3.4 General Operation

The general operating procedure for designing and simulating circuits using TopSPICE is as follows:

- Create or edit the circuit schematic drawing using the TopSPICE Schematic Editor. Or, create or edit the circuit SPICE netlist (text) file using the TopSPICE Circuit File Editor.
- Include links to any SPICE model files or library files necessary to simulate the circuit.
- Setup the desired simulation analyses. Setup the desired simulation output and "auto-plotting" options.
- Simulate the circuit. When the simulation completes successfully, the simulator window is closed and the TopVIEW post-processor is started automatically. If there were simulation errors, you can choose to browse the output file for error messages using the TopSPICE File Browser.
- The TopVIEW post-processor automatically plots any "auto-plot" commands, if any, or displays the "probe" selection menu. At this point, the user can plot any desired simulation data and perform further analysis of the simulation results.
- After you are finished with all the desired post-processing operations, you must exit and close TopVIEW. This will return you to either the Schematic Editor or the Circuit File Editor window. If you run another simulation without closing the TopVIEW window, another TopVIEW will be opened after the new simulation is finished. The data and plots for a TopVIEW window, which is already open, is not updated after each simulation.

Most circuit designs involve an iterative process of modifying the circuit and repeating the simulation until the desired performance is achieved. To speed up this process, the TopSPICE "zip mode" allows quick cycling between the editing, simulation and plotting functions. From the schematic or text editor, pressing Alt-Z saves the circuit, starts the simulation and plots the results without going through any menus. If you are in TopVIEW, Alt-Z saves the current plot options, exits TopVIEW and returns to the editor screen.

3.5 Creating a Design Project

Most TopSPICE circuit design projects that you create involve these steps:

1. Enter the circuit design. You can enter your circuit as a schematic drawing, a SPICE circuit (netlist text) file, or a combination of both.
2. Label all the circuit nodes on the schematic that you are interested in observing or need to reference later on. Although you could use the node numbers assigned by the schematic editor, these might change as you edit the schematic.
3. If any device models are used in the circuit, you need to provide the model definitions for them, and/or specify the model files or model library files where they can be found. This is not necessary if a model is in one of the default model libraries (these are automatically searched).
4. Specify the simulation analysis commands. Before you can simulate your circuit, you need to specify the type of analyses you want (DC, AC, transient, etc.), operating conditions such as temperature, output data options, and/or other simulator run-time options. This is analogous to setting up the measurements you want to perform when working with circuits built on breadboards. TopSPICE offers several options for specifying analysis commands: using the **Analysis|Setup** menu (the easiest way), add the commands directly on the drawing as schematic objects (useful for documentation purposes), enter them manually under the **Analysis|Misc. commands file** option, or use a combination of any of the previous methods.
5. Simulate the circuit. Compared to working with breadboards, running the simulation corresponds to turning the power on and recording the responses.
6. View the simulation results. After the simulation is successfully completed, TopVIEW is automatically invoked and the results plotted. TopVIEW functions as an oscilloscope that lets you view waveforms, perform measurements and other analyses of the data.

If you are using TopSPICE for the first time, it is recommended that you run some of the sample circuits provided before starting your own circuit (see the section "Running the Sample Schematic").

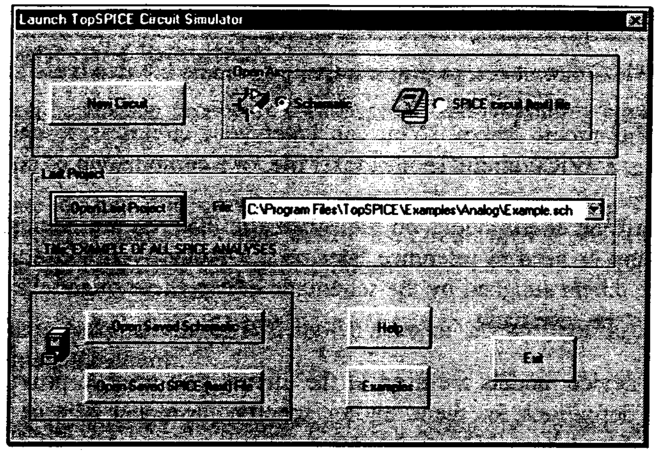

Figure 2 Launch TopSPICE/Win32 menu

3.6 Launch TopSPICE

The Launch TopSPICE screen simplifies the task of starting a new or previously saved circuit design project, switching between schematic or circuit file editors, and accessing the provided example circuit files.

New Circuit

Use the **New Circuit** button to start a new circuit project. Depending on the **Open As** preference setting it will start the schematic or text editor with an empty working area. By default your circuit project files are saved in the TopSPICE folder "\Circuits".

Open Last Project

Use the **Open Last Project** button to resume working on the last circuit project. The "Last Project" box shows the last project file name and title. You can also select to open any of the most recent projects listed by clicking on the desired file name first.

Open As Options

TopSPICE offers the convenience of working with a schematic drawing or a SPICE netlist (text file) directly depending on your needs or preference. The **Open As Schematic** option launches the Schematic Editor. The **Open As SPICE circuit (text) file** option launches the Circuit File Editor. You must select the **Open As** option before clicking on the **New Circuit** or **Open Last Project** buttons.

You can switch between the schematic and SPICE circuit file modes at any point during the project. If you start the project in schematic mode and then switch to the circuit file mode, the Circuit File Editor loads the last circuit file created by the Schematic Editor from the circuit drawing.

If you start the project in circuit file mode and later switch to the schematic mode, the Schematic Editor starts with an empty sheet (TopSPICE cannot create a schematic drawing from the circuit file). However, you can insert a previously created netlist to the schematic using the .INC command.

Whenever the Schematic Editor generates a circuit file, it overwrites the existing circuit file. Hence, any manual changes to the circuit file you make will be lost. If you need to make manual changes or entries to the circuit file, you should add them into the "miscellaneous commands" (.MIS) file.

Open Saved Schematic

Use this button to browse and open any previously saved circuit schematic file – the default file name extension is ".SCH". By default, it opens the last folder you opened, if any, or the "\Circuits" folder.

Open Saved Circuit File

Use this button to browse and open any previously saved SPICE circuit (text) file – the default file name extension is ".CIR". By default, it opens the last folder you opened, if any, or the "\Circuits" folder.

Examples

Use this button to open one of the example circuit files (located in the \Examples subfolder). To display the example schematic files, select "Schematic files (*.sch)" from the "File types" box (default). To display the example SPICE circuit (text netlist) files, select "SPICE circuit text files (*.cir)" from the "File types" box. The complete list of example circuits and descriptions is available in the test file EXAMPLES.TXT (open the TopSPICE program folder icon "List of Example Circuits").

3.7 Running an Example Schematic

1. Select the sample file by clicking on the **Examples** button on the "Launch TopSPICE" screen and clicking on the file name (such as `Example.sch`). When the schematic file is opened, the Schematic Editor is launched and the example schematic loaded.

2. To view and navigate the schematic use the zoom commands, scroll bars, arrow keys, shortcut keys, or place the mouse at the edge of the drawing window to automatically pan the schematic (if auto-panning is on). You can also select **View|Full sheet** command to fit the entire schematic to the drawing window.

3. To view the schematic title block, which includes the circuit title and other descriptive information, you can either zoom in on the lower right corner or use the **Edit|Title** block command.

4. To view the analysis commands specified, select **Analysis|Setup** menu. Additional commands are defined under the **Misc. commands file.**

5. Simulate the circuit by selecting the **Analysis|Run Simulation** menu command. After the simulation is completed, the results are automatically plotted by the TopVIEW post-processor program.

6. To return to the schematic, select **File|Exit** from the TopVIEW menu.

3.8 Running an Example Circuit (SPICE) File

1. Click on the **Examples** button on the "Launch TopSPICE" screen. From the file types, select "SPICE circuit files (*.cir)". Select the sample circuit by clicking on the file name (such as DCmotor.`cir`). When the circuit file is opened, the Circuit File Editor is launched and the sample circuit loaded.

2. To view the analysis commands specified, select **Analysis|Setup** menu. Additional commands are defined under the **Misc. commands file.**

3. Simulate the circuit by selecting the **Analysis|Run Simulation** menu command. After the simulation is completed, the results are automatically plotted by the TopVIEW post-processor program.

4. To return to the circuit file, select **File|Exit** from the TopVIEW menu.

3.9 Simulating Your Circuit

- From the "Launch TopSPICE" screen, select **Open As Schematic** if not already selected.

- Click on the **New Circuit** button. Enter your project file name.

- Select **Edit|Title** block to add a title for your circuit and other descriptive information.

- Place your circuit parts using the **Draw** menu commands. If you are not familiar with schematic capture programs, it is highly recommended that you hand draw your circuit on paper first. This will minimize the need to edit and move components around the schematic, which is the most time consuming part of drawing a schematic on screen.

- Wire all the parts on the schematic using the **Draw|Wire** command. All part pins must be either connected to a wire or another part pin (the ground, power rails and I/O pin symbols are also considered parts), or given a node label. You can wire the parts as you place them, or whenever is most convenient or efficient.

- To indicate an electrical connection between two crossing wires, or between a wire or a part pin contacting another wire at right angles, place a junction using the **Draw|Junction** command.

- All circuit nodes are automatically assigned node numbers. However, these numbers are not fixed. The node number may change as you edit the schematic. If you need to reference a circuit node elsewhere on the schematic or in a simulation command or plot, you should label the node using the **Draw|Label Node** command.

- You can use the **Draw|Text** command to add any text on the drawing for documentation or descriptive purposes.

- After the circuit drawing is complete, select **Analysis|Setup** to specify the desired analyses to be simulated and the results to be plotted.

- Select **Analysis|Run Simulation** to simulate your circuit and plot the results.

- If the simulation does not proceed because of an error, you can browse the output file by selecting **Analysis|Browse Output File** for error messages. Most error messages are self-explanatory. After making the necessary changes run the simulation again.

3.10 TopSPICE Files

After a TopSPICE session is completed several files may be generated on the project working directory for the following purposes:

*.SCH	schematic drawing files.
*.Snn	schematic sheet files (multiple sheet schematics).
*.SBK	schematic back-up files.
*.Bnn	schematic sheet back-up files.
*.SIM	simulation setup files.
*.MIS	miscellaneous commands files.
*.CMD	analysis commands files.
*.MAP	schematic node cross reference map files.
*.CIR	SPICE circuit files.
*.OUT	simulation output listing files.
*.OPB	simulation bias data binary output files.
*.SAV	simulation binary output data files.
*.Fnn	plot format files.
*.Wnn	plot history data files.
*.Onn	plot history output files.
*.LIB	device model library files.

TopSPICE also creates several temporary files while it is running. These are normally deleted when you exit the program. However, if there is an abnormal termination or system crash, your project directory may contain files with names such *.TMP, *.$$$, FOR* and *.PRE. These files are not needed and you should delete them.

Simulator Input File

TopSPICE simulator input files are text files using the standard U.C. Berkeley SPICE2 format. The TopSPICE netlist syntax is a superset of the standard SPICE2G6 syntax with extensions to handle digital elements, behavioral modeling and other TopSPICE enhanced features. TopSPICE also accepts many SPICE3, PSpice™ and HSPICE™ netlist syntax extensions.

Simulator Output Files

Most TopSPICE simulations create two output files: an output listing file (*.OUT) and a binary data file (*.SAV). The output listing file is in SPICE2G6 output format. It can be examined using the **Analysis/Browse Output File** command or any text editor program. The binary data file is created when the .PROBE (or .SAVE) command is specified. This file contains simulation results data in a compact proprietary binary format.

4 Tutorial

4.1 SPICE Circuit Simulation Basics

For new users of SPICE, this section will explain how to describe a circuit, select the appropriate type of analysis, how to interpret the output file data, and other general principles of circuit simulation. A "hands-on" approach is best for learning how to work with circuit simulators. Running some sample simulations using small circuits for which you can estimate the output results is the best approach.

Before going into the details of entering a circuit and running a simulation, some of the terminology used throughout this tutorial are defined here:

Circuit:	any electrically connected grouping of two or more components.
Circuit description:	schematic drawing or SPICE netlist text describing the components and their interconnections in a circuit.
Netlist:	a collection of SPICE statements that describe the components in a circuit and their interconnections.
Circuit file:	file containing one or more SPICE netlists along with the necessary SPICE control statements. This is the input file to the simulator program.
Stimulus:	an input source affecting the state of the circuit to evaluate its response.
Convergence:	SPICE uses numerical methods to solve the system of circuit equations. It converges to the correct solution using an iterative technique. In some cases, SPICE

does not converge (i.e.: can not find an acceptable solution).

Pre-processing: any processing of the input file prior to the start of the actual simulation. For example, model library searches.

Post-processor: a program that further processes and/or displays the output data file generated by SPICE.

When using TopSPICE or any other circuit simulator, it is a good practice to keep these points in mind:

- SPICE does not analyze circuits for topology errors (except those that violate SPICE rules). The results you obtain are "correct" only if your circuit input file is correct. If you get unexpected results check your circuit description carefully.

- Device models are accurate only over a limited range of the devices operating range. If a device in your circuit is operating outside the "typical" operating range, you should check that the model is still accurate.

- Numerical simulations are subject to round-off errors and approximation errors in the algorithm. These errors tend to accumulate for long simulations, very large circuit sizes, and circuits with large dynamic ranges. Always keep a skeptical attitude when analyzing simulation results. If the simulation output seems to be physically impossible probably it is wrong.

- Real life circuits are subject to parasitic effects. In high gain or high frequency circuits parasitic effects may dominate its performance and behavior. The only parasitic effects included in a SPICE simulation are those you put in. TopSPICE has no capability to include distributed effects (except in the frequency domain). You must come up with a lumped element equivalent.

- SPICE is an electrical simulator. Although it is possible to simulate systems that are not purely electrical, such as motors and lamps, with SPICE, you must develop an electrical analog of the non-electrical system first.

4.2 How to Create a Schematic

The TopSPICE Schematic Editor front-end provides an easy way to create and edit circuit schematic drawings, run simulations, and plot and analyze the simulation results.

4.2.1 Selecting Symbols

The TopSPICE Schematic symbol names are not arbitrary. The first letter in the name indicates the type of SPICE element it matches. For example, any name starting with the letter Q describes a bipolar transistor. The only exceptions are the ground and power supply symbol names. The name may include a description, which are preceded by the semicolon character. In the default symbol library the names for the standard SPICE elements are kept as short as possible (usually one or two characters) so they can be selected using a single keystroke if possible. Names for all the SPICE macromodel symbols start with the letter X since they are SPICE subcircuit elements.

4.2.2 Placing Parts

To draw a new part and place it on the schematic, select the **Draw|Part** menu command. A part selection dialog appears, which shows the previously placed part (if any) in the input box and the list of parts available in the current symbol library.

To specify a new part, either pick from the list using the mouse or keyboard, or type the new part symbol name (you don't need to enter the description portion which is preceded by the character ";"). To choose the previous part, click on the "OK" button.

The selected symbol is displayed on the screen in red. Move the symbol to the desired location using the mouse. To place the part, click the left mouse button or press Enter. To cancel, press the right mouse button or the Esc key.

4.2.3 Specifying Part Attributes

Most parts require that you specify the following set of attributes: reference name, value or model name, and optional parameters. To specify part attributes, select the **Edit|Attributes** menu command while placing the part or afterwards. TopSPICE Schematic automatically increments the reference name as the parts are added to the schematic.

You can also change the attributes by double-clicking on a part on the schematic.

4.2.4 Rotate, Mirror and Flip

You can change the view of most symbols by performing the following operations: rotate, mirror and flip.

The rotate, mirror and flip commands can be combined in any sequence to obtain the desired symbol orientation and view. If you perform more than 8 rotate, mirror or flip operations the symbol is restored to its original state.

Rotate

The **Edit|Rotate** command rotates the part symbol by 90 degrees counterclockwise centered at the cursor position. The rotation operation is not defined for all library symbols. Some parts include symbols in both horizontal and vertical directions. For example, for the resistor, the symbol R is horizontal and RV is vertical. The symbol R can be rotated 90 degrees to obtain a vertical resistor, however, the disadvantage is that the symbol labels (attributes) are also rotated. The RV symbol places the labels horizontally conforming to standard schematic drawing conventions.

Mirror

The **Edit|Mirror** command reflects the symbol image right to left or viceversa with respect to the vertical line passing through the cursor position. The mirror operation is defined for all library symbols.

Flip

The **Edit|Flip** command reflects the symbol image top to bottom or viceversa with respect to the horizontal line passing through the cursor position. The flip operation is defined for all library symbols.

4.2.5 Wiring the Circuit

Wires and junctions are used to wire together parts and indicate electrical connections. To draw a wire, select the **Draw|Wire** menu command. Move the cursor to the wire starting position and click or press Enter. Now move the other wire end to the desired location and click to complete the wiring.

The wire will form a right angle at the point where you move the cursor away from a vertical or horizontal line. Hence, you can draw a straight

wire or bent wire. If you want to change the direction of the wire being drawn (for example, if you started out as a vertical wire but you want to make it horizontal) the cursor must be moved back to the starting point and then moved in the desired direction. To cancel a wiring operation, press the right mouse button or the Esc key.

A right angle wire is saved as two wire segments (one horizontal and one vertical).

The junction symbol (a large dot) indicates an electrical connection between wires or between a wire and a part pin. To draw a junction, select the **Draw|Junction** menu command and move the cursor to the desired junction point. Click the left mouse button or press the Enter key. A junction must be manually placed to indicate connections between two wires crossing at right angles. For all other connections the junction symbol is optional or it is automatically placed if the "auto-junction" feature is enabled.

4.2.6 Electrical Connection Rules

TopSPICE Schematic uses a set of rules to determine if wires and part pins make electrical connections when generating the SPICE netlist for the schematic. Connections can be made by physical connection or by using node names (see next section).

The basic rule for a physical connection is that a wire end, or a part pin "connecting" end (indicated by a small square), must contact another wire anywhere, or pin connecting end, to make electrical connection between the two. If the "auto-junction" function is enabled, junction dots are automatically placed as required. However, the junctions are optional for these connections. To connect two wires crossing at right angles you must place a junction dot manually.

Note: a part pin can only be connected by its connecting end.

Labeled connections consist of any set of wires and pins with the same node label names.

4.2.7 Node Labels and Numbers

Node labels serve three purposes: to document meaningful points on the circuit; to provide a name so a node may be referenced somewhere else; to connect electrically nodes with the same names without the need of wire connections, which sometimes is impractical.

To assign a node label to a wire or pin, select the **Draw|Label Node** menu command and enter the label text at the prompt. Move the label to a point near the wire or pin to be labeled. The label must be within one grid unit of a wire or pin to be assigned to it. To place the label, click the left mouse button or press Enter. If the label was positioned properly next to a wire or pin it will be displayed using the same color as the wire indicating that it is a node label. Press the right mouse button or Esc to cancel a label drawing operation.

TopSPICE Schematic automatically assigns node numbers to nodes without user assigned labels. The default starting node number is 1 and the node number is incremented by one for subsequent nodes. To display these node numbers select the **View|Node numbers** menu command.

Node names for ground, power rails and I/O pins are special cases. The ground node is always named "0" regardless of any user assigned name. A ground node is any wire or pin connected to the ground (GND or 0) symbol. Power rails are represented by the V+ (or POWER+) and V- (or POWER-) symbols. Any wire or pin connected to a power rail symbol pin is assigned a node name which is identical to the power rail name (reference attribute). For example, if the power rail is given the reference VCC the node will also be named VCC. Hence, all power rail nodes with the same name are tied together as they should be. Any wire or pin connected to an I/O pin is assigned the name of the I/O pin.

4.2.8 Subcircuits

The TopSPICE Schematic Editor provides several functions for creating and managing SPICE subcircuit blocks and symbols.

A subcircuit consists of two parts: the subcircuit block definition (this can be in the form of a schematic, SPICE text netlist or a model in a library file) and the subcircuit symbol. TopSPICE Schematic allows complete flexibility in implementing these two parts and it does not require that the schematic and symbol be always linked.

4.2.8.1 Subcircuit Block

To start drawing a new subcircuit block schematic, select the menu command **Subcircuit|New**. On the "New Subcircuit" dialog, enter the subcircuit name. This name is used when writing the SPICE netlist .SUBCKT statement. At this time, the subcircuit parameters if any can also

be defined. After entering the desired option, choose **OK**. Now the status display changes to indicate that you are in subcircuit mode.

The subcircuit schematic is drawn exactly in the same way as the rest of the circuit. Subcircuits can include part symbols, ground, power busses, wires, junctions and node labels. The main difference is that most subcircuit blocks must include I/O pins, which are used as input and output connections to the subcircuit. Any schematic object added while in subcircuit mode becomes part of the selected subcircuit.

After you are finished with the subcircuit schematic, select the **Subcircuit|Close** command to terminate a subcircuit and exit the subcircuit mode.

To edit or add new objects to an existing subcircuit block schematic, select the **Subcircuit|Open** command and choose the desired subcircuit from the list. For some edit operations, such as cutting objects and modifying part attributes, it is not necessary to select the subcircuit first.

4.2.8.2 Subcircuit I/O Pin

Most subcircuits require input and output connections to the outside. The subcircuit I/O pin symbol is used for this purpose. The I/O pins are also used to define the subcircuit symbol pins when using the auto symbol generator.

To add a subcircuit I/O pin symbol, select the **Draw|I/O pin** menu command. Then specify the pin name (any alphanumeric string) and pin type option. There are two types of I/O pins: "output" and "other". The type is used to determine the symbol pin position when the auto symbol feature is used. Output type pins are placed on the right side and other pins on the left by default.

The order in which pins are added to a subcircuit is significant. By default the same order is used in creating the node list in the .SUBCKT netlist statement.

4.2.8.3 Auto Subcircuit Symbol

The auto subcircuit symbol function automatically creates schematic symbols for subcircuit blocks. After you select the desired symbol shape, TopSPICE Schematic automatically determines the number pins and their positions. You can also create symbols for any subcircuit (a schematic of the subcircuit block is not required) by selecting the **Subcircuit|Symbol** menu command. Once a symbol shape and number of pins are selected, it will assign default definitions for all the pins. To change the pin definitions, select **Pin Setup**.

The subcircuit symbol is placed in the "Symbols in use" list. To actually place the symbol on the schematic, select the **Draw|Part** command and click on the subcircuit symbol name.

4.3 Waveform Plotting and Analysis

Auto Plotting

When TopVIEW is first invoked, it will attempt to "autoplot" the simulation results. TopVIEW can autoplot either #AUTOPLOT commands specified by the user or .PRINT data. If there are more than one possible autoplot, a selection menu is displayed which allows the user to pick the desired plot. To display a new plot, select **Plot|Autoplot** command.

Probe Mode

The **Plot|Probe** command allows the user to choose and plot any of the output variables available. Probe mode is automatically invoked if there is nothing to autoplot.

Multiple Plots

A TopVIEW graph can include up to 8 separate plots. All the plots share the same X axis. TopVIEW attempts to put different type of variables into different plots. For example, voltages and currents are automatically plotted in two separate plots. You can add or remove plots, or move traces to different plots by changing the trace plot number using either the **Traces|Options** or **Traces|Move/Delete** menu command.

Cursor Measurements

Two cursors are available. To place cursor 1 on a trace, select **Cursors|Cursor 1 Active/Select** command. A "cursor window" will pop-up displaying the name of the trace and data point values for the cursor. Then use the up and down arrow keys to move the cursor to the desired trace. To place cursor 2 on a trace, select **Cursors|Cursor 2 Active/Select** command.

To move the cursor over the trace, you can click the left mouse button anywhere inside the plot at the desired X value (you don't have to click over the trace), or use the left and right arrow keys. You can use the Home and End keys to quickly move to the start or end of the trace. To rapidly step through the trace use the Tab key. You can also "fix" the cursor to a desired X or Y value by using the **Cursor|Position** menu command.

Axis Limits

There are two ways to set the plot axis limits. To quickly zoom in to a section of the plot, use the **View|Zoom** command. To precisely change the axis limits, select **Axis|Limits** command.

Waveform Expressions

The user can define and plot new output variables using waveform expressions. Since such expressions are evaluated for the complete data vector, the result of an expression is another data vector. An expression can include output variables, numbers (with scale factors), constant names such as PI, arithmetic operators, math functions and user parameters. Expressions must be specified in the following form:

name=expression

To specify a waveform expression, select either the **Plot|Probe** or **Traces|Add** menu command. An expression can also be specified when selecting the X axis variable using the **Axis|X variable** command.

4.4 Sample Simulation Session

In this tutorial session we will provide step-by-step instructions to create and complete the design project for the small sample circuit shown in Figure 3. We will draw the circuit schematic, define the device models needed, specify the analysis commands, simulate the circuit and plot the results. All the files for this sample project are available in the "\Examples" folder. The SPICE netlist for the circuit is also explained.

4.4.1 Drawing the Circuit

1. From the Launch TopSPICE screen select **New Circuit**. Make sure the **Open As Schematic** option is selected. Enter the circuit project filename SAMPLE and click **OK**. The Schematic Editor is invoked with a blank sheet.

2. Select the **Edit|Title** block command. In the title line, enter the following title:

```
SMALL SAMPLE CIRCUIT
```

Click **OK**.

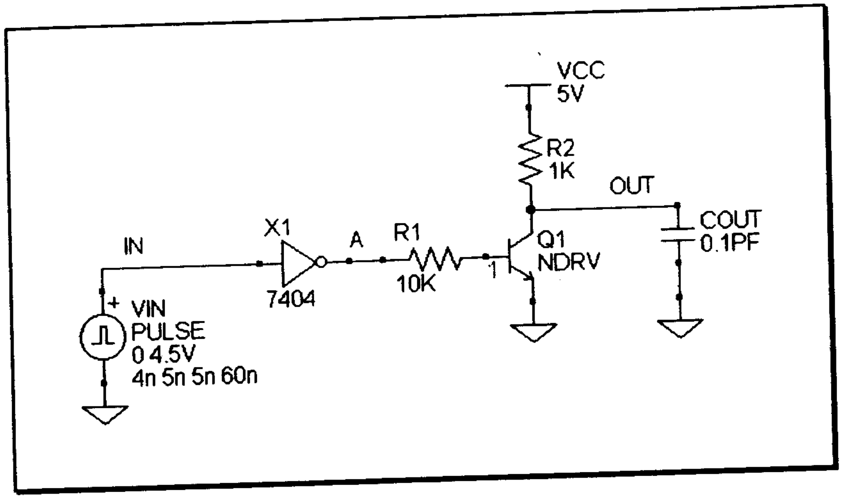

Figure 3 Small sample circuit schematic

3. You can now start drawing the circuit. Select **Draw|Part** command. Select the "V ;voltage source" symbol from the symbol list, or you can type "V" at the symbol name box, and then click **OK**. Place the red part outline approximately one inch from the left margin and two inches from the top and click. Immediately after placing the part the same part will be redisplayed. Since we don't want to repeat this part, right click to cancel. The program automatically assigns the part reference name "V1". We will change this to "VIN" later by editing its attributes.

4. Repeat for the inverter X1 part by selecting the "XINV" symbol and place it approximately as shown in the figure. The program automatically assigns the part number "INV" to it. Disregard this for the moment – it will be edited later.

5. Repeat for the resistor R1 ("R" symbol) and transistor Q1 ("Q" symbol). Make sure the right resistor pin end contacts the transistor base pin end.

6. For the resistor R2 and capacitor COUT we will use the "vertical" symbols: "RV" for the resistor and "CV" for the capacitor. Although we could rotate the standard symbols, the vertical symbol shows the attribute labels horizontally.

7. The parts attributes can now be specified or edited. Double-click on the R1 part. The "Part Attributes" dialog is displayed. In the "Value/name" box enter 10K (it can also be specified as 10000 or 1E4). Click **OK**. Repeat for R2 but this time enter 2K. For X1 and Q1 change the model names, INV and Q, assigned by the program to 7404 and NDRV, respectively. For the capacitor part we need to change both the reference and value. Edit the entry in the "Reference" box to COUT. For value enter 0.1pF.

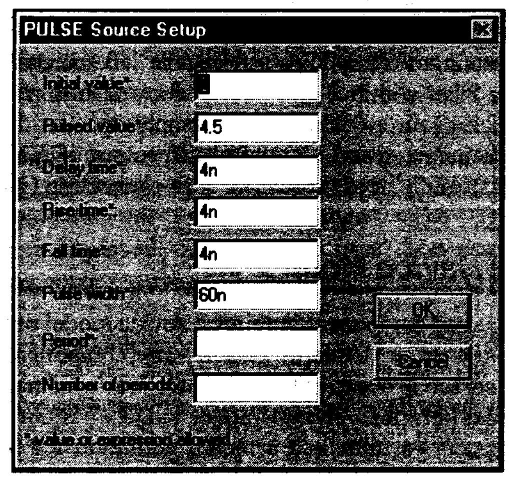

Figure 4 PULSE source setup menu

8. The voltage signal source part is a special case. Instead of the normal "Part Attributes" dialog box, the "Independent Source Setup" dialog box is displayed. Change the entry in the "Reference name" box to VIN. In the "TRANsient Spec" box select the PULSE radio button. This brings up the "PULSE Source Setup" dialog box. Fill the entries as shown in Fig. 4.

9. To add the VCC power rail symbol, select **Draw|Power rail**. On the "Add Power Rail" dialog box, enter the supply name, VCC, and its value 5V. Make sure the "positive" polarity option is checked.

10. Add all the ground symbols by selecting **Draw|Ground**.

11. The circuit is now ready to be wired. Select the **Draw|Wire** command. The cursor changes to the wiring cursor shape. Click at the VIN top pin end point. Move to the X1 input pin end and click. Repeat the procedure to wire the X1 output pin to the R1 left pin, and the Q1

collector pin to the COUT top pin. After you are finished with the wiring, right click to exit the wiring mode.

12. Place a junction symbol at the point where the Q1 collector pin, R2 bottom pin and the wire to COUT meet by selecting **Draw|Junction** command. This is needed to indicate an electrical connection to the wire.

13. Some of the nodes in the circuit need to be labeled so they can be referenced in the analysis setup commands. Select the **Draw|Label** node command. Enter the node name IN. Place the label outline above the wire connecting VIN to X1 and click the left mouse button. You must place the label within 2 grid marks of a wire or pin. To label the next node, click again anywhere on an empty spot on the schematic. Enter the node name A and place it at the output of X1. Repeat for node name OUT and place it above the wire connecting COUT.

4.4.2 Adding Device Models

Every active device and component requires a device model definition. The digital inverter 7404 is a standard TTL library component. No model definition is needed for it because the simulator will automatically search all the default model libraries. The Q1 model "NDRV" is not in the default model libraries, hence we must add a model definition for it. Select the **Analysis|Misc. commands file** command. The text editor is invoked with the file name SAMPLE.MIS. Type the following model definition:

```
.MODEL NDRV NPN (BF=20 RB=100 TF=0.1ns CJC=2pF)
```

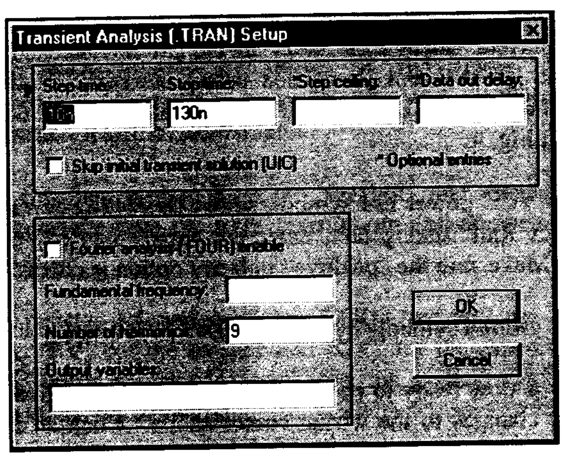

Figure 5 Transient analysis setup menu

Select **File|Exit** when you are finished.

4.4.3 Simulation Setup

Before you can simulate the circuit, you need to specify the type of analysis you want to perform on the circuit. To determine the steady-state bias point (dc operating point), specify the .OP command. To sweep the source voltage or current, use the .DC analysis command. To evaluate the circuit's time response, specify the .TRAN command. Frequency responses are obtained by using the .AC command which performs a small-signal steady-state linear analysis of the circuit.

For this example, we will perform a transient analysis. Select the **Analysis|Setup** command. The "Simulation Setup" dialog is displayed. Check the enable box for Transient Analysis. The "Transient Analysis Setup" dialog is displayed. In the "Step time" box enter `10ns`. In the "Stop time" box enter `130ns` as shown in Fig. 5. Click **OK** when finished.

TopSPICE does not save any simulation results unless output commands are specified. Check the enable box for "Save Data". On the "Save Setup" dialog box make sure the "Save everything" is checked. Click **OK**. This specifies that all the circuit variables will be saved to the binary output data file.

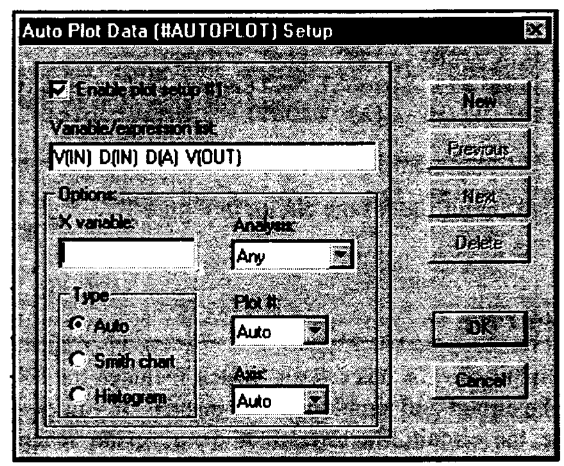

Figure 6 Autoplot setup menu

Next we want to specify to the TopVIEW post-processor program to automatically plot some of the circuit variables when the simulation is completed. Check the "Auto Plot" enable box. The "Auto Plot Data Setup" dialog box is displayed. As shown in Fig. 6, in the "Variable/expression list" box enter the following variable names:

```
V(IN) D(IN) D(A) V(OUT)
```

V(x) specifies a node voltage and D(x) specifies a digital node. For the node IN we can specify both an analog voltage and digital outputs because this is an analog/digital interface node. Click **OK** when finished.

Click **OK** to save the simulation setup. The program will create SPICE commands corresponding to the setup options and saved them in the file SAMPLE.CMD.

```
SMALL SAMPLE CIRCUIT
* Circuit description
VCC 4 0 5           ;Supply voltage
VIN IN 0 PULSE (0 4.5 4ns 4ns 4ns 60ns)  ;signal source
X1 IN A 7404        ;TTL inverter from digital library
RB A 2 10K          ;10K resistor
Q1 OUT 2 0 NDRV     ;Bipolar transistor
RC OUT 4 1K         ;1K resistor
COUT OUT 0 0.1PF    ;0.1pF capacitor
* Device model
.MODEL NDRV NPN( BF=20 RB=100 TF=0.1ns CJC=2pF)
* Control statements
.TRAN 10ns 130ns           ;do transient analysis
.PRINT TRAN/ALL V(IN) D(IN) D(A) V(OUT)
.SAVE      ;save all circuit voltages and currents
#AUTOPLOT V(IN) D(IN) D(A) V(OUT)  ;do autoplot
.END
```

Listing 1 Sample circuit file for the circuit in Figure 1

4.4.4 Circuit File

Listing 1 is a circuit file for this circuit, which includes the SPICE netlist description of the circuit and simulation commands. If you prefer you can manually create the circuit file instead of drawing the schematic. The Schematic Editor will create a similar circuit file automatically - you can view the circuit file for a schematic by using the **Analysis|View SPICE circuit file** menu command.

The following is a description of the circuit file:

Line 1 is the title line. SPICE always expects the first line to be the title line. Title lines can contain any text.

Line 2 is a comment line. Comment lines, which are indicated by a "*" character on column 1, have no effect on the circuit simulation but are helpful in understanding the netlist file.

Lines 3 through 9 describes the rest of the components in the circuit.

Line 11 defines the transistor model NDRV.

Lines 13-14 specify the simulation and output commands.

The last line in the circuit description is the .END statement. All other statements must come before it.

4.4.5 Run Simulation

To simulate a circuit, select the **Analysis|Run Simulation** command.

During the simulation, TopSPICE displays the simulation status information such as sweep voltage value or time.

If errors occur during the simulation (the most common are syntax errors), the simulation is aborted and error messages are printed to the output file. If the simulation completes successfully, the graphics post-processor TopVIEW will be automatically loaded.

After the simulation is successfully completed, the TopVIEW graphics post-processor is automatically invoked and the waveforms plotted as shown in Fig. 7.

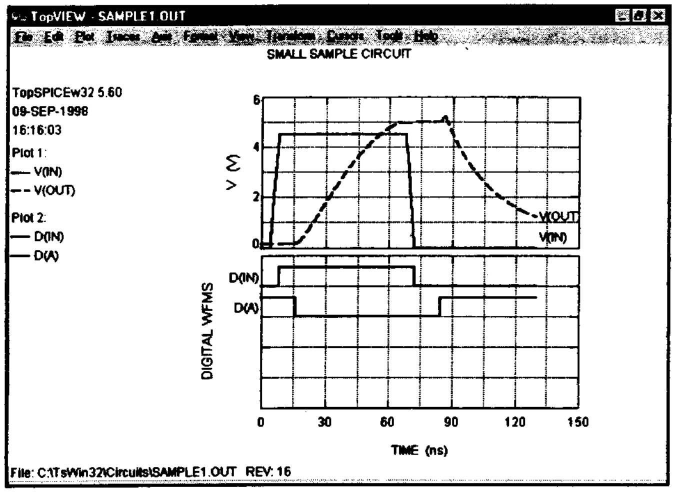

Figure 7 TopVIEW simulation results plot

A Simulator Commands

A.1 Device Statements

B	GaAs FET (same as J)
C	capacitor
	POLY nonlinear capacitor
	PWL piece-wise linear capacitor
	FECAP ferroelectric capacitor
D	diode
E	voltage-controlled voltage source
	analog behavioral modeling voltage source
F	current-controlled current source
G	voltage-controlled current source
	analog behavioral modeling current source
H	current-controlled voltage source
I	independent current source
J	JFET, GASFET and MESFET
K	inductive coupling and magnetic core
L	inductor
	POLY nonlinear inductor
M	MOSFET
O	digital/analog interface
Q	bipolar transistor (BJT)
R	resistor
S	switch
T	transmission line (ideal and lossy)
U	digital device
V	independent voltage source
X	subcircuit (macromodel)
Z	MESFET (same as J)

A.2 Device Models

ATOD	analog/digital interface
CAP	capacitor model
CORE	nonlinear magnetic core
D	diode
DTOA	digital/analog interface
FECAP	ferroelectric capacitor
GASFET	GaAs FET (levels 1, 2, 3 and 6)
HNPN	HBT npn
HPNP	HBT pnp
NJF	n-channel JFET
LTRA	lossy transmission line
NMF	n-channel MESFET
NMOS	n-channel MOSFET (levels 1, 2, 3, 7, 8, 44, 49, 53, 55)
NPN	npn BJT
PJF	p-channel JFET
PMF	p-channel MESFET
PMOS	p-channel MOSFET (levels 1, 2, 3, 7, 8, 44, 49, 53, 55)
PNP	pnp BJT
R	resistor model
RES	resistor model
TRN	lossy transmission line
U3GATE	tri-state gate
UALU	ALU function
UCOUNT	counter
UEFF	edge-triggered flip-flop
UGATE	digital gate
UGFF	gated flip-flop
UMUX	multiplexer
URAM	RAM memory
UROM	ROM memory
USREG	shift register
USTIM	digital stimulus generator
VSWITCH	voltage-controlled switch

A.3 Voltage and Current Signal Sources

EXP	exponential pulse
FILE	user data file
PULSE	pulsed (single pulse or periodic waveform)
PWL	piece-wise linear (table driven arbitrary waveform)
SFFM	single frequency FM waveform
SIN	sine wave

A.4 Analog Behavioral Modeling Options

FREQ	frequency response and s-parameter table
LAPLACE	Laplace transform
POLY	polynomial function
TABLE	look-up table
VALUE	expression transfer function

The following predefined math functions may be used in expressions:

ABS(x)	absolute value
ACOS(x)	arc cosine
ASIN(x)	arc sine
ATAN(x)	arc tangent
ACOSH(x)	arc hyperbolic cosine
COS(x)	cosine
EXP(x)	exponential
IF(t,x,y)	IF-THEN-ELSE logical evaluation.
LIMIT(x,y,z)	if $x<y$ returns y, if $x>z$ returns z, else returns x
LOG10(x)	logarithm base 10
LOG(x)	logarithm base e
MAX(x,y)	maximum of x and y
MIN(x,y)	minimum of x and y
POW10(x)	power of 10
PWR(x,y)	power (|x|^y absolute value of x to power of y)
ROUND(x)	round off to nearest integer value (ex: 2.3 is 2, 11.7 is 12)
SIGN(x)	returns -1 if argument <0 or 1 otherwise
SINH(x)	hyperbolic sine
SIN(x)	sine
SQRT(x)	square root
SQR(x)	square
STP(x)	step function. 1 if $x > 0$, otherwise 0
TABLE(x,*data*)	look up x value from table *data*.
TANH(x)	hyperbolic tangent
TAN(x)	tangent
TRUNC(x)	truncate to integer value
URAMP(x)	ramp function: x if $x > 0$, otherwise 0

A.5 Control Statements

.AC	AC analysis (frequency sweep)
.ALIASES	ignored (PSpice™ netlist compatibility only)
.ALTER	alter circuit
.DC	DC sweep
.DISTO	distortion
.END	end of circuit
.ENDL	end library entry section
.ENDM	end macro (same as .ENDS)
.ENDS	end subcircuit
.FOUR	Fourier analysis
.GLOBAL	global node
.HSPICE$	select HSPICE™ syntax priority
.IC	initial condition
.INCLUDE	include file
.LIB	library file or library entry
.LOADBIAS	load initial condition data from bias data file
.MACRO	start macro definition (same as .SUBCKT)
.MC	Monte Carlo analysis
.MODEL	device model definition
.NODESET	set node initial guess
.NOISE	noise analysis
.OP	operating point information
.OPTIONS	run time options
.PARAM	define parameters
.PLOT	"line printer" plots (obsolete)
.PRINT	print data
.PROBE	save data in binary format
.PROTECT	ignored (HSPICE™ netlist compatibility only)
.RENUMBER	renumber element reference names
.SAVE	save data in binary format
.SAVEBIAS	save bias data to file
.SENS	sensitivity analysis
.STAT	device statistical distribution
.STEP	step circuit parameter
.SUBCKT	start subcircuit definition
.TEMP	set temperature
.TF	transfer function
.TRAN	transient analysis
.WIDTH	input and output file line width (obsolete)

A.6 Post-Processor Directives

#AUTOPLOT	plot specified variables or expressions
#AUTOPLOT FFT	calculate and plot FFT
#AUTOPLOT HISTOGRAM	plot histogram
#AUTOPLOT SMITHCHART	plot Smith chart
#CALC	calculate expressions
#CALC FFT	calculate the Fast Fourier transforms
#MEASURE	measure performance specs.
#REVISION:	revision info
#SUBTITLE	mark following line as the subtitle
#TABULATE	tabulate variables

A.7 Comment Statements

*	Comment lines
$	Comment lines (HSPICE™ compatible syntax)
;	In-line comments
/* */	Multiple line comment markers