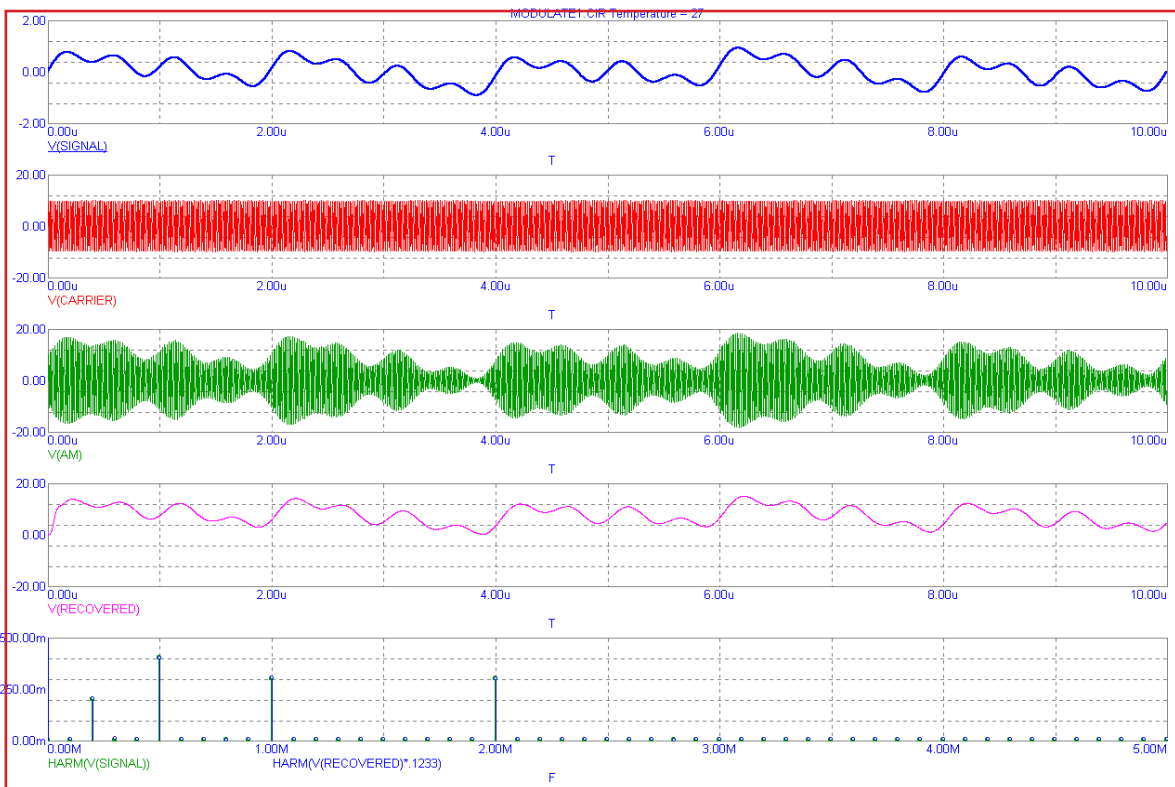# Summer 2000

Modulator / Demodulator Circuit

Featuring:
- Using Probe on a Spice File
- Initializing a Flip-flop and Parts That Use Flip-flops
- How to Build a Modulator / Demodulator

News In Preview

The first article describes how to use the new SPICE Probe feature. It functions much like schematic probing except that you click on part names and node names in a SPICE text file instead of part shapes and node locations in a schematic.

The second article demonstrates how to properly initialize the digital flip-flops and the digital library parts that use them, illustrating several method for insuring a known start-up state.

The last article describes how to build a modulator and a demodulator from basic Micro-Cap 6 components.

Contents

Book Recommendations

Micro-Cap / SPICE
• *Computer-Aided Circuit Analysis Using SPICE*, Walter Banzhaf, Prentice Hall 1989.
ISBN# 0-13-162579-9
• *Macromodeling with SPICE*, Connelly and Choi, Prentice Hall 1992.
ISBN# 0-13-544941-3
• *Semiconductor Device Modeling with SPICE*, Paolo Antognetti and Giuseppe Massobrio
McGraw-Hill, Second Edition, 1993. ISBN# 0-07-002107-4
• *Inside SPICE-Overcoming the Obstacles of Circuit Simulation*, Ron Kielkowski,
McGraw-Hill, First Edition, 1993. ISBN# 0-07-911525-X
• *The SPICE Book,* Andrei Vladimirescu, John Wiley & Sons, Inc., First Edition, 1994.
ISBN# 0-471-60926-9
• *SMPS Simulation with SPICE 3,* Steven M. Sandler, McGraw Hill, First Edition, 1997.
ISBN# 0-07-913227-8
• *MOSFET Modeling with SPICE Principles and Practice,* Daniel Foty, Prentice Hall, First Edition,
1997. ISBN# 0-13-227935-5

German
• *Schaltungen erfolgreich simulieren mit Micro-Cap V,* Walter Gunther, Franzis', First Edition, 1997. ISBN#
3-7723-4662-6

Design
• *High Performance Audio Power Amplifiers,* Ben Duncan, Newnes, First Edition,
1996. ISBN# 0-7506-2629-1

High Power Electonics
• *Power Electronics,* Mohan, Undeland, Robbins, Second Edition,
1995. ISBN# 0-471-58408-8

Micro-Cap 6 Questions and Answers

Question: What is rubberbanding and how do I use it?

Answer: Rubberbanding is a schematic mode that can be enabled or disabled from the Preferences dialog box on the Option menu. When it is enabled and you drag wires or components, they stay connected to their original nodes. If it is disabled, dragging can, and usually does, disconnect components and wires from other nodes.

Rubberbanding is extremely useful in "dressing up" a poorly drawn schematic. You can improve part spacing, loosen up densely packed circuit areas, and generally improve the readability of your schematic with rubberbanding. Just drag select the circuit fragment that you want to move, then left-click on it and drag it to a new position.

When using rubberbanding to edit a circuit, you often arrive at a point where you need to temporarily disable or re-enable the rubberbanding feature. Pressing SHIFT + CTRL + R can be used to rapidly toggle the rubberbanding mode off and on.

Question: What does the "Add DC Path to Ground" on the Preferences dialog box do?

Answer: In order to solve for the DC operating point, it is necessary to have a finite impedance from every node to ground. The necessity for this can easily be seen because a circuit without an impedance to ground has an infinite number of solutions and this poses something of a problem for the simulator. This option avoids the problem by automatically adding a 1/Gmin resistor to any node that is missing the required "path to ground". Usually it is best to disable this option and provide the impedance manually so that you know what node is involved and can better judge how to provide it with a path to ground.

Question: What does the "Floating Nodes Check" on the Preferences dialog box do?

Answer: A floating node is a node to which there is only one part connected. Most SPICE simulators will not run if there are any floating nodes present. The presumption is that there is no point in having only one part connected to the node, so it must be a circuit error. MC6 can simulate circuits containing floating nodes without any problem, so this option is used as a likely error checker.

Question: I'm having trouble using the WMF picture files. Most of the time it works correctly, but occasionally the text I've added to a plot does not scale well when I re-size it in a document. What am I doing wrong?

Answer: Probably nothing. All versions up to 6.1.0 were scaled at the highest resolution in the Y axis only. If you re-sized it in both directions, the text frequently did not scale the same as the plot. As of version 6.1.1 the program has been improved by scaling text in both directions. Version 6.1.1 can be downloaded from our web site anytime after June 19.

Easily Overlooked Features

This section is designed to highlight one or two features per issue that may be overlooked because they are not made visually obvious with an icon or a menu item.

Tiling Windows
Sometimes when you are pondering a waveform and wondering why it came out the way it did you need to see the schematic. If the simulation is still in progress or even if it is complete, you can take a quick look at the schematic by tiling the windows. Click on the Tile Vertical or Tile Horizontal buttons and the program will tile the schematic and analysis plot windows. You can scroll the schematic to view areas outside the window.

Viewing the numeric values while the simulation is in progress
Sometimes, especially during long simulation runs or runs in which you are only displaying the last 10% of the run to avoid initial transients, you may want to see what is happening with the simulation but the plot doesn't show much yet. You can peek at what is happening numerically using the "P" key. Pressing "P" toggles the numeric display of the waveform values directly on the plot. Be sure to toggle it off when you are finished looking, as it considerably lengthens the simulation run time in small to medium circuits.

There is another way to see the numeric values during a run. You can invoke the State Variables Editor prior to the run with F12. Then start the run (with F2) without removing the Editor. The State Variables editor display will be constantly updated as the simulation progresses. You can also use the Animate dialog box to insert a key press wait after each simulation step if you want to cycle slowly through the simulation states.

After the run you can view any numeric output that you selected with F5 and toggle back to the plot with F4.

Using Probe on a Spice File

Until very recently, you couldn't use the Probe feature on a SPICE file. It could only be used with a schematic. Well for you diehard SPICE fans, Probe is now available for SPICE files as well. Here's how it works.

You open a SPICE file, select Probe Transient, Probe AC, or Probe DC. MC6 presents an abbreviated Analysis Limits dialog box to let you change the simulation parameters. After making any changes you might want, click on the Close button. MC6 runs the simulation, saves the results to disk, and waits for a mouse click.

Only voltage and current are available. The Vertical menu lets you select which.

If voltage is selected from the Vertical menu:

Analysis Type	Node Number Click	Device Name Click
DC or Transient	Node voltage	Pin-pin voltage
AC	db(AC node voltage)	db(pin-pin voltage)

If current is selected from the Vertical menu:

Analysis Type	Node Number Click	Device Name Click
DC or Transient	Nothing	Terminal currents
AC	Nothing	db(Terminal currents)

When you click on a part, MC6 lists the voltages and/or currents available from the part and you select one.

To illustrate, consider this circuit:

```
C1 0 B 2UF
D1 3 A D0
D2 1 A D0
L1 B A 5
R1 0 B 10K
R2 0 A 10K
V1 1 0 SIN (0 100 50 0 0 0)
V2 0 3 SIN (0 100 50 0 0 0)
.MODEL D0 D (IS=10F CJO=10P)
.OPTIONS ACCT LIST OPTS  ABSTOL=1pA CHGTOL=.01pC CPTIME=1G DEFL=100u DEFW=100u
+ DIGDRVF=2 DIGDRVZ=20K DIGERRDEFAULT=20 DIGERRLIMIT=10000 DIGFREQ=10GHz
+ DIGINITSTATE=2 DIGIOLVL=1 DIGMNTYMX=2 DIGMNTYSCALE=0.4 DIGOVRDRV=3
+ DIGTYMXSCALE=1.6 GMIN=1p ITL1=100 ITL2=50 ITL4=10 ITL5=0 LIMPTS=0 PIVREL=1m
+ PIVTOL=.1p RELTOL=1m TNOM=27 TRTOL=7 VNTOL=1u WIDTH=80
.TRAN 0.00263158 0.05 0 0.0002
.TEMP 27
.END
```

If you run Probe Transient on this circuit and click on the "A" node name, you get a plot of the node voltage waveform on node A. Probe on the D1 name and you get a plot of the voltage across D1. The two plots look like Fig. 1.
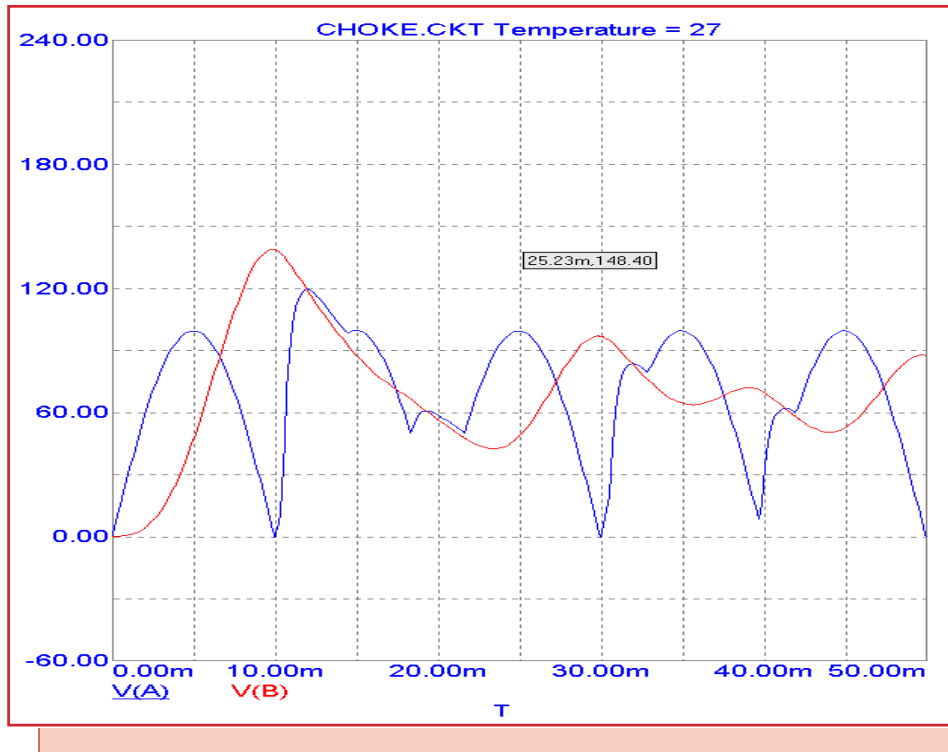
Fig. 1 - SPICE voltage probing

If you select Current from the Vertical menu and then click on the D1 name, you get a plot of the current through the diode D1. Likewise, click on the C1 name and you get a plot of the curremt through the capacitor C1. The two plots look like Fig. 2
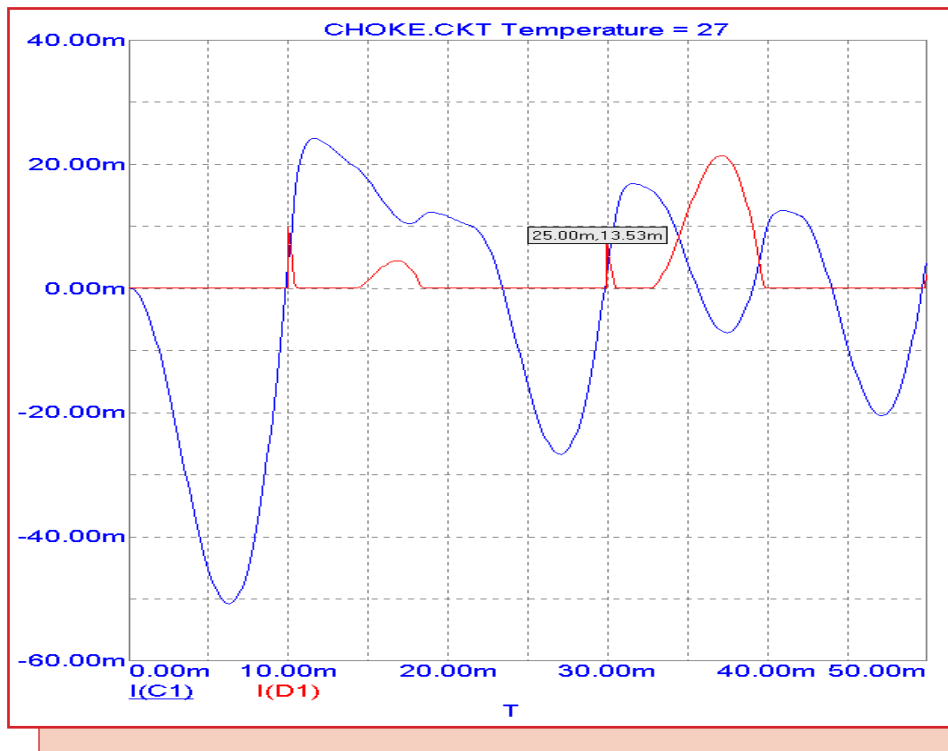

Fig. 2 - SPICE current probing

As an example of probing AC consider this circuit, translated to SPICE from the UA709 sample schematic.

```
C1 19 4 5NF
C2 24 21 200PF
Q1 2 5 6 QNL
Q2 2 6 7 QNL
Q3 4 9 10 QNL
Q4 5 11 10 QNL
Q5 8 8 7 QNL
Q6 12 13 7 QNL
Q7 12 4 13 QNL
Q8 15 15 16 QNL
Q9 10 15 17 QNL
Q10 1 12 20 QNL
Q11 23 21 22 QNL
Q12 16 23 24 QPL
Q13 1 23 24 QNL
Q14 1 2 3 QNL
Q15 21 7 18 QPL
R1 5 3 25K
R2 14 7 3.6K
R3 16 17 2.4K
R4 18 14 10K
R5 8 13 3K
R6 19 12 1.5K
R7 1 12 10K
R8 21 22 10K
R9 0 11 1K
R10 25 9 1K
R11 9 24 100K
R12 15 14 18K
R13 6 8 3K
R14 16 22 75
R15 24 18 30K
R16 18 20 1K
R17 1 23 20K
R18 1 2 10K
R19 4 3 25K
V1 1 0 15
V198 0 16 15
V709 25 0 AC 1 SIN 0 10M 10K
.MODEL QNL NPN (BF=80 VAF=50 RB=100 CJE=3P CJC=2P CJS=2P TF=300P TR=6N)
.MODEL QPL PNP (BF=10 CJC=4P CJE=6P RB=20 DEV=50% VAF=50 TF=1N TR=20N)
.OPTIONS ACCT LIST OPTS  ABSTOL=1pA CHGTOL=.01pC CPTIME=1G DEFL=100u DEFW=100u
+ DIGDRVF=2 DIGDRVZ=20K DIGERRDEFAULT=20 DIGERRLIMIT=10000 DIGFREQ=10GHz
+ DIGINITSTATE=2 DIGIOLVL=1 DIGMNTYMX=2 DIGMNTYSCALE=0.4 DIGOVRDRV=3
+ DIGTYMXSCALE=1.6 GMIN=1p ITL1=100 ITL2=50 ITL4=10 ITL5=0 LIMPTS=0 PIVREL=1m
+ PIVTOL=.1p RELTOL=1m TNOM=27 TRTOL=7 VNTOL=1u WIDTH=80
.AC DEC 6 1 1e+008
.TEMP 27
.PLOT AC VDB(24)
.PRINT AC VDB(24)
.END
```

Fig. 3 -SPICE circuit for Probe AC

Here is the result of clicking on any of the references to the output node 24. The voltage on node 24 is shown as DB(V(24)) and its phase is shown as PH(V(24)). The plots have been selected after disabling the Same Y scales option on the Scope menu.
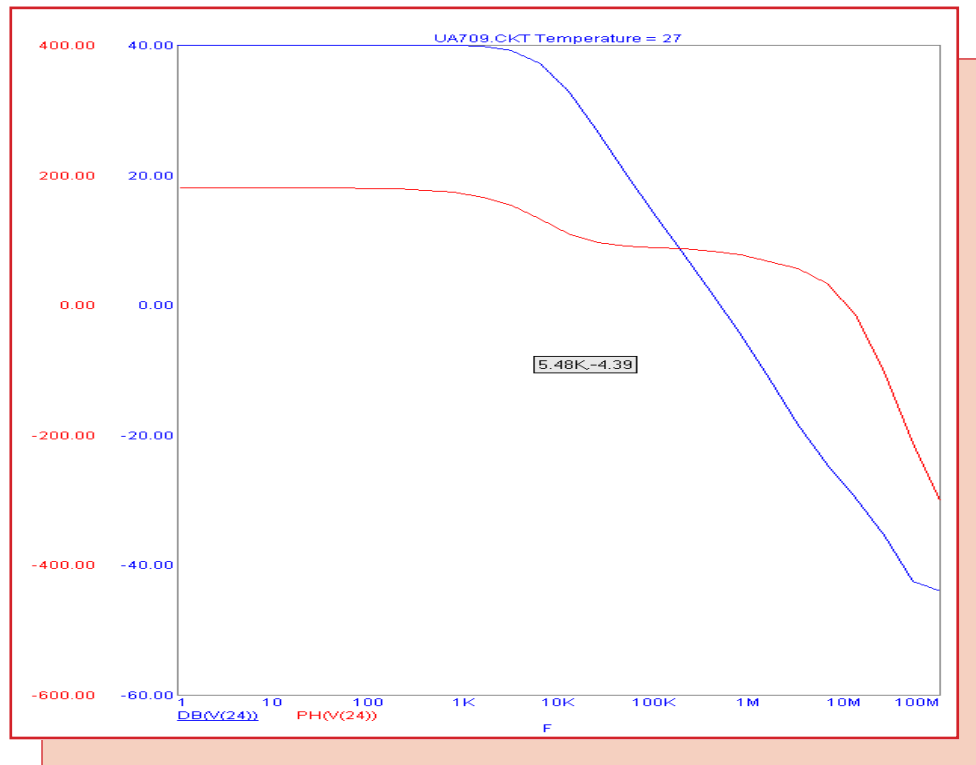

Fig. 4 -SPICE AC Gain and Phase probing

All of the features of schematic probing are available in SPICE probing except that only current and voltage can be "probed", although they can be manually added with the "Add Plot" command from the Probe menu.

Initializing a Flip-flop and Parts That Use Flip-flops

Flip-flops must be properly initialized before meaningful simulation results can be obtained. There are two general approaches to initialization:

- Using a STIM device to reset or preset the flip-flop during the first few nanoseconds of the run.

- Changing the DIGINITSTATE value to control the flip-flop initialization state.

Flip-flops are set to a value dictated by the value of DIGINITSTATE, which is set from the Global Settings menu, or by a .OPTIONS DIGINITSTATE=<value> command somewhere in the circuit. The initial state is set according to this table:

DIGINITSTATE	FLIP-FLOP INITIAL STATE
0	0
1	1
2 or any other value	X (default)

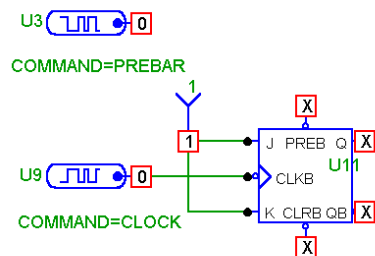To illustrate the first method, consider this circuit:

Fig. 5 -A flip-flop circuit without initialization

Since there is no signal on the PREB or the CLRB input, and the flip-flop is being initialized to X, the output stays at X forever. Now consider this circuit:
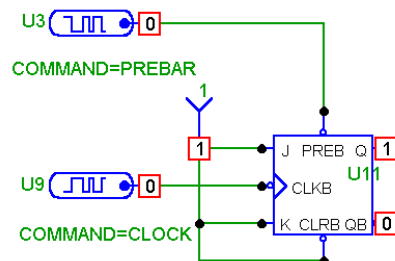

Fig. 6 -A flip-flop circuit initialized to Q=1

Here we have connected an initializing pulse to the PREB input and connected the node with a fixed digital "1" to the CLRB (active low) input. As a result the flip-flop output initializes to "1" due to the "0" on the PREB (active low) input from the U3 STIM device. We could just as easily initialized the flip-flop output to "0" this way:
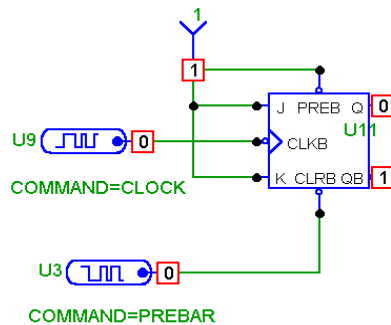

Fig. 7 -A flip-flop circuit initialized to Q=0

Here we have connected an initializing pulse to the CLRB input and connected the node with fixed digital "1" to the PREB (active low) input. As a result the flip-flop output initializes to "0" due to the "0" on the CLRB (active low) input from the U3 STIM device.

Note that we have used a digital STIM device and a fixed digital value to do the initialization. You could also use an analog pulse generator or an analog voltage as well. However, all-digital simulations are always faster than mixed-mode simulations, though for small circuits the differences are not significant.

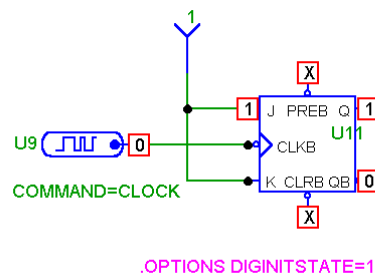To illustrate the second initialization method, consider this circuit:

Fig. 8 -A flip-flop circuit initialized to Q=1 using DIGINITSTATE

In this circuit there are no preset or reset signal sources on the PREB and CLRB pins. They aren't needed since the flip-flop is initialized to "1' by the command statement:

.OPTIONS DIGINITSTATE=1

This statement can be entered as grid text on the schematic, as shown here, or placed in the text area. The value of DIGINITSTATE can also be modified in the Global Settings, but doing that extends its effect to all circuits, whereas a command statement affects only the local circuit.

This general discussion of how to initialize flip-flops also applies to more complex parts that use flip-flops internally. For example, consider the CD4040B part which uses an internal bank of twelve flip-flops:
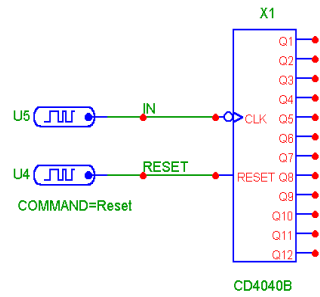

Fig. 9 - Initializing a complex part using a reset pulse

Here we have initialized the CD4040B using a reset pulse that is "1" for the first 100ns and then changes to "0" for the remainder of the run as can be seen from the transient plot (well the "0" part can be seen anyway).
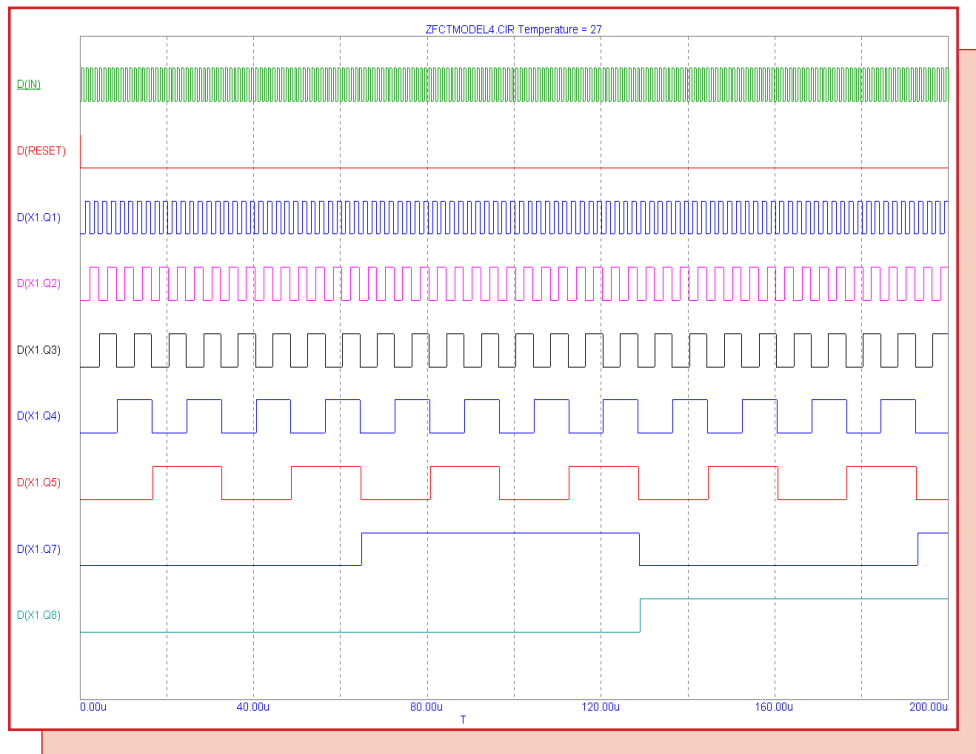

Fig. 10-The transient analysis run

Now consider the same circuit with the reset pulse removed and a command statement to set the value of DIGINITSTATE as follows:
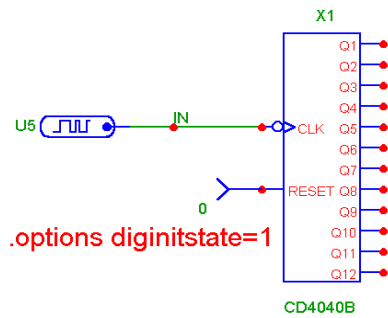

Fig. 11 - Initializing a complex part using DIGINITSTATE

Here we have initialized the CD4040B by setting the internal flip-flop states to an initial value of "1". Note that the Reset input must be set to an inactive state (0) using the fixed digital value device. Otherwise its initial X state would not allow counting.

Here is the transient analysis. Note that the run is slightly different that the preceding run. Here there is no reset to "0". Instead, all flip-flops are set to "1" as a result of the .options diginitstate=1 statement. On the first negative clock edge they all count to "0".
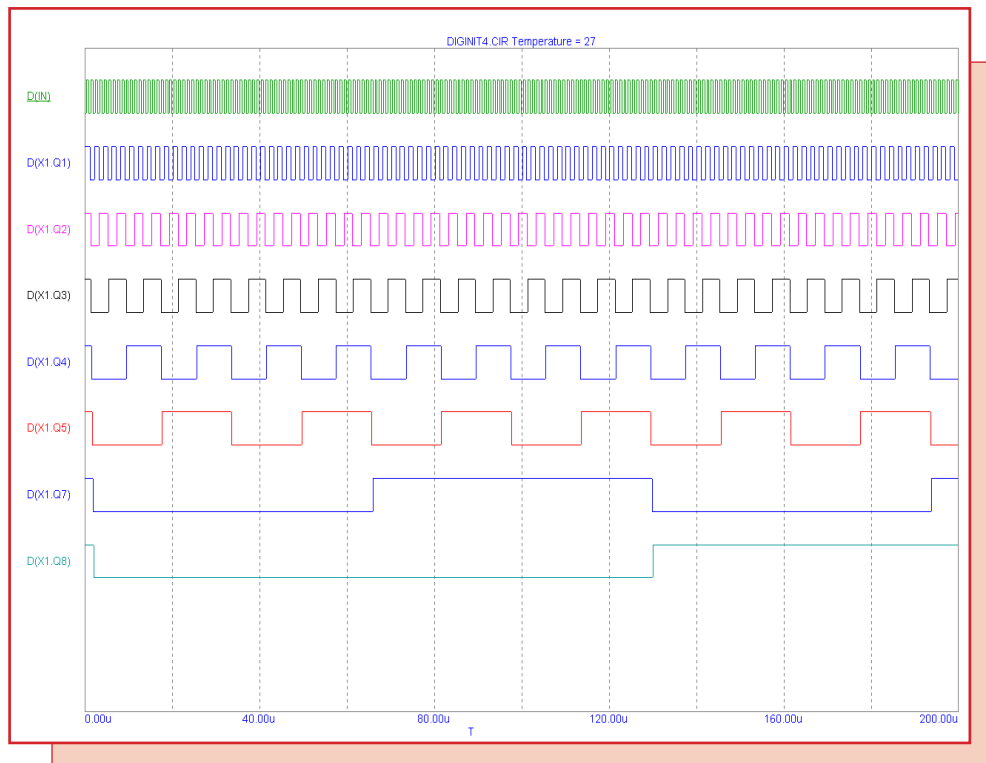

Fig. 12 - The transient analysis run

How to Build a Modulator /Demodulator

Ever wanted to build a modulator /demodulator circuit? It's simple. We'll demonstrate with this sample circuit:

Communications Circuits: AM Modulation/Demodulation

To create a modulated signal, multiply a signal source waveform and a carrier source waveform using the MUL macro, and add the carrier waveform using the SUM macro. These macros can be found at Component / Analog Primitives / Macros. This generates the amplitude modulated (AM) signal:

carrier(t) + carrier(t)*signal(t)

The AM signal can be recovered with a demodulator that consists of a diode detector and a low pass filter.

.2*SIN(2*PI*2E5*t)+.4*SIN(2*PI*.5E6*T)+.3*SIN(2*PI*1E6*T)+.3*SIN(2*PI*2E6*T)
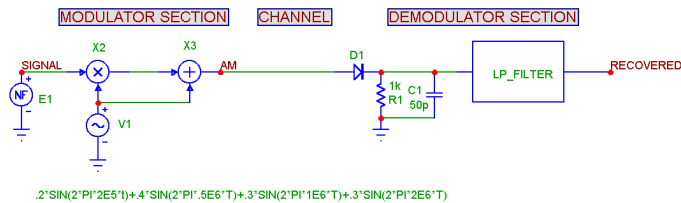
Fig. 13 -Modulator/Demodulator

In this circuit an NVF source is used to generate a test signal comprised of four sinusoids of frequency 200Khz, 500Khz, 1MHz, and 2MHz. The carrier sinusoid is supplied by a 50MHz sine source. The modulator is constructed from the MUL (Multiplier) macro, which is comprised of an
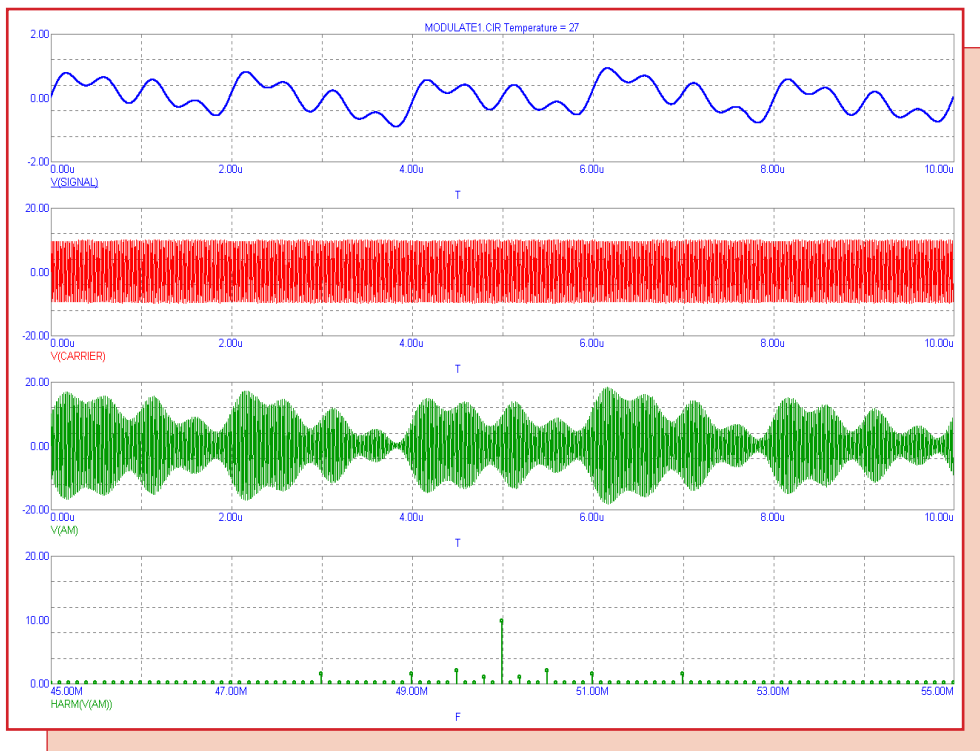
Fig. 14 -Spectral components of the AM signal

NFV source that simply multiplies the two signals together. The demodulator is composed of a simple diode detector, followed by a low pass filter.

Here is the transient analysis, showing the input signal, the carrier, the amplitude modulated output (AM), and spectral components of the AM signal. The signal spectrum is centered at the carrier frequency of 50 MHz, with sideband components at +- 100KHz, +-500KHz, +-1MHz, and +-2MHz.

Here is the complete transient analysis, showing the input signal, the carrier, the amplitude modulated output (AM), and the recovered signal. The last plot shows the spectral components of the original signal and the recovered signal occur at the same frequency, and differ in magnitude by a factor of .1233.
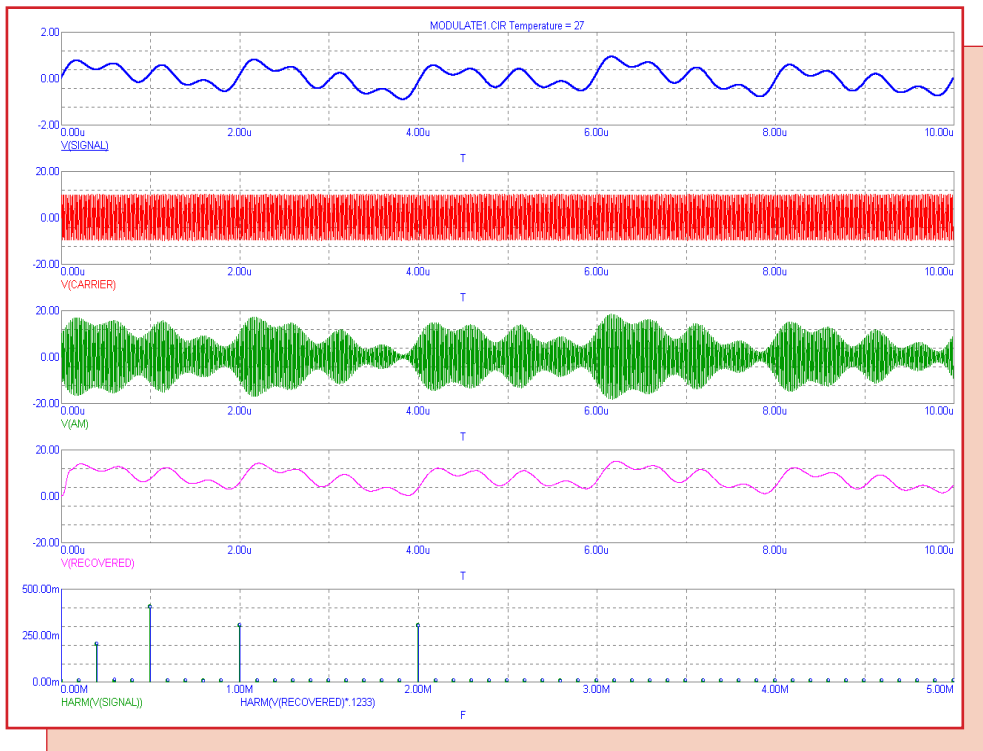
Fig. 15 -Spectral components of the original and recovered signals

The parts used in this circuit are:

Part	Location on Component menu
MUL macro	Analog Primitives/Macros
SUM macro	Analog Primitives/Macros
SINE source	Analog Primitives/Waveform Sources
NFV source	Analog Primitives/Function Sources
Diode	Analog Primitives/Passive Devices
Resistor	Analog Primitives/Passive Devices
Capacitor	Analog Primitives/Passive Devices
LP Filter	Created by the MC6 Active Filter program

Product Sheet

Latest Version numbers
Micro-Cap 6 .... Version 1.1.0
Micro-Cap V .... Version 2.1.2

Spectrum's numbers
Sales .... (408) 738-4387
Technical Support .... (408) 738-4389
FAX .... (408) 738-4702
Email sales .... sales@spectrum-soft.com
Email support .... support@spectrum-soft.com
Web Site .... http://www.spectrum-soft.com

Spectrum's Products
• Micro-Cap 6 LAN version (each, with a 2 seat min) . $3595.00
• Micro-Cap 6 Standard version $3595.00
• Upgrade from MC5 Ver 2 to MC6 .... $500.00
• Upgrade from MC5 Ver 1 to MC6 .... $750.00

Prices are subject to change. You may order by phone or mail using VISA, MASTERCARD, or American Express. Purchase orders accepted from recognized companies in the U.S. and Canada. California residents please add sales tax.