analog voltmeter, a stoplight, an LED, and seven-segment displays showing the addends into and their sum from a digital 4-bit full adder.

The motor rotates at  $2\text{volts} * 1\text{RPSV} = 2\text{ RPS}$  (revolutions per second). The analog bar is at 8/10 of its full height.

Double click on the S1 switch. It changes state. Because S2 is ganged with it, S2 also changes state.