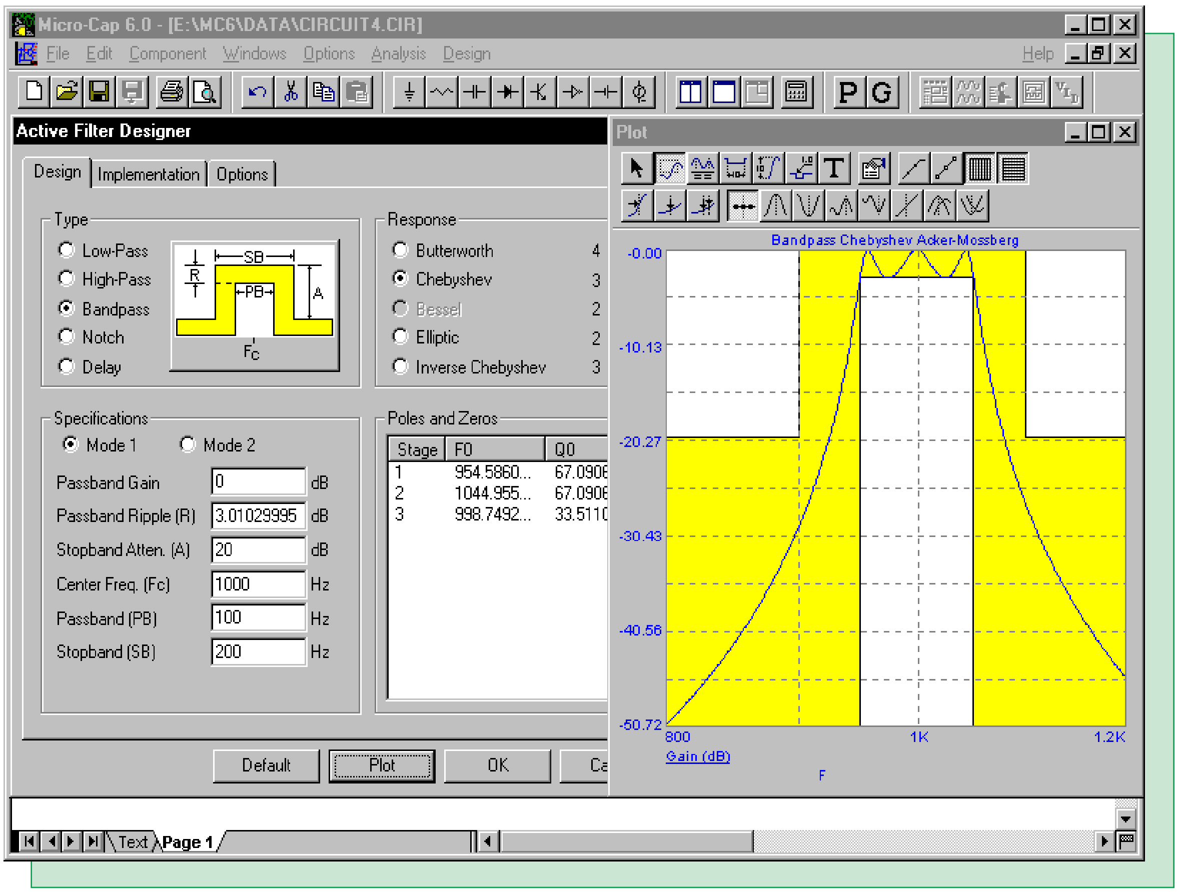
# spectrum news

Applications for Micro-Cap™ Users

## Spring-Summer 1999

## Introducing Micro-Cap 6




Featuring:

- Introducing Micro-Cap 6
- Table Defined Resistance
- Digital vs Analog Pullup Resistors
- Perfect Transformer vs Ideal Transformer

## News In Preview

This issue introduces the next generation of Micro-Cap simulators, Micro-Cap 6. This version of the program has added such features as an active and passive filter designer, the capability to export PCB netlists, the BSIM 3 Ver 3.2 MOSFET model, transfer function analysis, sensitivity analysis, a dynamic DC analysis, GMIN stepping, an expanded DC analysis capability, automatic macro creation and much more. The second article describes the use of the table operator within a resistor to produce a time dependent, table defined resistance. The third article describes the use of the pullup digital resistances and when these must be used in place of the standard analog resistor. Finally, the last article details the difference between an ideal transformer and the perfect transformer that the Micro-Cap transformer model is based on.

## Contents

News In Preview .... 2
Book Recommendations .... 3
Micro-Cap V Question and Answer .... 4
Easily Overlooked Features .... 5
Introducing Micro-Cap 6 .... 6
Table Defined Resistance .... 15
Digital vs Analog Pullup Resistors .... 17
Perfect Transformer vs Ideal Transformer .... 20
Product Sheet .... 22

## 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

## Micro-Cap V Question and Answer

**Question:** Whenever I try to enter an analysis, I keep getting the error: "Can't find file: Nom.Lib". This prevents me from simulating my circuit. How do I get around this?

**Answer:** This can occur if the circuit you are attempting to simulate does not reside in the default DATA directory. The problem is that MC5 is looking only in the current directory for the libraries, and if the libraries aren't in this directory, then it will not be able to find them. The way to fix this is to use the environmental variable, MC5DATA, which should have been set when MC5 was installed if your system allowed it. If MC5 can't find the libraries in the current directory, it checks the environmental variable for alternative paths. The procedure for setting the environmental variable is as follows:

For Windows 3.1, Windows 95, and Windows 98
Open up your Autoexec.bat file in a text editor. This file resides on the C: root directory. Add this line inside your Autoexec.bat on a new line:

Set MC5DATA=C:\MC5\DATA

where C:\MC5\DATA is the directory the libraries are stored in. The default directory that the installation stores the libraries in is the DATA directory so in most cases, the path specified should lead to where the DATA directory was installed. More than one directory may be specified for this variable, with the paths being separated by a semicolon. The system would then need a full reboot.

For Windows NT 4.0
Click on the Start menu. Go to Settings. Click on Control Panel. Double click on the System icon. When the System dialog box appears, click on the Environment tab. At the bottom of the Environment section, there is a list of User Variables. The MC5DATA variable needs to be added to this list. Below the list are two text fields called Variable and Value. In the Variable field, type in:

MC5DATA

In the Value field, the path to the DATA directory needs to be specified such as:

C:\MC5\DATA

Again, multiple paths may be specified by using the semicolon delimiter. Click on the Set command button. Once this is done, the MC5DATA variable and value should appear in the User Variables list. Click OK. If MC5 was loaded during the procedure, it needs to be restarted. Otherwise, loading MC5 should enable the variable.

In MC5, it is possible to see what value the MC5DATA variable has been set to. In the schematic area, hit Alt+Z and a Statistics window will appear. One of the lines in this window will specify the MC5DATA value.

## 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.

### Changing the Format on the Analysis Scales

The default format for the X and Y scales in an analysis are for them to be specified with two digits to the right of the decimal point. In some situations, this may not provide enough accuracy. A waveform that has very little movement in the Y direction or a waveform that has been zoomed in on may need more digits to view meaningful values on the scale. In fact, if a waveform is zoomed in on enough, eventually all of the values on the scale will be the same because the change in the displayed waveform will be too small for the scale format to display.

The scale format may be changed within the Analysis Characteristics dialog box. Simply double click in the plot while in Select mode or hit the F10 hotkey to invoke the Characteristics dialog box. In the dialog box, click on the Format command button. The settings for both the X and the Y scales will appear. The X and the Y scales are each controlled separately by the Scale Format text field in each section. This text field value must be a single integer that specifies the number of digits to the right of the decimal point. Editing the field will change the scale for all waveforms that share that scale. Clicking on the Format All command button will set the scales for all waveforms to the values shown on the screen.

The default Scale format for new circuits can be edited in the Preferences which is located under the Options menu. In the Preferences dialog box, click on the Format option in the list. The scale format may be changed by editing the value in the Default Plot Scales text field.

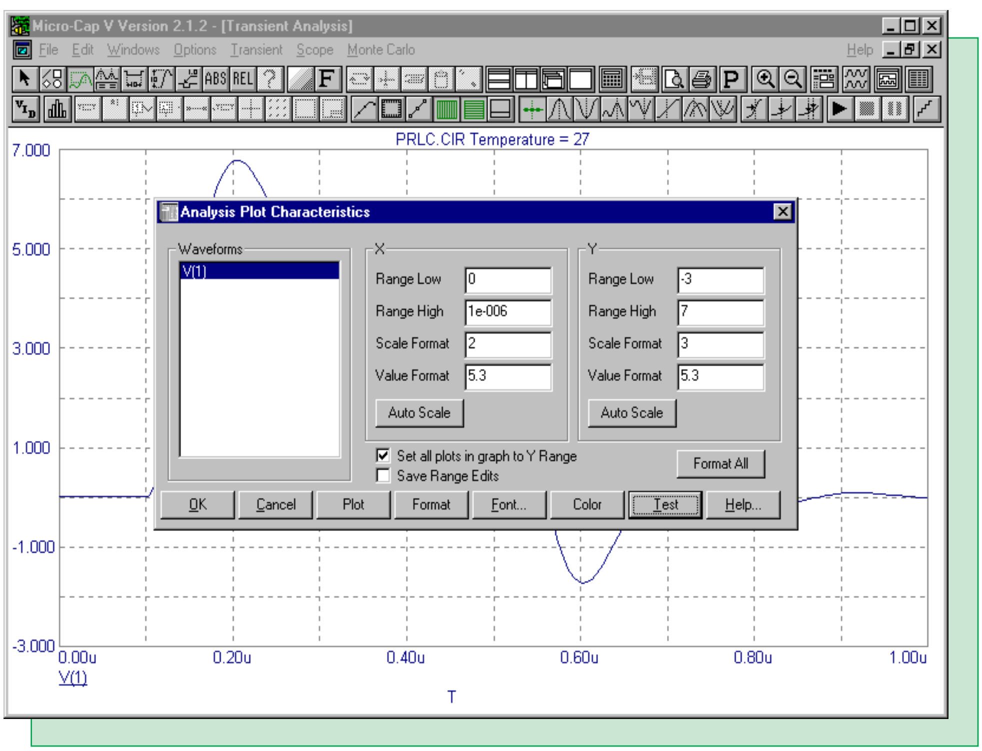



*Fig. 1 - Analysis Characteristics Dialog Box*

## Introducing Micro-Cap 6

The next generation of Micro-Cap, Micro-Cap 6, will be available in July. This latest version has added many more features and improvements in both the schematic and analysis area. A preview of some of the features follows. Upon request, all purchases of Micro-Cap V Version 2.0 between June 1, 1999 and the introduction date of Micro-Cap 6 will receive a free upgrade. Contact our sales department at sales@spectrum-soft.com for upgrade or pricing questions.

**Active and Passive Filter Designers**

Micro-Cap 6 offers new design features for active and passive filters. The active filter designer supports low pass, high pass, bandpass, notch, and delay filters that can be set up with a Butterworth, Chebyshev, Bessel, Elliptic, or Inverse-Chebyshev response. The possible implementations available are Sallen-Key, MFB, Tow-Thomas, Fleischer-Tow, KHN, Acker-Mossberg, and Tow-Thomas 2. The active filter designer appears in Figure 2. The passive filter designer supports low pass, high pass, bandpass, and notch filters than can be set up with a Butterworth or Chebyshev response. The two implementations for passive filters are standard and dual.

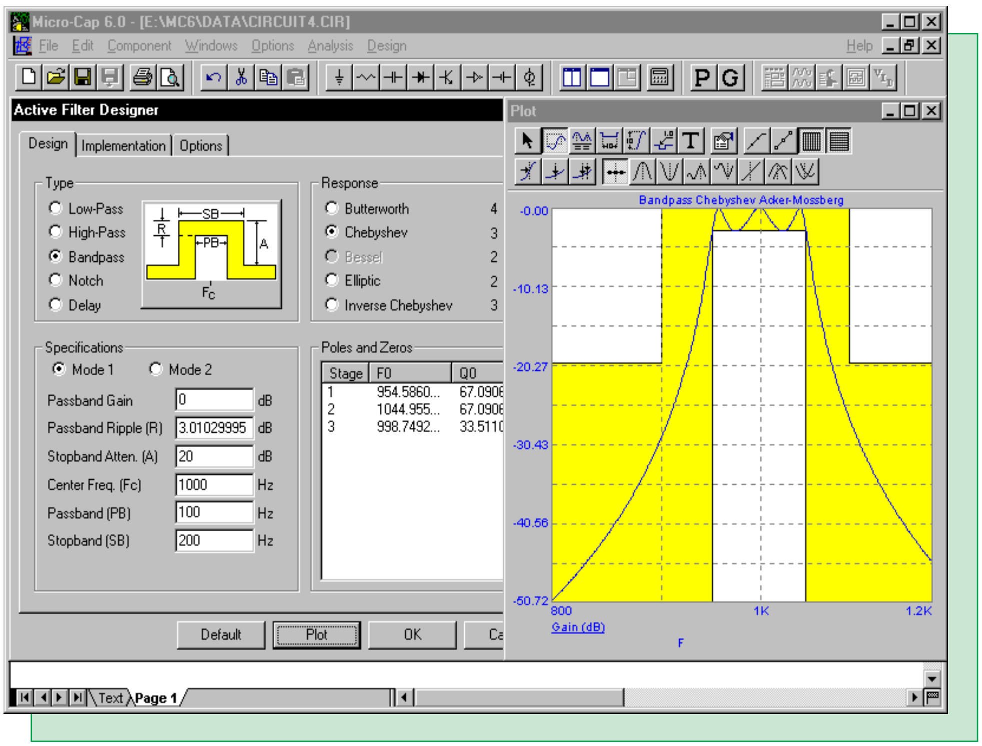



*Fig. 2 - Active Filter Designer*

The created filter can be sent to a new circuit, to a currently loaded circuit, or can be made into a macro component that is easily accessed through the Component menu. The designer also offers the user the choice of opamps to use for an active filter and can produce the filters with either exact impedance values or standard impedance values.

## Exporting PCB Netlists

Micro-Cap 6 has the capability to convert schematics into text netlists that can be imported into PCB layout programs. There are four possible PCB layout netlist formats that the schematic can be converted into: Protel 1, Protel 2, Accel, and Orcad PCB II. These netlists can then be imported directly into a PCB layout program that can perform layout and routing on the circuit. This feature saves the problem of having to create the schematic in two separate programs. Simply create the schematic and run the simulation in Micro-Cap, export the schematic to a netlist, and then import that netlist into a PCB program that can accept it.

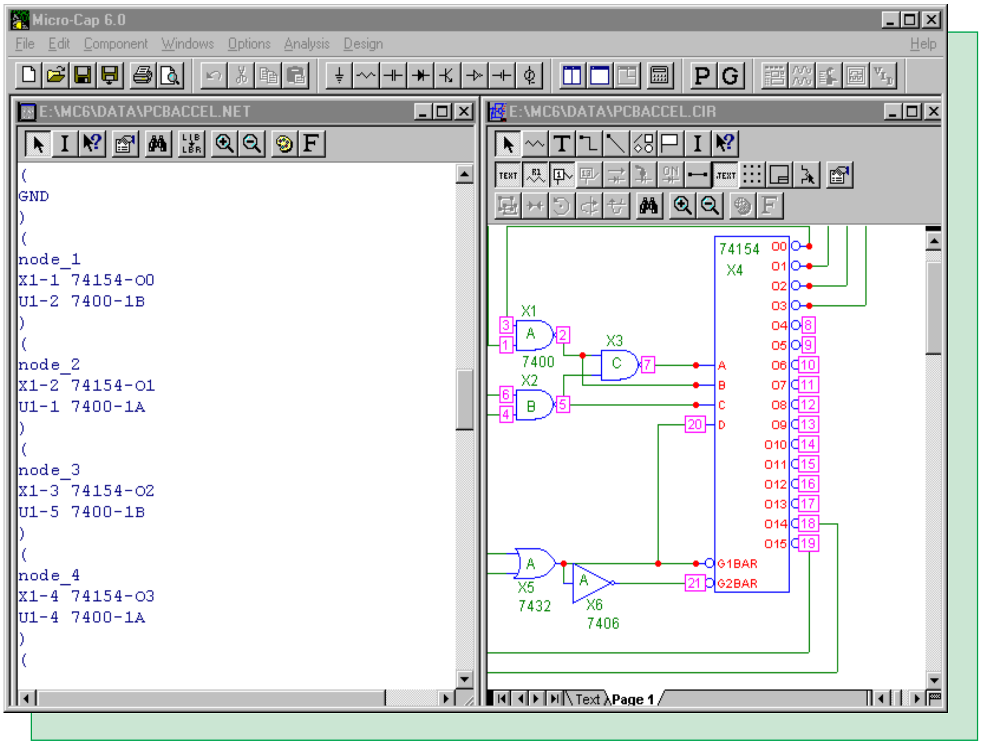



*Fig. 3 - PCB Netlist Output*

Over 10500 of the components in the Component library have at least one package defined for them. The packages can be chosen when the component is placed in the schematic. After placement in the schematic, click on the Package attribute drop down arrow and a list of available packages will be shown. Each of the components has its package pin configuration defined for a package within the Package library.

The Package library is controlled by the Package editor which lets the user add or edit the package information for any of the components within Micro-Cap. The Package editor lets you define basic packages and complex packages. Complex packages are when more than one instance of the component is present within the package. This editor links the pins used in the schematic to the pins that will be used in the PCB netlist.

**Sensitivity and Transfer Function Analysis**
Two new analyses have been added to Micro-Cap 6. These are the transfer function analysis and sensitivity analysis. These analyses are similar to the .TF and .SENS SPICE commands.

The transfer function analysis calculates the small-signal DC transfer function from a specified input source to a specified output expression. Depending upon the input source and the output expression, the transfer function calculated may be the voltage gain, the current gain, the transconductance, or the transadmittance. To measure the transfer function, Micro-Cap makes a small change in the input source and measures the change in the specified output expression. The ratio of these two changes produces the transfer function. Along with the transfer function, the input impedance at the source, and the output impedance at a specified node voltage expression will be calculated.

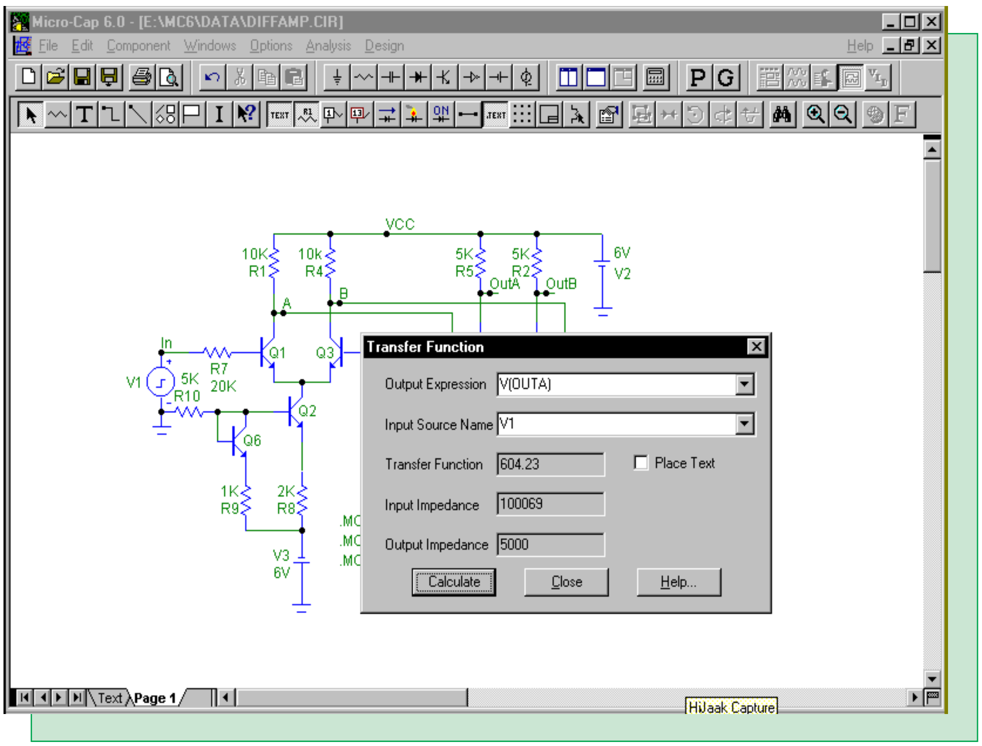



*Fig. 4 - Transfer Function Analysis*

Sensitivity analysis calculates the small-signal DC sensitivity of one or more output expressions to one or more input variables. This analysis determines the DC sensitivity of the specified output expression to any changes in the specified circuit parameters. Sensitivity is defined as:

Change in an output expression / Small change in an input variable

It produces two output values: the absolute sensitivity expressed as a pure ratio and the percentage change sensitivity ratio. Sensitivity analysis is like transfer function analysis, except that it calculates the sensitivity of almost any DC expression to almost any circuit parameter. Circuit parameters for sensitivity analysis include component values, model parameters, and symbolic parameters.

## Dynamic DC Analysis

A new schematic operating point display analysis has been added to Micro-Cap 6. In previous versions of Micro-Cap, to see the DC operating point voltages on the schematic, a transient analysis would need to be run with the Operating Point Only check box enabled before returning to the schematic to display the voltages. In Micro-Cap 6, simply enable the Dynamic DC analysis, the operating point will be calculated, and the resulting values will be displayed on the schematic. It is an interactive process in which the user can modify the circuit, the program will calculate the operating point, and then display one or more measures of the DC state on the schematic.

There are now multiple display options available for the schematic. Along with the node voltages and digital states, currents, device power, and device conditions may now also be viewed. Figure 5 displays a circuit where the voltages (blue), currents (red), and the device conditions (black) are shown. All of these displays will be updated whenever an edit is made to the schematic. All of these displays are also available when returning from transient, AC, or DC analysis in order to view the last calculated data point. These display values may be moved to help clean up the schematic or to place a display in a more visible area.

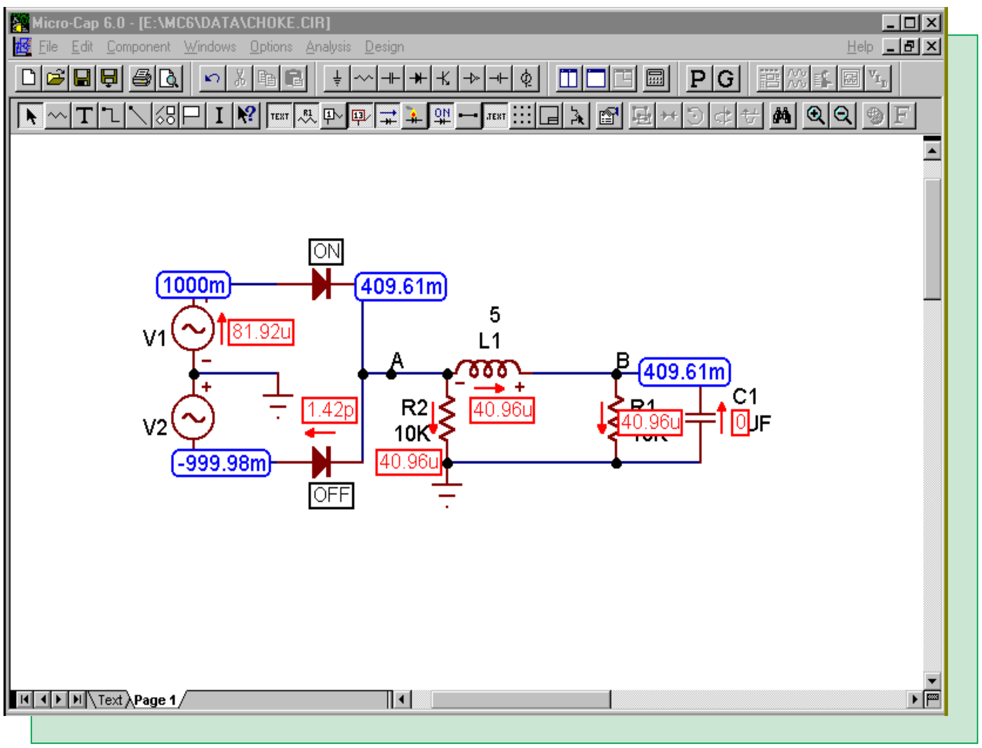



*Fig. 5 - Dynamic DC Analysis*

The circuit parameters may be edited through the Attribute dialog box in the standard manner. However, the battery, I source, V source, and resistor have an optional slider feature also that can be used to manipulate their values. This slider will appear next to the component and can be dragged on to adjust the value of the component. The up and down arrow cursor keys can also be used to change one of these components.

**Enhanced DC Analysis**

DC analysis has been enhanced to add even greater capabilities for simulation. There are now more possible input variables that can be swept in a larger variety of ways. The variable 1 input can be swept through an Auto, Linear, Log, or List method. The variable 2 input can be swept through a Linear, Log, or List method. This not only provides greater sweeping capabilities, but it also aids convergence. If one method produces a convergence error, another method may be used to get beyond that error. A new convergence routine was also added to DC analysis so that if a single data point fails to converge, that data point will be skipped and the next data point will be calculated. This lets Micro-Cap display the maximum amount of data that it can produce for the simulation instead of just the data prior to the convergence failure.

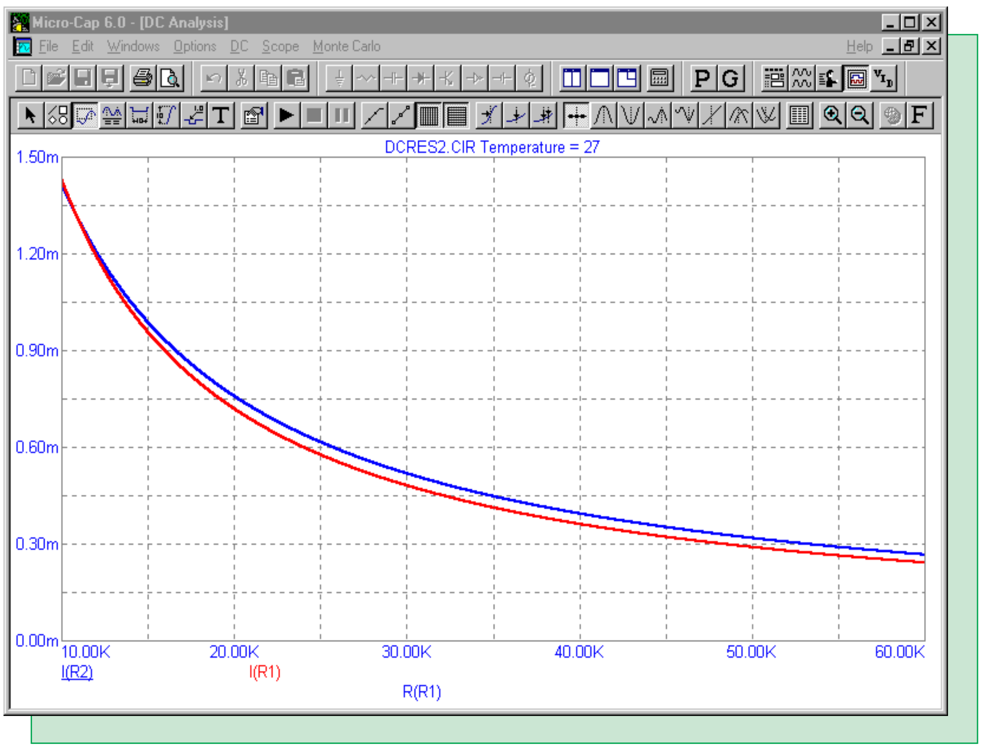



*Fig. 6 - Enhanced DC Analysis*

In previous versions of Micro-Cap, DC analysis only allowed a voltage or a current source to be swept. In Micro-Cap 6, the input variables that may be swept consist of the temperature, voltage and current sources, model parameters, and symbolic parameters. For example, a resistor can be easily swept now by sweeping the R parameter in its model statement. Figure 6 displays the results of a DC analysis when sweeping a resistance.

Transient, AC, and DC analyses have all added in the capability to step the temperature of an analysis by defining a list of values rather than just a linear progression.

## New Macro Capabilities

Micro-Cap has added a few new features to its macro capability. Macros may now be automatically created through the Make Macro command. If you want to convert a portion of your schematic into a macro, simply draw a select box around the circuit area and choose the Make Macro command. This command will cut the circuit at all spots where the wire hits the select box and produce pins at each spot. A single component will be created where the select box is. Everything inside the box will be copied to a new circuit and an entry will be made in the Component editor to define the new macro which lets the macro be used in other circuits. The shape of the macro component may have its size and location of the pins adjusted both on the schematic and in the Component editor. This turns the creation of the macro into an easy one-step process. Figure 7 displays two exact circuits of MOSFET inverters in series. On the bottom circuit, the middle inverter has been converted into a macro through the Make Macro command.

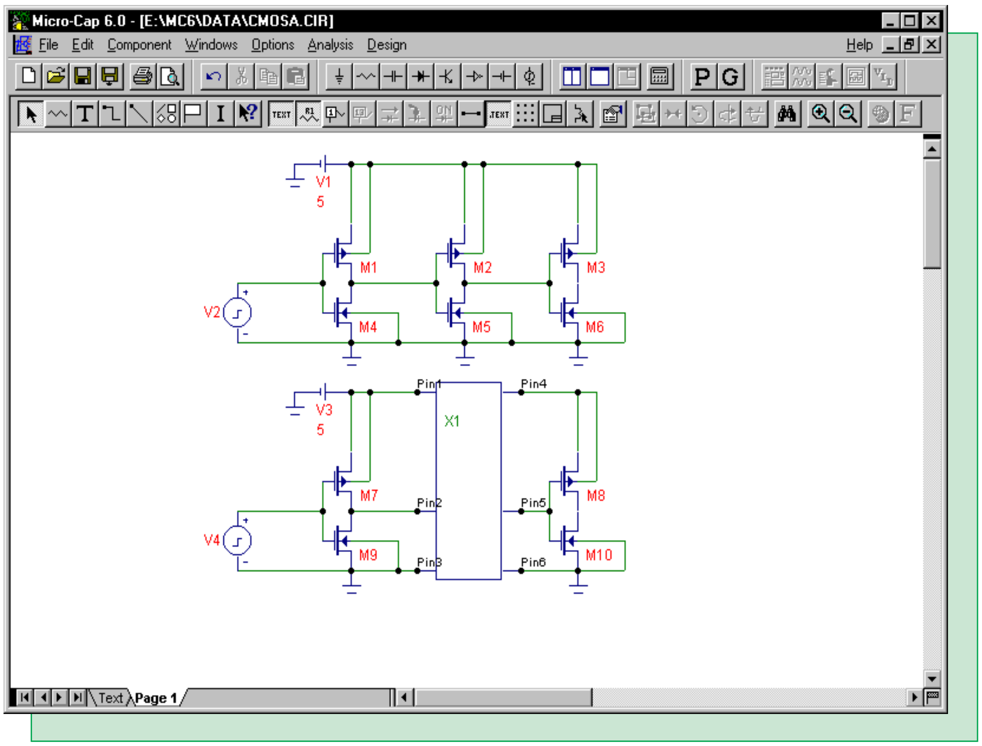



*Fig. 7 - Make Macro Command*

Macros that have parameters passed to them may now have default values for the parameters defined in the macro circuit. This eliminates the need to always define all of the parameters every time a macro is placed in a schematic.

The extension .MAC is now available for a macro circuit in order to differentiate it from common schematics. Both .CIR and .MAC may still be used for a macro circuit, but .MAC provides an easier recognition of macros for the user.

**Multiple Component and Shape Files**
Both the Component editor and the Shape editor can now manage multiple files. This gives the user the capability to put their own components and shapes in files separate from the standard ones that come with Micro-Cap. This makes transferring user created components to other people a simple task since only the needed component information can now be passed along.

**Updated BSIM3 V3 Model**
The latest BSIM3 V3.2 model has been added to the available MOSFET models in Micro-Cap 6. This model is an improved version of the BSIM3 V3.0 model that exists in Micro-Cap 5.

**Larger Component Library**
Over 12000 components are now available in the Micro-Cap library for simulation. New vendor supplied subcircuits have been added from Harris, Motorola, Analog Devices, Siemens, and others. Microsemi diodes are now available in the Micro-Cap binary libraries.

**Improved User Sources and User Files**
The format for the user files has been changed to be more compatible with standard output from programs such as Excel. Waveforms may now be saved to any user file after the simulation is finished. The waveforms in the user files can be used by User sources in transient, AC, or DC analysis. The waveforms may also be invoked directly in a plot by plotting them through the new WaveformX and WaveformY operators.

**Normalize at Cursor Option**
The Normalize at Cursor option is available through the scope features of Micro-Cap. This option will normalize the selected curve at the point that the cursor is situated at.

**GMIN Stepping**
A new DC operating point convergence routine has been added that will step the GMIN global setting in order to produce a solution. GMIN Stepping will be attempted after both normal methods and Source Stepping have failed to achieve DC convergence.

**Monte Carlo Improvements**
New tolerance options have been made available for Micro-Cap 6. The capability to tolerance .define parameters now exists and new LOT and DEV extensions allow unique distributions for individual model parameters rather than a single distribution for all parameters.

**New Expressions**
- IHD function - New IHD() operator computes individual harmonic distortion.
- DSP expressions - DSP math functions such as dB(harm(v(1))) are now possible.
- Bessel Functions - Complex Bessel functions of the first and second kind.
- Complex Series Functions - Complex Series functions. Series(m,1,10,(1+fact(m+2)*(1+J)))

**Other New Features:**
**Analysis Features**
- Power Variables - Power variables for plotting generated, stored, and dissipated power in each device and for the total circuit.
- Symbolic Parameters - Stepped symbolic parameters may be used in time range and time step fields.
- Numeric Trackers - Numeric trackers report analysis plot values at cursor, intercept, and mouse positions.

• Data Point Reduction - Reduce data points command now works in both immediate and/or after-run mode.
• X-Y Cursor Bars - Optional Y or X and Y numeric cursor bars.
• Lockable Data Tags - Data tags can be made immobile.
• Analysis Plot Scales - Right-justified scale numeric formats.

**Schematic Editor**
• Global Nodes - Grid text of the form $G_name connects nodes between main circuit and subcircuits and macros.
• Select Mode Toggle - Spacebar toggles between Select mode and the last used mode.
• Grid Coordinates - Grid coordinates displayed in Shape and Schematic editors and on the waveform plot.
• Recent Files List Size - Set the length of the recently used files list.
• Whole Word Search - Whole word search option in text area.
• Rubberbanding - One direction or any direction rubberbanding option.

**Parameter Stepping**
• Analysis Stepping - Individual stepping control for AC, DC, and transient analysis.

**Probe**
• Macro/subckt Plot Selector - Digital component and macro / subcircuit waveform selector list.
• Analysis/properties Control - Analysis Limits and Properties dialog boxes are available within Probe for changing run parameters and controlling the waveform displays.
• Plot/Circuit Top Selector - New controls allow selection of floating display of plot or circuit on top selection.

**BJT model**
• Substrate diode - Substrate junction capacitor and diode connect between substrate and collector for NPN and PNP, or substrate and base for LPNP and PNP4.
• Lateral PNP model - BJT model statements now handle the alternative LPNP form .model L1 LPNP (model parameters).
• NK parameter - High-current roll-off parameter, default = .5
• NS parameter - Substrate p-n emission coefficient, default = 1.
• ISS parameter - Substrate p-n saturation current, default = 0.

**Resistor model**
• New noise parameter - Noise multiplier parameter can be used to eliminate or reduce noise from individual resistors.

**Function source model**
• New noise expressions - Nonlinear function current sources noise expressions provide user-defined noise formulas.

**Sample and Hold model**
• Sample Control - New sample expression controls sampling.

**Component Library/Editor**
• Hidden Pins - Component pins may now be hidden from view on the schematic and automatically connected to a node specified by name.

**Miscellaneous**

• New Peak Function - Peak performance function that returns the x value at a y peak.
• New Slope Function - New slope performance function.
• Branch Selector - Selector in Go To Performance dialog box allows waveform branch selection.
• Specification Polygon - Polygon object can be used in analysis plots to show design specifications.
• Selected Branch Color - Optional selected waveform branch color
• Cut and Paste - Copy and paste calculator results to other fields.
• Analysis Type Variable - Analysis type variable for use in range fields.
• Message Delay - User-set time delay for warning messages.
• User Text in Plots - User defined title headers for plots.
• Time and Date Stamps - Simulation Time and Date in output file.
• User Definitions File Variables - Model parameters and symbolic parameters from the User Definitions file are now steppable.
• Colored waveform Branches - Optional different colors for stepped waveform branches.
• Rubberband Hotkey - Hotkey and icon for toggling rubberband mode.
• Improved Find Control Keys - New command hotkeys. CTRL + F to invoke and F3 to repeat.
• Attribute Dialog Box - Model names list tracks typed model name.
• Paste Offset - Offset to repeated paste operations avoids overlap.
• Scale Display - Use of #### numeric format in small windows.

## Table Defined Resistance

The VALUE attribute of the resistor has the capability to be defined with either a constant or an expression. However, not all applications for a resistor can conform to a standard expression. In some cases, a resistor may need to be defined with a table of values. For example, the load of a circuit may need to be varied to simulate the connection of different devices over time. The Table operator in Micro-Cap is available for this situation. The syntax of the Table operator is as follows:

TABLE(x,x1,y1,x2,y2,...xn,yn)

where x is the variable used as the input. This operator returns a value for y associated with the value of x, interpolated from the table. X values less than x1 generate an answer of y1. X values greater than xn generate an answer of yn. The data points must be entered in input ascending order where x1 < x2 < ...xn.

The circuit in Figure 8 displays an example of a resistor using the Table operator. This resistor is used as the load for a lossless transmission line that is defined with a characteristic impedance of 50 ohms and a delay of 100ns. The resistor has its VALUE attribute defined as:

Table(t,0,50,4.9u,50,5.1u,100,9.9u,100,10.1u,150)

Time has been declared as the input variable. The output of the table is defined where the load resistance will be 50 ohms between 0s and 4.9us, 100 ohms between 5.1us and 9.9us, and 150 ohms from 10.1us on. The time frames of 4.9us to 5.1us and 9.9us to 10.1us are the periods

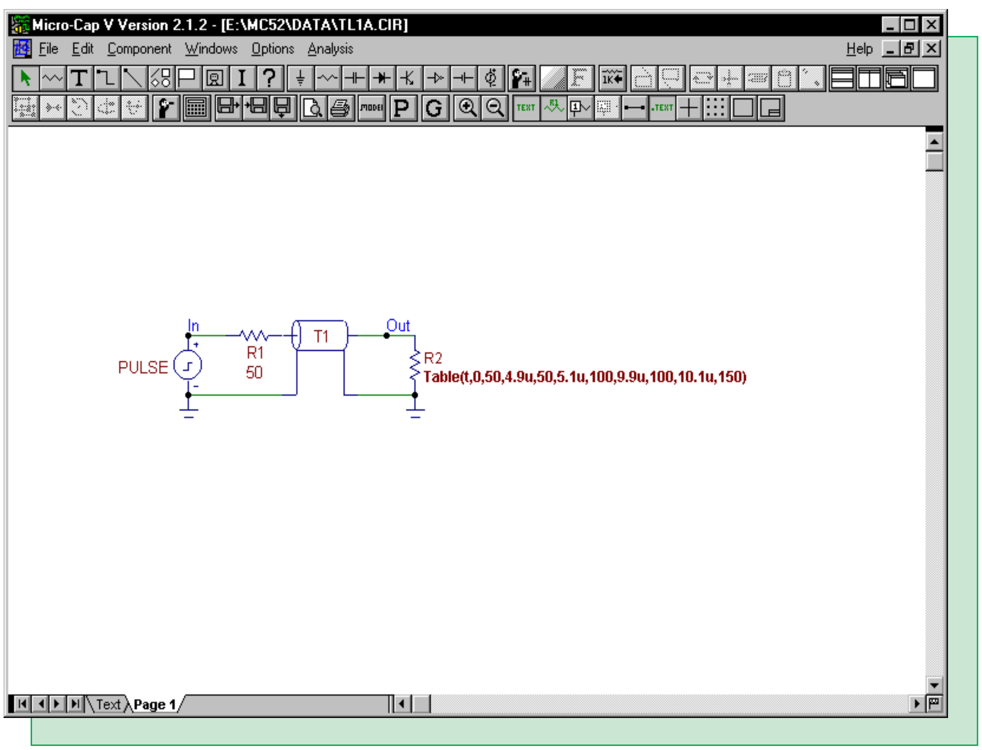



*Fig. 8 - Table Defined Resistance Circuit*

where the resistor linearly transitions to the new value.  The input to the transmission line is a pulse source with a 50 ohm series resistance.  The pulse source uses the following model statement:

.Model Pulse Pul (Vzero=0.5 Vone=4 P1=5n P2=20n P3=500n P4=600n P5=1000n)

which provides a pulse from .5V to 4V of width 480ns every 1us.

Figure 9 displays the transient analysis results of this circuit for a 15us simulation.  The top plot is the voltage at both the In and Out nodes.  The magnitude of the voltage at node Out is dependent on the load resistance.  At 50 ohms, the magnitude is 2V.  At 100 ohms, the magnitude is 2.67V.  At 150 ohms, the magnitude is 3V.  This matches the results obtained by a voltage divider equivalent of this circuit.  The middle plot is a plot of the resistance of the resistor R2.  As can be seen in the figure, the plot of the resistance matches exactly with the specified data points within the Table operator.  The bottom plot of the figure displays the power dissipated in the R2 resistor.  The power was plotted with the expression I(R2)*V(R2).

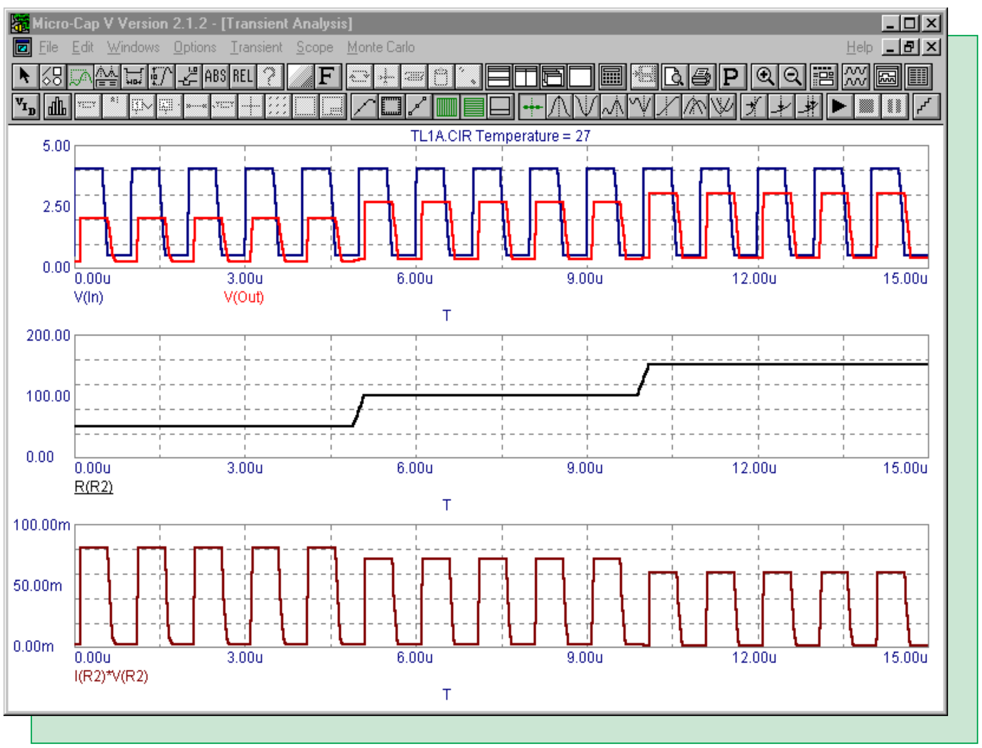



*Fig. 9  - Table Defined Resistance Analysis Results*

While this example used a simple time based input, the Table operator can also use dynamic variables such as voltage or current as its input.  This allows great flexibility in using this method to create models of switches, thermistors, voltage suppressors, and other resistor varying models.  The actual data values for x1,y1 through xn,yn must be constant values when entering an analysis so dynamic variables are not available when defining these.

## Digital vs Analog Pullup Resistors

Pullup resistors can be modelled with two components in Micro-Cap. The first component is the basic resistor. This is the analog pullup resistor. The second component is called Pullup in the Digital Primitives. This is the digital equivalent of the pullup resistor. These two parts are not necessarily interchangeable. There is one basic guideline that should be followed when determining which component to use:

Use the digital pullup resistor when a digital output is connected directly to a digital input, and use the analog pullup resistor in any other configuration.

A violation of this guideline appears in Figure 10. In this case, a 5K ohm resistor, connected to 5V, has been placed at the junction between the output of a 7405 inverter, X1, and the input of a second 7405 inverter, X2. These inverters have open collector outputs, and therefore need a pullup resistor to operate correctly. The problem that occurs is that a single DtoA subcircuit is automatically placed to convert the digital output of X1 for use with the resistor. However, the input of X2 uses the digital state of X1 directly. The digital state of the node has not taken into account the pull up effect of the resistor and is still at high impedance since there is no AtoD component before the X2 input to recalculate it. The high impedance input fails to drive the second inverter which subsequently produces an erroneous output. Figure 11 displays the transient results of this circuit. While the V(PU1) waveform is correct, the digital state failed to propagate through the second inverter, and the V(PU2) waveform produces a constant one state. D(X1.1$DTOA) is the digital state at the output of X1.

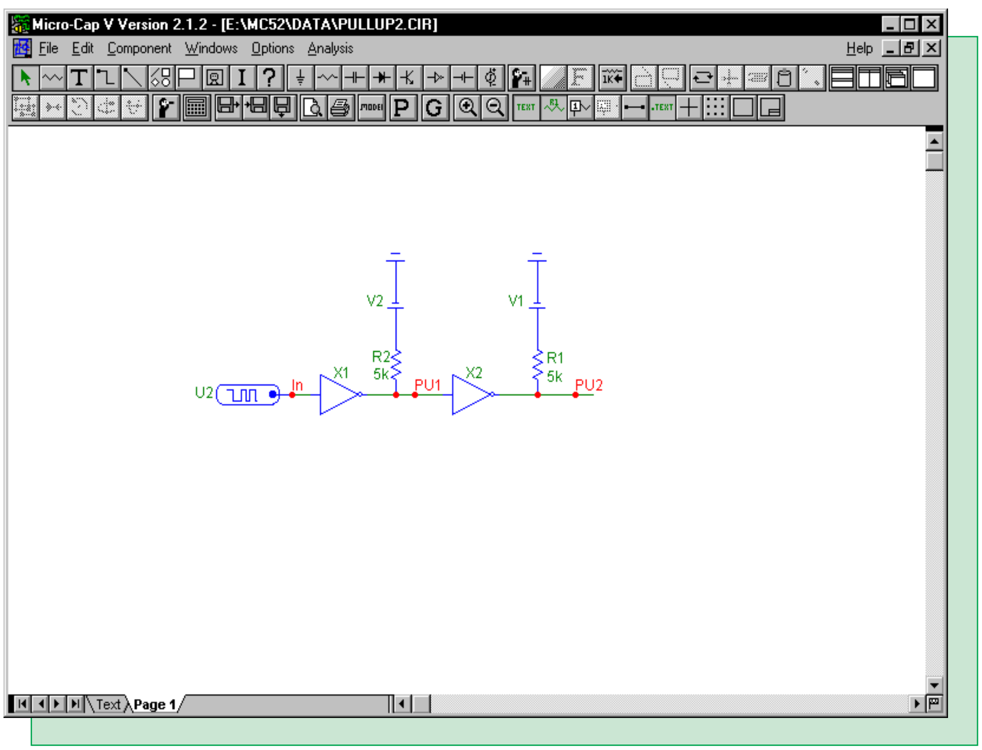



*Fig. 10 - Analog Pullup Resistor Circuit*

There are three methods to fix this. The first is to place an AtoD model before the input of X2. Another is placing a small resistor in series with the input of X2 which would place an AtoD model in automatically. The third is to substitute a digital pullup in place of the resistor and battery. The circuit in Figure 12 uses a digital pullup between X1 and X2. The pullup's strength is used when determining

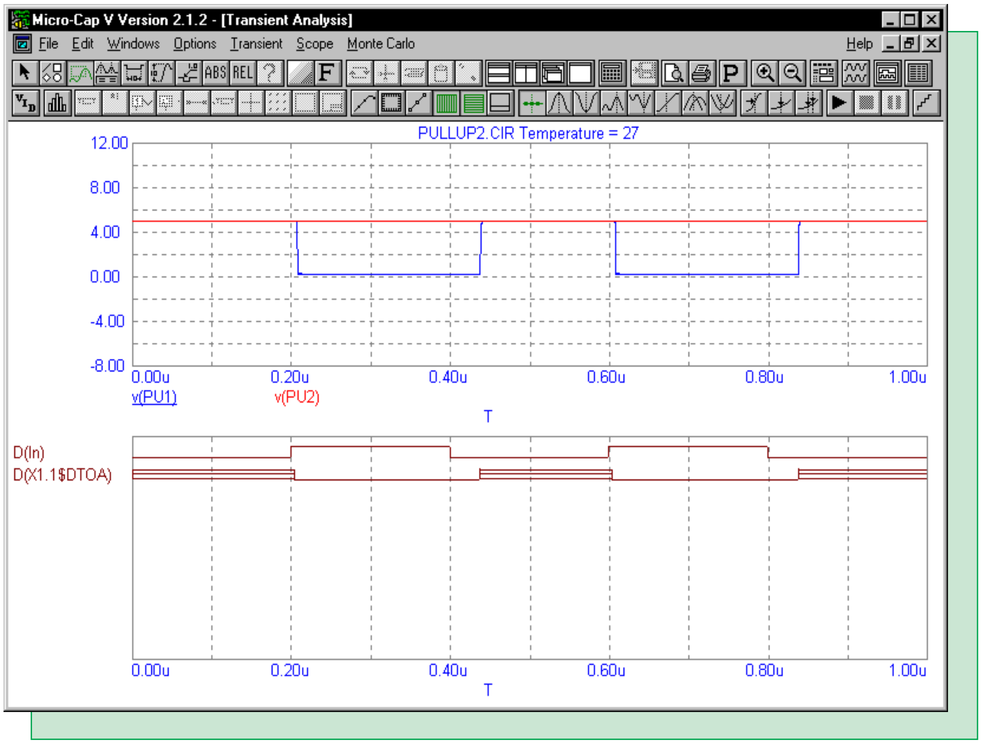



*Fig. 11 - Analog Pullup Resistor Analysis*

the digital state of the node and will produce the correct state. The pullup has been defined with the I/O model of IO_PULLUP. In the text area of the circuit is the following model statement:

```
.MODEL IO_PULLUP UIO (
+  DRVH=5K DRVL=1MEG
+   ATOD1="ATOD_STD"  ATOD2="ATOD_STD"
+   ATOD3="ATOD_STD"  ATOD4="ATOD_STD"
+  DTOA1="PULLUP_DTOA" DTOA2="PULLUP_DTOA"
+  DTOA3="PULLUP_DTOA" DTOA4="PULLUP_DTOA"
+  DIGPOWER="DIGIFPWR" )
```

The DRVH and DRVL parameters determine the strength of the component. The DRVH defines the strength at the one state, and the DRVL defines the strength at the zero state. This model statement places the equivalent of a 5K ohm resistor at the node when the digital state is a one. The ATOD_STD subcircuit was chosen for the AtoD parameters because the 7405 uses the IO_STD_OC interface model which also uses that subcircuit. The DIGIFPWR for the DIGPOWER was also chosen for the same reason. The PULLUP_DTOA subcircuit for the DtoA parameters should be used every time a pullup resistor is in the schematic. These subcircuits all reside in the DIGIO.LIB file.

Figure 13 displays the results for the digital pullup circuit which produces the expected results. PU1 must now be plotted with the D operator since it is a purely digital node. Note that the resistor at the output of X2 doesn't need to be changed because there is no digital input directly connected to the node.

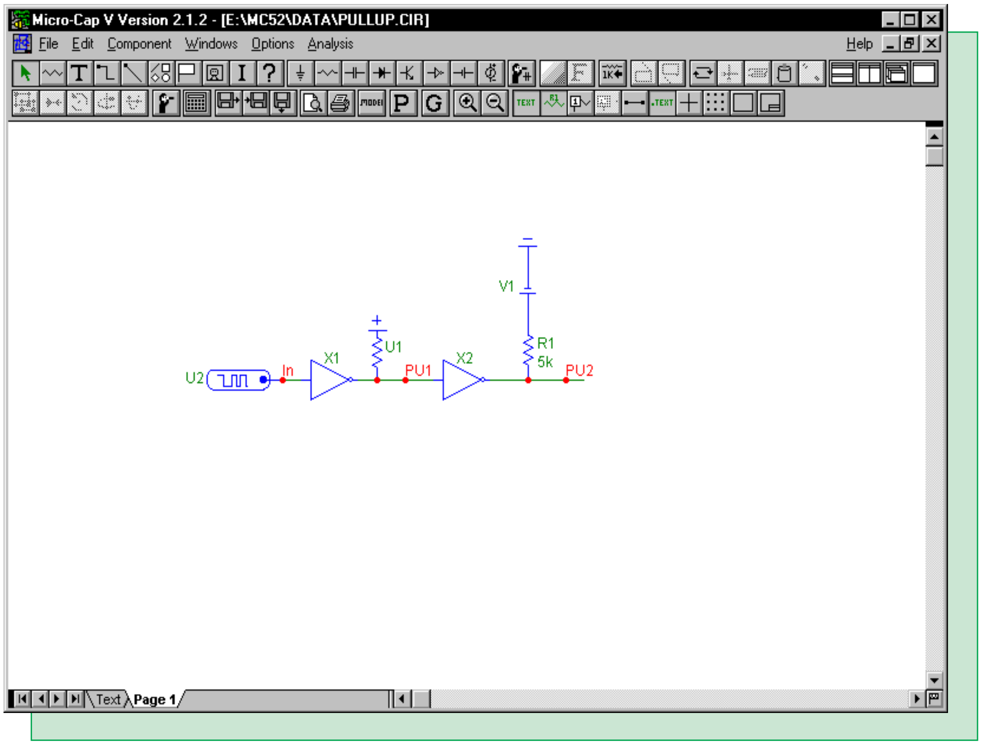



*Fig. 12 - Digital Pullup Resistor Circuit*

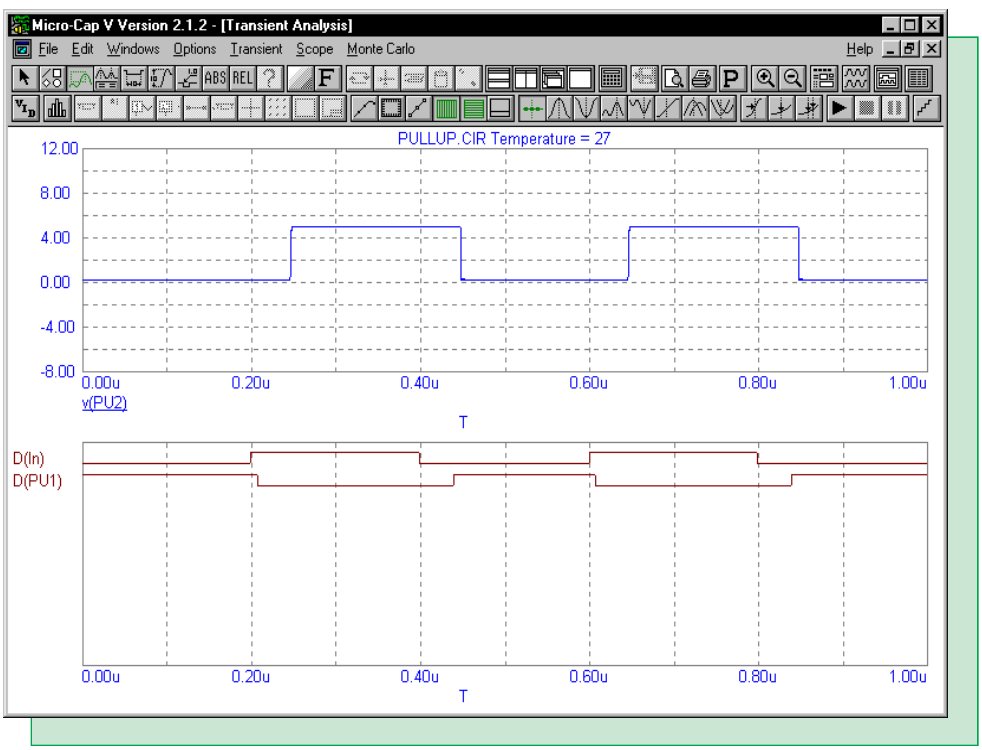



*Fig. 13 - Digital Pullup Resistor Analysis*

## Perfect Transformer vs Ideal Transformer

Questions have arisen about the operation of the transformer component. In some cases, while the voltage gain produced the correct results, the current values of the transformer for the simulation were completely unexpected. The reason for this is that the transformer component, along with its equivalent of two inductors and a K device, operates as a perfect transformer rather than an ideal transformer. The familiar transformer equations for an ideal transformer are (all equations in this article are assumed to have a coupling of 1):

$V_1 = V_2 * n$
$I_1 = I_2 / n$
$Z_1 = Z_2 * n^2$

The equations for the perfect transformer are:

$V_1 = V_2 * n$
$I_1 = I_2 * (1 + Z_2/(s*L_2)) / n$
$Z_1 = Z_2 * n^2 / (1 + Z_2/(s*L_2))$

As can be seen in the above equations, the value of the secondary inductance can have a significant impact on both the current and impedance transformation ratios while the voltage gain is independent of the secondary inductance.

A circuit that demonstrates the effect that the secondary inductance can have appears in Figure 14. This circuit consists of merely a sine source, two one ohm resistors, and a transformer. The sine source has been defined as a 1kHz, 5V sine wave with the following model statement:

.MODEL INPUT SIN (F=1k A=5)

The transformer has its VALUE attribute defined as:

16u,1u,1

so that it is a 4:1 step down transformer whose primary inductance is 16uH, secondary inductance is 1uH, and coefficient of coupling is 1.

The transient analysis results for this circuit appear in Figure 15. The circuit has been simulated for 10ms with a maximum time step of 10us. The voltages at the primary and secondary windings appear in the top plot. The voltage gain from the primary to the secondary is the expected .25 but the actual voltage values are lower than an ideal transformer would produce. This is because the impedance reflected back to the primary is lower due to the secondary inductance effect, and this causes a larger voltage drop across the RPri resistor. The primary and secondary currents appear in the bottom plot. The current gain is nowhere near the gain of 4 that would be produced from an ideal transformer because the 1uH secondary inductance is small enough to have a major effect. The current also has a phase shift factored into it due to the reactance of the inductance.

In order to produce the ideal transformer results, the primary and secondary inductances would need to be set high enough where they would have a negligible effect on the current and impedance gains. In this case, setting the transformer inductances to 16H and 1H would have produced values equivalent to an ideal transformer. Dependent sources may also be used to model an ideal transformer.

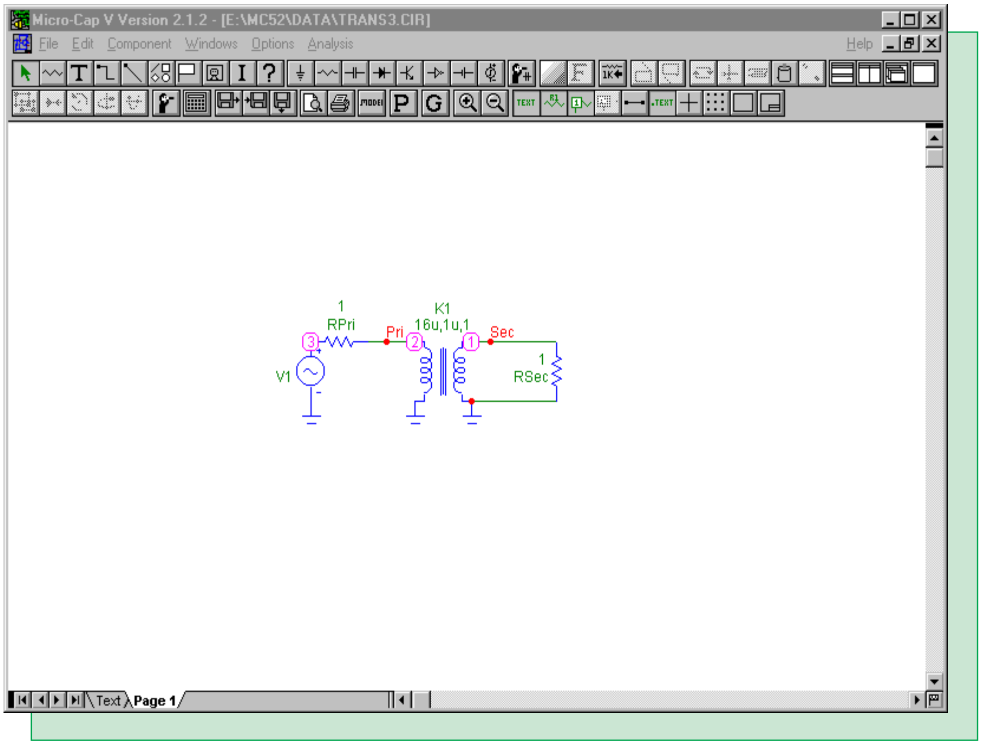



*Fig. 14 - Perfect Transformer Circuit*

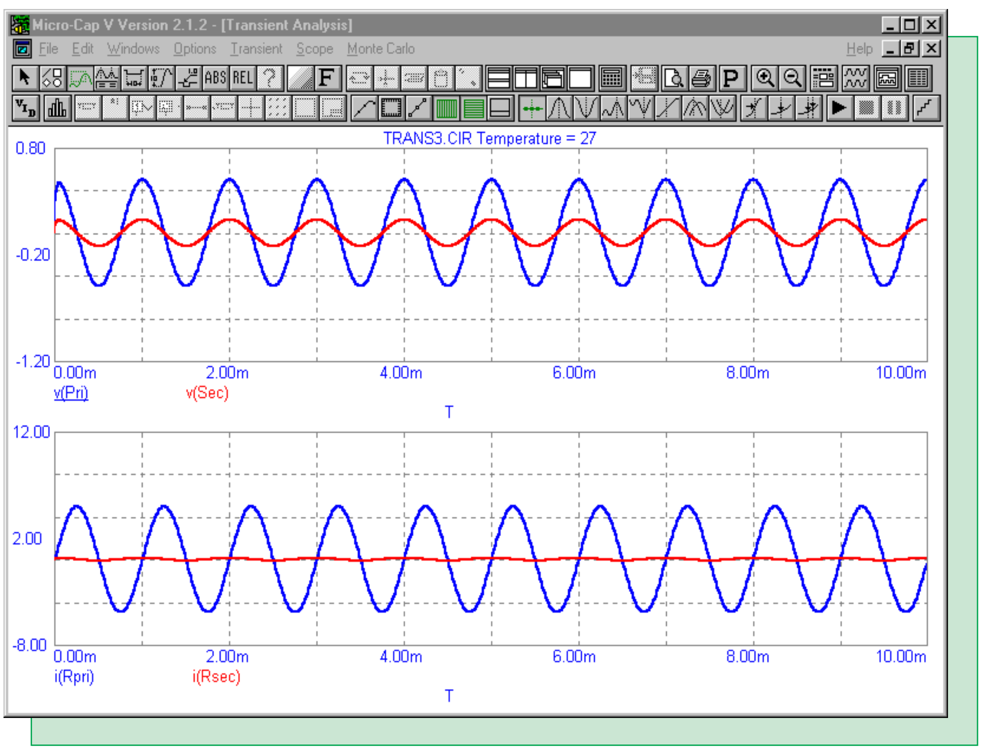



*Fig. 15 - Perfect Transformer Analysis*

## Product Sheet

### Latest Version numbers

Micro-Cap 6
Micro-Cap V ...................................................................... Version 2.1.2
Micro-Cap IV IBM/NEC/MAC .................................... Version 3.04

### 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 or standalone - 1 license) .......... $3595.00
- Upgrade from MC5 Ver 2 to MC6 .............................. $500.00
- Upgrade from MC5 Ver 1 to MC6 .............................. $750.00
- Upgrade from MC4 to MC6 ......................................... $1100.00

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.