An Introduction to AnaLOG

Version 5.00

The original version of this manual was written by
David Gillespie and John Lazzaro
This version was edited by M.D. Godfrey for use with AnaLOG,
with permission of John Lazzaro

ISL, Electrical Engineering Department
Stanford University

30 March 1993

Table of Contents

1. General Information

1.1 Getting Started

1. AnaLOG is a part of a large circuit editing and simulation system called LOG. LOG has facilities for digital simulation (the original LOG), analog simulation (AnaLOG), and network generation (LOGNTK). This manual refers only to AnaLOG. It is also possible to learn to use anaLOG from a set of lesson files located in the .../log/lib directory. There are five lessons. If you are in the lib directory, or your path to that directory has been set by your local administrator, they may be referred to by:

   `analog lessonN` (where N is a number from 1 to 5)
2. To run AnaLOG, give the command:

   `analog` <simulation file>
3. To exit at any time, use the Exit command from the menus.

1.2 Using the System

1. You control AnaLOG mainly with the mouse, but also from the keyboard. It takes some practice to get comfortable with the mouse in AnaLOG, but it will soon become second nature. There are three basic mouse motions you need to learn:
   - TAP. Press and release the left button on a spot. This should feel like tapping a key on a keyboard. The spot you are tapping is indicated by the cursor on the screen, usually a small arrow.
   - PRESS/DRAG. Press down and hold the left button, move the cursor somewhere else, and release.
   - RESET. Press the right button and release. This is used to cancel operations like drawing wires.
2. Some AnaLOG functions appear right on the screen, so you can activate them just by tapping the word. Others are in "menus," which you get by pressing on one of the words Frills, Editing, Cursor, or Misc, then dragging to and releasing on the word you want. If you decide you didn't want the menu after all, just release on some other part of the screen.

3. There are also keyboard commands, though everything you really need as a novice can be found in on-screen menus. When you press a key for AnaLOG, be sure the cursor is in one of the AnaLOG windows (it doesn't matter which one). There are several features of the keyboard to notice:
   - SINGLE KEYS. Many commands appear on keys as well as menus, just for convenience. For example, you can press the letter "s" to do the same thing as SCOPE in the Misc menu.
   - ":" COMMANDS. You can give full-word commands by pressing the ":" key, then typing the command. This is sometimes more convenient; for example, if you know the name of the file you want to load, typing ":LOAD FOOBAR" is faster than selecting LOAD from Misc, then pointing to the file name. [NOTE: The Unix version of AnaLOG does not support this feature yet.]
   - <ctrl>-c. Press the Control key and the "c" key together to exit from any mode you may find yourself in.

SURVIVAL TIP #1:

If you get lost or confused, press <ctrl>-c until you know where you are.

   - SPACE BAR. The space bar at most times means to refresh the screen. If the screen has gotten trashed by too much editing, press this key to clean it up. AnaLOG will also silently clean up the screen if you stop editing for a few seconds.

1.3 Circuit Editing

1. At the bottom of the screen are several circuit symbols. In AnaLOG lingo, these are called gates. The area above the blue lines is the drawing area, where you draw your circuit diagrams. When you start AnaLOG, two very special gates, the the clock icon and the AnaLOG "scoreboard," show up automatically. You are welcome to move these around or delete them if you like, but you will find them extremely handy to have nearby.
2. To begin drawing a circuit, choose the circuit symbol you want, press down on it and drag it (still holding down) to the desired place. To move a gate, just press and drag. To delete a single gate, just "throw it away" by moving it to the bottom corners of the screen.
3. If you don't see the gate you want in the menus, tap the word CAT to see the "gate catalog." You can then press and drag any of these gates to add it to your circuit. If you will be using a lot of a particular kind of gate, you can drop it into one of the several menu slots to replace another less interesting gate. If you get into the catalog by accident, press an empty part of the screen, or, as usual, hit <ctrl>-c to get out.
4. If you look closely at a gate that has been put into the drawing area, you will

see some red dots. These are called the gate's red dots, or pins, and they are the places to connect other circuit elements. The easiest way to connect two gates together is to put them down with the appropriate red dots on top of each other.

SURVIVAL TIP #2:

> The single most common mistake in AnaLOG is to stick two gates, e.g. a capacitor and a ground, so close together that they look connected but their red dots pass through each other. The gates are NOT connected, and the circuit will NOT simulate correctly. The solution is to nudge the gates apart a bit, then refresh the screen (space bar) to make sure the red dots are aligned.

---

5. If the gate comes in an awkward orientation, you can tap it to rotate it by 90 degrees. This works for gates in the circuit as well as for menu-area gates. Notice that AnaLOG distinguishes between tapping a gate, to rotate it, and pressing a gate, to move it. Practice a few times until you can tap and press at will, without getting confused. (Some gates, like switches, do special things when you tap them in the circuit. These can only be rotated in the menu area.)
6. A much neater way to connect things is with wires. To draw a wire, just tap the place you want to start, move horizontally or vertically, then tap the endpoint. This automatically starts a new wire at the endpoint, so you can make a chain of connected wires. To finish the chain, just tap the right mouse button. The whole sequence of tap-tap-...-tap(right button) takes a few minutes to learn but soon feels very natural.
7. You can move wires by pressing and dragging, just like gates. If you press in the middle of the wire, you move it from side to side but keep the same length. If you press one of the ends, you also get to stretch that end. As usual, to delete a wire just grab it and drag to the edge of the screen.
8. Wires are connective along their whole lengths, so you can be sloppy about connecting to gates as long as you at least overlap the red dot. Wires and gates are drawn in slightly different colors so you can tell them apart if you need to.
9. If you connect wires in a "T", AnaLOG "solders" them automatically. If you cross them in a +, AnaLOG assumes they are unconnected, so you must go back and tap the intersection to solder them. Because of this, "T"-connections are preferable.
10. If you should run out of room on the screen for your circuit, you have several options: press ">" or "<" to "zoom" in or out on the circuit (these are also available in the Frills menu), or use the arrow keys to "scroll" up, down, left, or right. The drawing area is essentially infinitely large; so large, in fact, that you can get lost due to excessive scrolling. If this happens, HOME in the Cursor menu will take you back to the center of the drawing universe.
11. AnaLOG can keep up to 9 circuit diagram "pages" in memory at a time. Each page is associated with a different file. To switch to a new page, press one of the digit keys 1 through 9.

1.4 Configuring Gates

1. Once you have drawn, say, a capacitor, you may wonder what its capacitance is, and how to set the capacitance you want. The way to do this is called configuration.
2. In the lower-right corner of the screen is the word ROT with a curving arrow. This tells AnaLOG that when you tap a gate, it should rotate it by 90 degrees. If you press the word ROT, it changes to MIR. Now if you tap a gate it mirrors itself left-to-right. Pressing MIR leads to MIR with a bar (up-down mirroring), then to CNFG in yellow, then back to ROT. If you tap a gate while in CNFG mode, instead of reorienting it you open it up for configuration.
3. Each kind of gate has a set of "attributes," usually numbers, that tell more specifically what kind of circuit element it is. For example, AnaLOG capacitors have a "capacitance" attribute, and transistors have a hat full of attributes about width/length, kT/q, parasitic capacitances, etc.
4. One of the attributes is highlighted. To highlight a different attribute, press the up- and down-arrow keys.
5. To enter a new value for an attribute, highlight it, then just type in the value you desire.
6. Some attributes can have scale factors, e.g., "m" for milli- and "u" for micro-. These will show up as, say, "1.2mV", meaning 1.2 millivolts. When you type a new value, you can always leave off the V since that part is obvious, but don't forget to include the "m" if that's what you want! A common mistake is to change 20pA to 30A, thus seriously perturbing the operation of the simulated circuit. When in doubt, you can always type in scientific notation. (i.e. "1.2E-4")
7. Some attributes are choices instead of numbers. For example, voltage sources let you choose a waveform of either "DC," "Pulse," or "Sine." You can type whole words to change these attributes, or use the arrow keys to cycle through the choices.
8. To return an attribute to its default value, just press ENTER all by itself on that line. Some attributes can be blank, in which case whatever feature they correspond to is disabled. Others must have values, and AnaLOG will use some default if you try to enter "blank."
9. When a gate is first created, all its attributes have their default values. However, if you change a gate's attributes, then drop that gate into one of the menu area slots, then new gates drawn from that slot will get those new values for their attributes. You can also reconfigure gates in their menu slots using Probe mode (in the Frills menu.)
10. To finish configuring a gate, press the right mouse button again, or hit <ctrl>-c.

1.5 More about Editing

1. In the Editing menu there are commands for doing large-scale circuit editing that

would be unreasonable with just the basic mouse motions.

- DELETE: Press Editing, drag to Delete, release left mouse button. Now when you move up into the drawing area, the cursor is shaped like a pair of scissors. You can now delete individual things by touching them (with the point where the blades of the scissors cross), or you can delete everything in a rectangular region by pressing at one corner of the region and dragging to the opposite corner. Press the right button to exit Delete mode. You can also press the "d" key to get the Delete mode scissors. Due to an intermittent bug, you may need to click the right button a second time for the scissors to appear.
- PASTE: After you have deleted a group of objects, you can select Paste to "paste" down copies of what was in that region. The Paste command highlights the region that was actually deleted; you can paste back in there for a simple "undelete," or you can paste other places for moving or replicating a chunk of the circuit. This is also on the "*" key.
- COPY: This is similar to Delete, except that it doesn't actually delete the contents of the rectangle. It is for making copies of something without disturbing the original. After you have copied your object or area of objects, you are automatically switched into Paste mode. This is also on the "/" key.
- MOVE: Move an area. This command is like a DELETE followed by a PASTE; it moves an object, or all the objects in a given area, to a new location. It is also on the "m" key. Note that the pressing "m" and tapping on a wire moves the wire in a different (often useful) way than simply grabbing the wire.
- Implicit MOVE/COPY: Pressing in empty space and dragging out a box automatically does a combination MOVE and COPY command. The first place you select moves the contents of the box to that place; if you press at additional places, extra copies are drawn there.

2. Open/Close commands: These are for expert users only, because using them improperly may damage AnaLOG's internal data structures. What they do is shift things around on the screen to make extra room for additions, or to close up excess space. If you want to do this, use Delete/Paste instead, or ask someone.
3. The Labels command in the Frills menu lets you add text labels to your diagram. Select Labels, type the label you want, then grab it and drag it, as usual, into place. You can edit a label by tapping it, then using the keyboard. Labels generally have no effect on the simulation, but they make plots look much nicer. Also, certain tools such as LOGNTK can read the labels for additional information about the circuit.
4. The Boxes command in the Frills menu adds yellow dashed boxes to the diagram, again for cosmetic purposes only. Once you're in Boxes mode, you can drag out boxes in the usual fashion. To edit boxes, grab a corner for reshaping or an edge for moving. Both labels and boxes can be deleted by throwing them off the bottom corners of the screen.
5. To save your circuit, select Save in the Misc menu, then type the file name to

use. It's a good idea to hit Shift-S, or type ":SAVE", periodically to save the circuit under the previous name if any changes have been made. ":SAVE" with no file name argument saves all pages which have been editing back into their original files. Circuits are saved in files ending with ".lgf" (for LOG Format). The previous version of the file is backed up under the same name but with ".lfo".

6. To load a circuit you have saved before, select Load from the Misc menu. You will get a list of saved circuits in your directory. You can tap one of these to load it, or just type the name. AnaLOG saves the complete simulation state along with the circuit, but you may want to do a RESET after loading anyway.
7. The Grid command in the Cursor menu switches between the usual arrow cursor and a cross-hair cursor which is useful for aligning things.
8. The Yardstick command in the Frills menu lets you draw an arrow on the screen which you can use to check the geometric arrangement of objects on the screen. The yardsticks are not "real"—they go away when you refresh the screen. You can also press the "y" key to get a yardstick.
9. If you select Status from the Misc menu, you get a set of screens describing various aspects of the AnaLOG system. Move from screen to screen with the arrow keys. The first screen includes the date and time, amount of memory free (approx.), plus various other stuff. Later screens display the status of the AnaLOG editing buffers and keyboard mappings, then there is a screen for the analog simulator.

SUMMARY OF SURVIVAL TIPS:

SURVIVAL TIP #0:

Select Exit from Misc to exit AnaLOG.

SURVIVAL TIP #1:

If you get lost and confused, press <ctrl>-c until you know where you are.

SURVIVAL TIP #2:

The single most common mistake in AnaLOG is to stick two gates, e.g. a capacitor and a ground, so close together that they look connected but their red dots pass through each other. The gates are NOT connected, and the circuit will NOT simulate correctly. The solution is to nudge the gates apart a bit, then refresh the screen (space bar) to make sure the red dots are aligned.

---

2. Analog Simulation

2.1 Basics of Analog Simulation

1. AnaLOG is simulating your circuit constantly, even as you edit. However, it requires your circuit to be "complete" before it can start the simulation. That is, every pin must be connected to another pin, not left dangling. If the circuit is incomplete, i.e., there are still unconnected pins on some of the gates, AnaLOG will display a message to that effect on its scoreboard. As soon as the circuit is complete, the scoreboard will switch to "Simulation in Progress".
2. The gate that looks like a box with an arrow on it is the METER. To monitor your simulation, sprinkle meters liberally around on all of the interesting voltages. There is no limit to the number of meters you can have. Remember that the meter's red dot is at the tip of the arrow, so the tip must be exactly touching the wire or pin of interest. If the meter is unconnected or unhappy, it will display a "~ " sign.
3. The "clock icon" is the round thing in the upper-right corner that looks vaguely like a clock. Its importance is in the two numbers directly below it. These are the current time and current timestep, respectively. At the beginning, these are both zero. As soon as the simulation begins, the current time will start advancing. How fast it is advancing is shown as the current time-step, and may vary depending on how fast things are happening in the circuit.
4. A common error is to watch, say, a capacitor charging, then go in and double the capacitance, and complain that it still seems to be charging "at the same rate." If you look at the clock icon, though, you will find that the timestep is twice what it used to be: AnaLOG can simulate the ramp function just as quickly in "real time," but the circuit is (correctly) half as fast in "simulated time." Make a habit of paying attention to the clock so that your observations have the correct frame of reference in time.
5. Although simulation happens continuously, it is sometimes necessary to "reset" the circuit back to its starting conditions. For example, suppose you remove a wire from a running circuit and the voltages that were in the circuit at the time are "impossible" for the new circuit. The simulator will work as hard as it can to figure out what to do, but may eventually give up and print a message. Another example is that you may want to reset various nodes to the initial voltages that you have programmed for them, as described in the next paragraph. In any case, all you have to do is select RESET in the Misc menu, or press the "R" key. This sets the simulation clock back to 0 seconds, and returns the voltages to their initial values.

6. If you open any AnaLOG gate for configuration, one thing you will find is a set of three attributes for every pin on the gate. These attributes are present voltage, reset voltage, and parasitic capacitance, respectively. Every pin has a slight parasitic capacitance to ground, initially 10fF. This capacitance must exist for the simulation to run, but you can set it lower if you find that it's in your way. The "present voltage" is an actual indicator of the voltage currently on that pin, and it changes while you watch during simulation. You can enter new values here if you wish, but the simulator may not like you if you go around making instantaneous changes in the circuit's voltages. The "reset voltage" is used by the RESET command. If you leave it blank, AnaLOG will solve for an appropriate initial voltage at RESET time. Notice that Present and Reset voltages are for nodes, not gates: if you change a pin's reset voltage, then open up another gate connected to that same node, the reset voltage shows up there, too.

7. When you reset, you will notice that the timestep jumps down, usually to about 1E-18 seconds, then starts increasing. This is partly because AnaLOG is cautiously trying to discover how fast it can safely proceed in the simulation, and partly because it has to solve for unspecified initial voltages by simulating the brief charge-up that would occur when you powered up a real circuit from scratch. If you push RESET and the voltages seem to be frozen, don't despair – wait for the timestep to get out of the femtosecond range!

8. The AnaLOG scoreboard has some words on its left and right sides. On the left is Memory, with indicators Set and Erase. If you tap the word "Erase," all of your "reset voltages" will be erased. You might do this if editing has caused your reset voltages to be no longer appropriate, and you want AnaLOG to solve the circuit from scratch. If you tap the word "Set," all of the present voltages in the circuit are copied into the reset voltages. For example, you might turn your input waveforms off, let the circuit settle to a stable state, then push Set so that next time you RESET, the circuit will start out at that stable state and you won't have to wait again.

9. AnaLOG is normally very cautious about voltage changes. If it is going along at a particular timestep and some node suddenly starts changing more than about .1V per timestep, AnaLOG will reduce the timestep and simulate in smaller chunks. This is because trying to simulate fast voltage changes all at once results in ugly and inaccurate waveforms.

10. Sometimes all you are looking for is a DC point anyway, and you don't care how ugly your transient waveforms are. In this case, you can touch the word "Relaxed" on the scoreboard. This tells AnaLOG to take timesteps as large as it can regardless of how fast the voltages are changing. Touch "Exact" to return to an accurate simulation. Often what people do is touch Relaxed right after a RESET so that the initial charge-up proceeds quickly, then touch Exact when the timestep gets up to the picosecond-nanosecond range.

2.2 A Tour of the Analog Gates

1. Clock Icon (TIME). This displays the current time and timestep. It also has a square in the center that changes color to indicate how the simulation is going:

Black	Simulation is turned off
Red	Computing the next timestep
Yellow	A timestep has been computed

   If you configure the clock icon you will find that it can display a few other time-related values which may be of interest.

2. Global power supply (VDD and GND). VDD defaults to +5V. You can change the value of Vdd in the NUMBERS gate, but Gnd is always zero, and Vdd is always positive.

3. Terminals (TO, FROM). These are the two arrow-shaped gates in the Catalog. They are equivalent except for appearance. They are used for putting names on signals: to name a signal FOO, connect a terminal to it, then tap the space next to the arrow and type FOO. If there are several terminals in the circuit with the same names, they are all electrically connected as if by wires. Signal names are also used with the Scope mode, to be described below.

4. Scoreboard (NUMBERS). This gate serves many purposes. On the screen it displays whether simulation is proceeding and why; it gives controls for set, erase, exact, or relaxed modes; and, when you tap the center bar, it lets you configure AnaLOG's "global" parameters, such as the value of VDD and the maximum allowable timestep. More on these parameters below.

5. Transistors (NFET5, PFET5, PFET6). Typical MOSFETs, with models accurate enough for circuits in Carver Mead's *Analog VLSI and Neural Systems* book. Compared to SPICE, they are between levels one and two in complexity. AnaLOG transistors are tuned for a 2 micron process, and to be most accurate in the subthreshhold regime. This makes them well-suited to analog circuits that depend on subthreshold operation, but only roughly accurate for, say, simulating digital circuits. For the simulation to be more accurate in strong inversion, change the Beta of the NFETs to be 59uS/A and the Beta of the PFETs to be 25uS/A; there parameters result in a close match to the simple above-threshold MOS model presented in most textbooks. The AnaLOG transistors are completely symmetrical between source and drain. PFET6 is a four-terminal transistor, for simulating PFETs with floating wells. NSPC1 and PSPC1 are experimental FET models contributed by Bhusan Gupta. In addition to simulating in analog, they are perfect for use with the SPICE netlist tool, LOGSPC (see LOGSPC section of on-line HELP). The bipolar transistors and the diode (NPN1, PNP1, and DIODE) are experimental Ebers-Moll models.

6. Capacitor (CAPFLOAT). Typical capacitor, defaults to 1pF. Notice that, like all circuit elements, it has a slight parasitic capacitance to ground as well as the capacitance between the two leads.

7. Resistor (RESFLOAT). A plain linear resistor. For a more VLSI-like resistor, see

HRES below.

8. Transconductance amplifiers (OPAMP, WRAMP). These are simulations of the narrow- and wide-range amplifiers in Carver's book. Their behavior is modeled after the circuits at the transistor level (as modeled by AnaLOG), at least if you use them in the normal range of voltages (between 0 and 5V). A little red light turns on if the voltages go outside the range in which the model is a safe approximation for the "real" circuit. Note that the "Iset" pin at the bottom wants a voltage, not a current. If you want to set the current manually, connect a voltage source to the pin and also to a standalone transistor, then measure the current through the transistor.
9. Rectifiers (HWR, FWR). These are half-wave and full-wave rectifiers. An HWR models an OPAMP driving a two-transistor current mirror; an FWR models is like a pair of HWR's.
10. Horizontal resistor (HRES). This big ugly thing is the "horizontal resistor" circuit, which predates the circuit in Carver's book. Obsolete for new designs.
11. Ganglion (GANGLION). This is the ganglion circuit, as described in Carver's Book.
12. Voltage/current meter (MMETER). This meter displays the voltage on whatever wire or pin it is connected to. If you configure it you can change it to a current meter, in which case it measures current into the gate from whatever pin it is connected to. (If you connect a current meter to the middle of a wire, it doesn't work because the direction of the current is not well- defined.)
13. Special magic current meter (ISCOPE). This is a widget for use in plotting currents in Scope mode, and will be discussed in the later section on Scope mode.
14. Voltage source (VDIFF). This gate creates a differential voltage between its two pins (the bottom pin is typically connected to ground). It has a finite output resistance (initially 50 ohms), so it is not an "ideal" source in the sense of SPICE. You can use it as a simple DC source with a certain voltage (set by configuring the gate), or you can set to output a pulsed or sinusoidal waveform. This is all described below, under Input Waveforms. Voltage sources display the current-limiting condition by lighting up a little red light, just like the ones on the lab bench! They also have a switch, in the lower-right, that you can switch to freeze the waveform in time.
15. Current source (IDIFF). Basically a current-sourcing version of the voltage source. It can generate the same kinds of waveforms. It voltage-limits if the voltage across the pins goes below the "crowbar voltage," 0.1V by default.
16. Step generator (STAIRS). A voltage source generating a stairstep function, which moves from one voltage to another in a certain number of steps, then repeats.
17. Voltage switch (VSWITCH). A voltage source that can be manually switched between "on" and "off" voltage levels. This one also has a tiny switch on its face, this time to select which voltage is generated. It can also be set in "monostable" mode, in which a tap on the switch will turn it on for a certain period of time, then back off again automatically. Note that voltages here are implicitly relative

to ground.

18. Current switches (ISWITCH1, ISWITCH2). Current-source and current-sink versions of the VSWITCH. The current source (arrow pointing out) is like an IDIFF connected to VDD, and the current sink (arrow pointing in) is like IDIFF connected to GND.
19. Piece-wise linear two-terminal element (PWL, contributed by Harold Levy). Implements an arbitrary two-terminal element, with an i-v transfer curve. Cd to the log/lib directory, start analog, and load the file pwl-test.lgf to learn more.
20. Resonant tunneling diode (RTD, contributed by Harold Levy). Models a class of quantum well devices. If you do research in this area and want more details, contact har@caltech.edu.

2.3 Analog Input Waveforms

1. DC. This is just a constant amount of voltage or current.
2. Pulse. Depending on how you set it up, this can be a square wave, triangle, or sawtooth. Referring to the attributes for Pulse mode, the output waveform starts at "initial voltage," then, after "delay time" (plus "reset time" for the first cycle), it begins ramping to "pulsed voltage," at a rate determined by "rise time." It stays at that voltage for "pulse width," then ramps back to the "initial voltage" in "fall time." The cycle repeats the the period specified. NOTES: If rise and fall time are both relatively small, you get a square wave. If one is large and the other small, you get a sawtooth. If both are large, and the pulse width is zero, you get a triangle.
3. Sine. This generates a sine wave, with the frequency, offset, and amplitude desired. The "phase" is specified in degrees. If the "delay" is nonzero, the wave waits that amount of time after RESET before it begins to change.
4. Stairs. Starts at "initial voltage," waits there for "time per step" time, then ramps to next voltage in time "rise/fall time." This occurs "number of steps" times, for a total of "steps"+1 voltages between "initial voltage" and "ending voltage," inclusive. After the cycle, it ramps all the way back to "initial voltage," again in "rise/fall time," and starts over. If "initial delay" is nonzero, the waveform remains constant for that long before beginning.
5. Bistable switch. If the switch is "off," output is "initial voltage." If it is "on," output is "pulsed voltage." When the switch is changed, output switches from initial to pulsed voltage in time "rise time," or from pulsed to initial voltage in time "fall time" (regardless of whether the voltage was actually increasing or decreasing). On RESET, the switch is set to on or off according to "reset state."
6. Monostable switch. Output is normally "initial voltage." When switch is tapped, output ramps to "pulsed voltage" in time "rise time," holds for "pulse width," them ramps back in time "fall time."

3. Other Simulation Features

3.1 Simulation Goodies

1. If there are any circuit elements with unconnected pins, AnaLOG will refuse to simulate. If you meant to leave the pin unconnected, you must connect it to something like a "dummy" capacitor to ground. If there are unconnected gates, the NUMBERS icon will display a message of the form "OPAMP is unconnected".
2. If you give a ":DIM" command, AnaLOG gates will be drawn in dim colors until they are fully connected. Another ":DIM" turns this mode off.
3. Select Probe in the Cursor menu (or press the "." key) to enable Probe mode. The cursor changes to a different style arrow; when you move this arrow over a gate or wire, a message below the AnaLOG NUMBERS icon shows the internal state of the gate and/or node. Probe mode also displays the name of a gate being touched. Select again to turn the mode off. You can also press "e" or "x" ("Examine") for a temporary Probe mode which turns off automatically when you click the right button.
4. Select Glow in the Cursor menu (or press the "g" key) to enable Glow mode. Wires "glow" in colors according to the voltages they carry:

Black	means at (or below) ground.
Dark red	means within one transistor threshold of ground.
Red	means an intermediate voltage.
Pink	means within one transistor threshold of VDD.
White	means at (or above) VDD.
Green	means the wire is not driven by an analog gate.

   Glow mode may be left on at all times, though it may make the simulation a little slower.

5. Select Simulation in the Misc menu (or press the "o" key) to turn the simulator off or back on again. When simulation is off, the gates and nodes of the circuit appear to freeze at their last simulated values. Turning the simulator back on resumes the simulation.
6. Press the "t" key to simulate for one time-step, then stop. You can press t as many time as you wish to see the simulation evolve, then press o to turn full-speed simulation back on.
7. LOG tries to balance simulation speed against response time for the mouse and keyboard. If you let go of the mouse and don't press any keys, LOG slowly shifts the balance to prefer simulation. When you again use the mouse or keyboard the balance reverts to favoring editing. You can press the "f" key to switch immediately to "fast" mode (preferring simulation over editing).

3.2 Scope Mode

1. Numbers are great, but real people like to see plots of their output waveforms. In LOG, the mechanism for this is called Scope mode. To use it, first connect terminals (TO or FROM) to each signal to be monitored and give the signals unique names, then select SCOPE in the Misc menu, or press the "s" key. The scope can display both digital and analog signals.
2. The Scope screen consists of a large field of dots representing the "divisions" on a normal scope face, an area to the left of the dots where signal names can go, and some menu words at the bottom. Use the mouse as usual to select menu words for the functions that you want. In particular, touch QUIT (there's one on each side, for your convenience!) or hit <ctrl>-c to return to the circuit diagram.
3. To display a certain signal on the scope, just type the name of the signal and press ENTER. The name will appear somewhere at the left edge. You can use the mouse to press and drag the name to any vertical location. To remove a signal name, drag it off the top or bottom edge.
4. Initially, the scope is not "triggered," so nothing is displayed. To begin taking data, touch Trigger. The scope also automatically triggers when you reset the simulation; there is a handy Reset on the menu too. A few moments after you trigger the scope, lines to the right of the signal names will begin to appear.
5. In all probability, the lines will be absurdly wide or ridiculously thin. In other words, the time scale is wrong. To fix this, press the "<" or ">" keys to shrink or expand (respectively) the plot until it fits nicely on the screen. You can also use the arrow keys to scroll around the plots in time; by scrolling and expanding you can focus in on any part of the simulation history.
6. Touch Config to change the various Scope mode parameters. You will find here the seconds-per-division and seconds-at- left-edge parameters that the zoom and scroll commands manipulate graphically; you can set them numerically if you prefer. The current time and timestep are displayed for your reference in setting these values.
7. Scope mode has a limit on the number of data points it will take before stopping, initially 2000. You can set this in the Config screen, but beware that if you set it too high AnaLOG will run out of memory and lose ALL of your data!. This screen also displays the number of data points taken so far. What happens when this reaches the maximum depends on the "trigger mode" attribute:

   Manual:

   Scope starts taking data when you touch "Trigger," and stops when you touch "Trigger" again, or when all the data points are used. Beware that when memory fills, there is no way to resume taking data except by retriggering and losing all of your old data! LOG prints warning messages when you are about to run out of data points.

   On Reset:

   Like Manual, but also triggers automatically on RESET. This is the default mode.

Triggered:

Like Manual, but also triggers automatically when a triggering pulse is detected on the signal that you specify. Not relevant for AnaLOG, because triggering on analog signals has not yet been implemented.

Continuous:

Triggers manually or on RESET. When data memory is full, old data points are removed from the left and new ones added to the right. Never stops triggering unless you turn it off by hand.

On Reset mode is typically used when you are interested in the first data points of a long simulation; Continuous mode is for when you are interested in the most recent data points.

Note that the word "Trigger" is lit whenever the Scope is taking data, and unlit otherwise.

8. Normally, Scope mode records exactly one data point per timestep returned by the simulator. The "minimum timestep" and "maximum timestep" attributes can modify this. If you would like to conserve memory and take data points only at, say, 100us intervals, you could set "minimum timestep" to 100us. Scope mode will then record a data point only if it's been at least 100us since the last point was taken. The other attribute, "maximum timestep," does not work with the analog simulator.

9. When you touch (instead of dragging) a signal name on the left edge, you Configure the parameters for that signal's particular trace. In AnaLOG, you can set volts-per-division, plus an "origin" voltage for plotting small signals (for example, if you have a signal that varies slightly around 2V, set the origin to 2V and the scale to, say, 50mV per division). You can also change the color of the trace, very helpful for plotting several traces on top of one another. You can also set logarithmic mode, in which case you specify "Vzero" which is the voltage to plot at the origin, with one decimal order of magnitude per division above and below.

10. To plot an AnaLOG current, you must take an ISCOPE gate, which is the funny-looking meter with a hook coming out the back, connect it to the pin to be measured, and connect a terminal (TO or FROM) to the "hook." Now, when you monitor that name in Scope mode, you will measure Amps instead of Volts. Note that the "hook" pin of the ISCOPE is a special "current-mode" data type, and any attempt to connect it to a normal analog signal will fail. If an ISCOPE ever appears to undergo spontaneous combustion, it is because you put it down with the hook touching an analog wire.

11. If you try pressing down the mouse anywhere in the grid of dots, you will get a horizontal and/or vertical line whose meaning depends on two indicators at the bottom of the screen:

Absolute:

Display the value of time, or of the currently-selected signal, that corresponds to the mouse position.

Delta:

Display the difference in time or signal value that corresponds to the distance between the current mouse position, and the position at which it was first pressed down.

Value:

Display the value of the current signal as of the time that corresponds to the mouse position. For example, you move the mouse to the "2.3ns" point, and the Scope displays the value that the current signal had at that time.

Slope:

Display the slope of the line between the starting and current mouse positions, where rise is measured in the units of the current signal, and run is measured in seconds.

Time:

Display absolute or delta time, in seconds.

Freq:

Display delta time, in Hz (= 1/seconds).

Signal:

Display the value of the signal whose name is highlighted, in Volts or Amps. As you touch this word, each signal is highlighted one by one. You can also select a current signal by tapping its trace on the screen.

12. The word ON is a "power switch" for the simulation. If you touch it, it changes to OFF and the simulation halts. When you touch it again, simulation continues. (In the circuit-diagram screen, this function is called Simulation in the Misc menu.)
13. When you touch Dump, the Scope writes out all of the data into a file in a standard format. You may want to do this in order to use a more hefty data analysis tool with data generated by AnaLOG.

4. Plotting Circuits and Data

4.1 Plotting a Circuit

1. To make a hardcopy plot of your circuit, press "p" or select Plotting in the Misc menu. You get a screen with an (admittedly ugly) view of what your circuit will look like on paper.
2. To zoom in on an area, just use the mouse to sweep out a rectangle around that area. Press Zoom out to return to viewing the full circuit. The Zoom feature uses "markers," which are little red brackets that show up in the circuit diagram. You can move the markers around in circuit-editing mode, too, and turn them on or off with the Markers command in Frills.
3. Touch Config to configure the plotter. You can change the font that is used for labels (there is a wide selection of fonts available, though most of them are silly). You can select how the plot will be oriented on paper (default is to let LOG choose the orientation to make it come out as large as possible). You can set the size of the paper if your plotter is too primitive to know that for itself. (For the laser printer, the printer always determines the paper size itself.)
4. Touch Options to configure the circuit-diagram plot. The only interesting attribute here is the size of the solder dots (oh, boy!).
5. When you are all ready press File and away it goes, to the system Postscript printer. (The PLOT button used to control pen plotters directly, but in the Unix version it does nothing.)
6. You can also plot into a PostScript file, for inclusion, say, in a TeX document. Set the file name in Config, then touch File.

4.2 Plotting your Data

1. Take data on all interesting signals using the Scope screen.
2. To get a high-quality plot, touch the word PLOT on the SCOPE screen. This switches to an alternate menu of commands.
3. The word "Y-axis" is lit up, meaning that if you touch any signal name or signal trace, or the word "Time," or type any signal name or arithmetic expression involving signals, then that signal will go on the Y axis of the graph.
4. When "X-axis" is lit, you are selecting what should go on the X axis of the graph. After both axes have been specified, you can change your mind by touching one of the two words, then touching a new signal.
5. In the Options screen, you get function name, plus label, units, log/linear, and minimum and maximum values for each axis. If you don't specify a min and max,

they are calculated from the data. You can enter any arithmetic expression as the function; variables in the expression may be any signal name from the Scope screen, or Time, or one of A through F. These last are user-defined parameters you can change in the Variables screen.

6. You can also specify line style, and a symbol to plot on data points.
7. Normally the axis labels are drawn at a larger size so that you can read the numbers and exponents. When plotting on paper, the axis labels are regular-size.
8. Zoom out just clears the min and max for both axes to blank.
9. Do not use Plot. It does nothing in this version.
10. Touch File to send the current graph to a file or device, as described in the Config screen. Default is the Apple LaserWriter.
11. The simulation is still running while you are in the plotting tool. If the scope is still triggered, the set of data plotted will be those taken as of the time the screen was last refreshed, or the Plot button was pushed. Thus if you just sit and press the space bar, the plot will be slightly different each time. If this is confusing, we suggest turning the simulator off first by changing ON to OFF in the Scope menu.
12. Touch QUIT to return to the Scope.

5. Reference Material

5.1 List of Commands

1. This is a partial list of commands that may be entered in their full-length form (i.e., press ":", then type the command). Many of these commands may also appear in the file "log.cnf" which is read when LOG starts up.
2. In this list, "ON/OFF" means either the word "ON", to turn an option on, or "OFF", to turn an option off, or no word at all, to "toggle" the option between on and off.
3. Many commands have single-key equivalents. The bindings shown here are the defaults, which may be changed by the MACRO command.
4. The following commands, when used in the form of single-key commands, may be used even in the middle of drawing a wire, selecting a rectangle, etc.:

ALTPOSN	GLOW	OFF	SIM/ONOFF/POWER
ARROW	GLOWSOLDER	ON	SNAP
AVOID	GRID	PAGE n	TRACE
CLOSEFILES	HELP	PROBE	TRACEFILE
HOME	QUIET	VERBOSE	DOTS
INVISIBLE	RESET	ZOOMDN	DUMP
INVLABEL	RESPONSE	ZOOMUP	DUMPFILE

   Other commands will cause any other operation in progress to abort before the new command begins.

5. Here is the complete list of LOG commands:

   ABORT

      (<ctrl>-c) Abort the current command, mode, or screen.

   ALTPOSN

      ("," key.) Scroll to an alternate location on the page. LOG essentially maintains two current locations on the page, like the "mark" in the Emacs text editor; this command exchanges them.

   ARROW

      Revert to standard arrow-shaped cursor. This undoes any previous PROBE or GRID command.

   BOX

("b" key.) Draw a dashed box.

CAT

(Shift-"C" key.) Switch to the Catalog screen.

CENTER

Shift the entire circuit so that it is centered in the LOG circuit universe, i.e., so that a HOME command will home to the center of the circuit. Note that this actually moves the circuit elements, whereas scrolling with the knob just changes your view on the elements.

CLEAR

Delete the entire circuit page. Use with caution!

CLOSEFILES

Close the "dump" and "trace" files, if they are open. These files will automatically be re-opened (for appending) if they are again used. The dump file is used by the "Dump" command in the Scope. The trace file is used by various LOG debugging commands.

CLOSEH

Enter "close-horizontal-space" mode, similar to OPENH except that it deletes the contents of the box you draw and shifts other things to the left. DANGEROUS!

CLOSEV

Enter "close-vertical-space" mode, similar to OPENV. DANGEROUS!

CNFG

("c" key.) Switch gate-tapping mode to open a gate's configuration screen when it is tapped.

COLOR screen-color red-value green-value blue-value

COLOR color-name screen-color

Customize the LOG color scheme. LOG's colors are determined in two steps. A "palette" of up to 16 available colors is specified by the first form of the command. "Screen-color" is either a number from 0 to 15, or one of the names listed below. Each of the red, green, and blue values run from 0 (black) to 255 (full intensity). The second form of the COLOR command determines which of these 16 colors is used for various kinds of objects. The color names defined by LOG are listed in a later section.

Screen-colors:

0 GRAY	4 MRED	8 BLACK	12 DCYAN
1 RED	5 ORANGE	9 PINK	13 DRED
2 GREEN	6 CYAN	10 DYELLOW	14 LGRAY
3 YELLOW	7 WHITE	11 XGREEN	15 CRED

On eight-color screens, all sixteen names or numbers may still be used; LOG combines similar colors internally. The effect will be that changing the red-green-blue content of one of the above colors may affect other colors as well on an 8-color screen.

COPY

("/" key.) Enter copy-area mode.

DEFINE

(Shift-"D" key.) Display the simulator's definition for a gate-kind, specified either on the command line or by touching a representative gate. Pressing shift-"D" in the Catalog screen also displays the definition for the gate the cursor is over.

DEL

("d" key.) Enter delete mode.

DOTS ON/OFF

Control dots-visible mode. When this is on (default), red connection dots on gates are always visible. When this is off, red dots are usually visible only on pins that are not connected to wires.

DUMP

Write out a "debugging dump" to the printer. For LOG maintainers only.

DUMPFILE filename

Close any current dump file, then select "filename.text" as the new dump file. If this file already exists, new text will be appended to it.

EXAMINE

("e" or "x" key.) Enter Probe mode temporarily. The cursor returns to normal as soon as you click the right button. This is in contrast to the PROBE command, which turns the Probe cursor on until you explicitly turn it off again.

EXIT

Exit from LOG. Also in the Misc menu, or on the Shift-Q, Shift-Z, and Control-D keys.

EXTRACT

Delete and re-paste the entire contents of the page. If LOG's electrical connectivity information becomes corrupt, this will usually fix it.

FAST

("f" key.) Turn on fast mode. This happens automatically after a certain amount of time passes with no mouse or keyboard inputs.

GET gatename gatename gatename ...

Load the specified gates into the Catalog screen from the gate library. The gate names may include wildcard characters '*' and '?'.

GLOW ON/OFF

("g" key.) Turn glow mode on or off.

GLOWSOLDER ON/OFF

Turn glowing-solder mode on or off. This controls whether solder dots are visible while in GLOW mode. It is on by default; turning glowing-solder off may improve simulation performance in GLOW mode.

GRID

(Shift-"G" key.) Change the cursor to a cross-hair shape. If the cursor is already a cross-hair, change it back to the regular arrow cursor.

GROUP n "Name"

Set the name that appears on the top of page number "n" of the Library screen. The number "n" must be from 0 to 8. If the "Name" argument is omitted, this command instead selects which page number the Library screen will display next time it is entered.

HELP

("?" key.) Display this file using the Chipmunk Help System.

HOME

("h" key.) Reset the scroll and zoom amounts to the default position on the page.

IDENTIFY

Used by the LOG-to-NTK conversion tool.

INVISIBLE ON/OFF

("i" key.) Turn "Invisible" mode on or off. In this mode, wires and most gates are invisible. However, labels, boxes, and switch/indicator gates are visible. Also, operations that edit the circuit are disallowed. Thus, a user can switch into Invisible mode and then operate a simulated "front panel" without confusion or fear of changing the circuit.

INVLABEL ON/OFF

(Shift-"I" key.) Turn "Invisible Labels" mode on or off. In this mode, labels and boxes are invisible.

LABEL message

("l" key.) Create a new label on the page. If a "message" argument is supplied, a label with the specified text is created on the lower-left corner of the screen. If no argument is supplied, LOG interactively enters the text for the label.

LIBRARY

("l" key or LIBR item in the Catalog screen.) Switch directly to the Gate Library screen.

LOAD filename

(Shift-"L" key.) Load the specified circuit file onto the current page, replacing anything that was on that page before. If no file name is specified, LOG displays a list of files in the current directory and asks the user to select one, or type the name. If the file name is '*', LOG simply asks the user to type the name. The shift-L key is bound to "LOAD *".

LOGNTK

Start the LOG-to-NTK conversion tool.

LOGNTK: parameter value

Set up a configuration parameter for the LOGNTK tool. Commands of this form usually appear in the file "/lib/log/logntk.cnf".

MACRO key-name command

Associate the specified command with a key. The key-name may be a single letter or other character (lower-case letters represent unshifted keys; upper-case letters represent shifted keys); or one of the words "bs", "tab", "sp" (space bar), or "cr" (Return or Enter); or a control key of the form "ˆ A" or "ˆ ["; or a three-digit decimal ASCII code. The "command" may be any of the commands listed here, and may include arguments. If you give only a key-name, the computer displays the command that is currently bound to the key. MACRO with no arguments displays a list of all current key bindings.

MAKE gate-name

Bring a gate in from the catalog by name, and put it in the drawing area. A shorthand for using the mouse in the the CAT and LIBR screens. If the gate-name contains wildcard characters, multiple gates may be loaded into the Catalog but only one of them will be created on the circuit page.

MARKER ON/OFF

("m" key.) Turn printing markers on or off. The printing markers are used mainly by the "PLOT" command; they indicate the area of the circuit page which is to be fitted into the printed page. The printing markers appear on the corners of the visible screen, and you can move them by regular editing operations to bracket any rectangular area of the page.

MESSAGE message

Display the specified message in the "message area" of the screen. The message area runs down the left edge of the screen; messages are drawn in yellow overlaying the drawing area.

MIRX

Select "mirror-in-X" gate-tapping mode. When a gate is tapped in this mode, it is flipped across its vertical axis. This works both in the menu area and in the drawing area, although some gates (such as switches) have special behaviors when tapped in the drawing area.

MIRY

Select "mirror-in-Y" gate-tapping mode.

MOVE

("m" key.) Move an area of the diagram. This is similar to deleting the area with DEL, then pasting it back in a new location with PASTE.

NAME file-name

See and/or change the "current file name" for the circuit page. This is the file name that will be used by default in later "SAVE" commands.

OFF

Turn simulation OFF.

ON

Turn simulation ON.

OPENH

Enter open-horizontal-area mode. Sweep out a rectangle; everything to the right of the left hand edge of the rectangle is moved to the right by the amount of the width of the rectangle. Wires straddling the left edge are stretched appropriately. BEWARE: This command may corrupt the LOG data structures if used in such a way as to break or make electrical connections in the circuit. Unless you really know what you are doing, use the cut-and-paste method of moving things.

OPENV

Enter open-vertical-area mode, similar to OPENH. DANGEROUS!

PAGE n

Switch the display to circuit page "n", from 1 to 9. "n" may also be "+" or "-", to switch to the next or previous numbered page.

PASTE

Enter Paste mode.

PLOT

Run the circuit plotting tool.

POPUP menu position kind command message

Change one of the locations in the pop-up menus. "Menu" is a number that selects which menu (1=Frills, 2=Editing, 3=Cursor, 4=Misc); "position" is an integer from 1 to 8 that identifies which spot in the menu:

[1 3 5 7]
[2 4 6 8]

"Kind" is normally zero; "command" is the command (from this list) that is to be executed if the menu is selected, enclosed in "quotes" if it contains spaces; and "message" is the wording for that menu element. If "message" begins with '*' or '#', it will be colored red or light blue, otherwise it will be green.

PROBE

(“.” key.) Change to the Probe cursor. If already using the Probe cursor, change back to an arrow. In Probe mode, holding the cursor near a wire or gate pin displays the value on that electrical node, in a simulator-dependent way. Holding the cursor near a gate in the drawing or menu areas displays the name of the gate on the screen. Tapping a gate or node opens it for configuring, if it has anything to configure. (Neither AnaLOG nor Digital LOG nodes are configurable, but many gates are.) Tapping and configuring a gate in the menu area sets the defaults for newly created gates pulled out of that menu slot. Finally, tapping out in the drawing area draws a yardstick. See also the EXAMINE command (on the "e" and "x" keys) which selects a temporary Probe mode.

QUIET

("q" key.) Turn sound prompting on or off.

READ filename

Read in a circuit from a file. This is similar to the LOAD command, but it rebuilds the circuit's connectivity information as it loads, rather than simply reading it from the file. As a result, READ is much slower than LOAD, but it is far more robust in case the file is corrupt or out of date. For example, if you run LOGED and add more pins to a gate, then try to LOAD a circuit diagram containing that gate, LOG may crash since the connectivity recorded in the file is out of date. But the READ command will read the file with no problems. In general, use the fast LOAD command unless you have a good reason not to.

REFRESH

(Space bar.) Refresh the screen.

RESET

(Shift-"R" key.) Send a Reset signal to the simulator(s). This resets simulation time back to zero and resets all voltages to their initial values.

RESPONSE min max rate

Set response-time parameters. With no arguments, displays current parameters. Initial values are 2, 50, and 35, respectively. During editing, LOG tries to check the mouse and keyboard at least every "min" centi-seconds. If the user stops editing, this rate slowly increases until the maximum of "max" centi-seconds. The rate of increase is one unit per "rate" centi-seconds of idle time.

ROT

Select "Rotate by 90 degrees" gate-tapping mode. When a gate is tapped in this mode, it rotates 90 degrees counter-clockwise. This works both in the menu area and in the drawing area, although some gates (such as switches) have special behaviors when tapped in the drawing area.

SAVE filename

Save circuit page(s). If a name is given, saves the current page in the file "filename.LGF." If the name is "*", prompts the user to enter the name to save. If no name is given, re-saves all circuit pages which have been modified since last saved or loaded. Shift-"S" is bound to "SAVE *".

SCOPE

("s" key.) Switches to the Scope screen.

SHELL shell-command (or: !shell-command)

(Also, "!" as a single-key command.) Start a sub-Shell. If a shell-command is provided the shell executes this one command and returns. If no shell-command is provided, you get an interactive sub-shell. Type quit or press control-D to return from the sub-shell.

SIM (or ONOFF or POWER)

Turn simulation ON or OFF (toggling).

SNAP

("$" key.) Turn "snap-to-grid" mode on or off. Default is off. In this mode, the cursor arrow always jumps to the nearest spot on the LOG editing grid rather than moving smoothly.

STATUS

Enter the "Status" display. Use the knob or arrow keys to switch among various pages. A page-name may also be given with the STATUS command:

LOG	General system status
MEMORY	Memory usage status
MACRO	Keyboard mappings
32	Analog simulator
LOGNTK	LOG-to-NTK tool

STEP n

Turn simulation OFF, if necessary, then step the simulation by one time-step. If an integer "n" is provided, executes "n" time-steps before stopping.

TAPMODE

(menu selection, or shift-"M" key.) Select the next gate-tapping mode, in order: ROT, MIRX, MIRY, CNFG.

TRACE

Turn the debugging "wallpaper" mode on or off. Please don't do this unless a LOG maintainer is there. Trace output goes to the "Trace" file, which by default is an appropriate printer.

TRACEFILE filename

Close any current trace file, then select "filename.text" as the new trace file. If this file already exists, new text will be appended to it.

VERBOSE

("v" key.) Turn informational messages on or off. These messages are mostly of the "Glow mode is now OFF" variety.

VMESSAGE

Similar to MESSAGE, but works only if in Verbose mode.

YARDSTICK

Draw a "yardstick", useful for measuring the spacing of objects in your diagram.

ZOOMDN

Zoom down, to view more of the circuit page.

ZOOMUP

Zoom up, to view part of the circuit in greater detail.

5.2 Configuration Files

1. LOG reads the file "log.cnf" every time it starts up. If you do not have one in the current directory, it uses "/lib/log/log.cnf". You can give the "-c" command-line switch to specify your own configuration file. The "use log" script actually loads the program "diglog" into the computer and aliases the "log" command to "diglog -cdlog", causing "dlog.cnf" (in the current directory or in /lib/log) to be used. Similarly, "use analog" loads the program "analog" and aliases "log" to "analog -calog".
2. To create your own configuration file, the best approach is to make a file named "log.cnf", "alog.cnf", or "dlog.cnf", whichever is appropriate, in your current directory. In that file, put a line something like "include /lib/log/alog.cnf", followed by any additional commands to override the system defaults. Do not copy the system CNF files if you can avoid it; these files are subject to change at any time.
3. For example, your CNF file might look like this:

   { A comment saying what this is used for }
   include /lib/log/dlog.cnf { Inherit from Digital LOG }
   gates + .../log/mygates { Add on my own gates file }
   menu a b c d e vdd gnd { Choose my own menu gates }
   glow on { Turn on "glow" mode }

   In this example, the "+" in the GATES command means to add the name(s) to the existing list of gates files. Since the MENU command does not have a "+", it replaces the list from the other file. This notation works with the GATES, GET, GETGROUP, and MENU commands in configuration files.
4. The following ":" commands may also be used in a "log.cnf" configuration file:

AVOID	GLOWSOLDER	OFF	QUIET
COLOR	GROUP	ON	RESPONSE
CONFLICT	INVISIBLE	SIM	SNAP
DOTS	INVLABEL	POPUP	VERBOSE
GLOW	MACRO	PROBE	

5. The following additional commands are allowed in configuration files:

COMMAND toolname command command command ...

Register that the listed commands are handled by the indicated tool. If a command is given and its owning tool has not been initialized, that tool will be initialized.

DO command-line

Add the command-line to a list of ":" commands to be done as soon as LOG is started. These commands are executed as if typed from the keyboard, i.e., they may be commands which are ordinarily forbidden in CNF files. The "DO" commands are executed in the order they originally appeared in the CNF files.

GATES filename filename filename ...

Register the specified ".gate" files into the LOG gate library. Files are searched for a given gate starting at the first filename of the first GATES command in the CNF file. Ordinarily, the presence of the first GATES command in a given CNF file overrides all GATES commands in previously read CNF files (i.e., INCLUDE files). If the first word after GATES is "+", the listed filenames will instead be added to the existing list of gates files. If no explicit directory is given, LOG looks first in the current directory, then in the "home" directory, then in /lib/log. If the gates file is not found, it is ignored.

GET gate gate gate ...

Load the specified gates into the gate catalog. The gate names may include wildcard characters "*" and "?", and may also specify one or more group numbers from 0 to 8: "GET A*;34" gets all gates whose names begin with A, which are found on pages 3 or 4 of the Library. If this is the first GET command in its CNF file, it overrides any previous GET commands from other files unless GET is followed by the word "+". (The obsolete word LOAD is accepted as a synonym for GET.)

GETGROUP gate gate gate ...

Load the specified group of gates into the gate catalog. This like a normal GET command, except the gates are guaranteed to be placed together (provided there's room), and a box is drawn around them on the catalog screen.

HELP filename

Name of a ".help" file to be used when the user asks for help.

HOME directory-name

Specify a directory to be used as the "home" directory. If a CNF file, gates file, help file, or news file is not found in the current directory, LOG looks next in this "home" directory, then finally in "/lib/log". The default "home" directory is " /log", that is, the "log" subdirectory of the user's home directory.

INCLUDE cnf-file

Read commands from the specified ".cnf" file, then resume reading from this file. Include files may be nested.

MENU gate gate gate ...

Load the specified gates into the menu area. If there are more gate names than menu slots, only the first few names are used. The first word may be "+" to add the specified gates to a list from a previous CNF file.

NEWS file-name

Name of a ".text" file whose contents are displayed when LOG begins.

SIGNALS number

(or NODES.) Specifies the maximum number of TO/FROM signal names that can exist at one time during the LOG session. Default is 200.

TABLET number

Set the HP-IB address of the graphics tablet to be used. Default is 0, which means look for the tablet on the HP-HIL or at the standard address.

TOOL tool-name "full-name"

Register a tool name so that the run-time TOOL command will display its name. (Ordinarily, tools are automatically registered as soon as they are needed.)

UNDO

Delete all "DO" commands registered up to this point.

tool-name: tool-command

This directs a command to a particular LOG "tool". The standard tool names are:

16	Digital simulator
32	Analog simulator
LOGNTK	LOG-to-NTK tool

5.3 AnaLOG Simulator Commands

1. The following additional ":" commands are available with the AnaLOG simulator.

DIM ON/OFF

Turn dim-unconnected-gates mode on or off.

ERASE

Same as touching Erase on the "Scoreboard" gate. If you are selecting Set and Erase often, you might want to use the MACRO command to put SET and ERASE on a pair of keys, for convenience.

EXACT

Same as touching Exact on the "Scoreboard" gate.

NEWMODEL

Set non-saving device-model mode, in which modeling parameters for transistors, and simulation constants such as "Vdd", are reset to their default values rather than being loaded from each circuit file.

OLDMODEL

Set the default device-model mode, in which modeling parameters for transistors, and simulation constants such as "Vdd", are saved in circuit files along with the transistor and "Scoreboard" (NUMBERS) gates.

PROBEPREC digits

Set the number of decimal digits for voltages displayed by the "probe" cursor. Default is 3.

RELAXED

Same as touching Relaxed on the "Scoreboard" gate.

SET

Same as touching Erase on the "Scoreboard" gate.

2. Also, a great many AnaLOG parameters may be set in ".CNF" files. These commands all begin with the prefix "32:". Defaults appear in the file "models.cnf" in the analog /lib directory – ask your local maintainer for details

5.4 Single-Key Commands

Many commands also appear in single-key forms. In this list, lower-case letters represent unshifted keys, and capital letters are shifted keys.

:	Enter a full-line command.
space	Refresh the screen: REFRESH.
knob	(or arrow keys.) Scroll across the page.
<ctrl>-c	Abort the current command or mode.
1-9	(Digits.) Switch to a new page. Same as PAGE 1 to PAGE 9.
!	Shell escape: SHELL.
*	Draw the contents of the Paste buffer: PASTE.
,	(Comma.) Switch to alternate viewing position: ALTPOSN.
.	(Period.) Enter or leave PROBE mode.
/	Copy an object or area into the Paste buffer: COPY.
<	Zoom down, to see more of the diagram at once: ZOOMDN.
>	Zoom up, to focus on a smaller area of the diagram: ZOOMUP.
?	Display this help file.
+	(also NEXT on Bobcats.) Switch to next higher page number.
-	(also PREV on Bobcats.) Switch to next lower page number.
b	Draw a dashed box: BOX.
c	Switch to "CNFG" gate-tapping mode: CNFG.
C	Enter the Catalog screen: CAT.
d	Delete objects: DEL.
D	Show simulator's definition for a gate: DEFINE.

e	Enter temporary PROBE mode: EXAMINE.
f	Turn on FAST mode.
g	Turn GLOW mode on or off.
G	Turn GRID (cross-hair cursor) mode on or off.
h	Return to home (unscrolled) position: HOME.
i	Turn INVISIBLE mode on or off.
I	Turn INVLABEL (invisible labels) mode on or off.
l	Draw a label: LABEL.
L	Load a new circuit page: LOAD.
m	Move an object or area: MOVE.
M	Select the next gate-tapping mode: TAPMODE.
$	Turn SNAP-to-grid mode on or off.
o	Turn simulation on or off: ONOFF.
p	Enter the circuit-plotting tool: PLOT.
r	Switch to "ROT" gate-tapping mode: ROT.
R or 0	Reset the simulators: RESET.
q	Turn QUIET mode on or off.
Q or Z	(or Control-D.) Exit from the LOG system: EXIT.
s	Enter the Scope screen: SCOPE.
S	Save the current page to a file: SAVE.
t	Run simulation for one time-step, then stop: STEP.
T	Enter the Tool screen: TOOL.
v	Turn VERBOSE mode on or off.
x	Enter temporary PROBE mode: EXAMINE.
y	Draw a YARDSTICK.

5.5 Color Names

1. The following list is the complete set of color-names defined so far in the LOG system. See the COLOR command for details on how to reconfigure these color names.

2. After each color-name is its default color, then a description of what screen objects that color-name governs.

BACKGR	Gray	Screen background color.
BASELINE	Cyan	Color of line above menu area.
BLUEWORD	Cyan	Light blue text in pop-up menus.
CATALOGBOX	Dim yellow	Gate grouping box in Catalog screen.
CATGATE	Cyan	Color of gates in Catalog screen.
CHART	Yellow	Default color of traces in Scope.
CURSOR	White	Regular cursor color.
DASHBOX	Dim yellow	Color of dashed boxes in circuits.
DEFINEBACK	Black	DEFINE command's box background.
DEFINEBOX	Cyan	DEFINE command's box outline.
DEFINETEXT	Green	DEFINE command's text color.

DIMGATE	Dim cyan	Color of "dimmed" gates in circuits.
DIVISION	White	Color of grid dots in Scope.
GATE	Cyan	Normal color of gates in circuits.
GATEBLACK	Black	Black parts of gate pictures.
GATEGREEN	Green	Green parts of gate pictures.
GATEORANGE	Orange	Orange parts of gate pictures.
GATEPIN	Red	Color of gate connection dots.
GATERED	Red	Red parts of gate pictures.
GATEWHITE	White	White parts of gate pictures.
GATEYELLOW	Yellow	Yellow parts of gate pictures.
GLOW1	Black	AnaLOG glow color for <= 0 volts.
GLOW2	Dim red	AnaLOG glow color for 0-15
GLOW3	Medium red	AnaLOG glow color for 15-50
GLOW4	Red	AnaLOG glow color for 50-85
GLOW5	Pink	AnaLOG glow color for 85-100
GLOW6	White	AnaLOG glow color for >= Vdd.
IDENTIFY	Red	Color of identified things in LOGNTK.
IMETER	Orange	AnaLOG current meter readout.
INSTANCE	Dim yellow	Name shown in Digital instance gates.
KINDGATE	Green	Color of gates in menu area.
LABELTEXT	Dim yellow	Color of circuit labels.
LIMITON	Red	AnaLOG current-limit indicator.
MARKER	Pink	Color of printing markers.
MENUWORD	Green	Color of text in menu area.
MESSAGE	Yellow	Informative messages in drawing area.
OFFLED	Black	Digital LOG logic "0" signal.
ONLED	Red	Digital LOG logic "1" signal.
PAGE1	Green	Color of the word "PAGE".
PAGE2	Yellow	Color of current page number.
PAGE3	Green	Color of the word "OF".
PAGE4	Yellow	Color of maximum page number.
PEN1	White	Color for plotter pens #1-6.
PEN2	Red	. These colors are used by the PLOT
PEN3	Green	. command for circuit and data
PEN4	Orange	. plotting.
PEN5	Cyan	.
PEN6	Yellow	.
POPUPBOX	White	Outline of pop-up menus.
POPUPWORD	Green	Text in pop-up menus.
POPUPSEL	Pink	Highlighted text in pop-up menus.
PROBE	Dim yellow	AnaLOG probe-mode display color.
REDWORD	Red	Red text in pop-up menus.
SCROLL	White	Color of scroll-indication lines.
SELECT	White	Color of area-selection rectangle.
SELWORD	Yellow	Highlighted words in menu area.
SIGNAL	Pink	Color of TO/FROM signal names.

SOLDER	Green	Color of solder dots on wires.
SWITCHOFF	Light gray	AnaLOG switch “off” color.
SWITCHON	Light gray	AnaLOG switch “on” color.
TEMPLATE	Pink	Name shown in Digital template gates.
TIME	Dim yellow	Color of TIME gate display
VMETER	Dim yellow	AnaLOG voltage meter readout.
WIRE	Green	Color of non-glowing wires.
XWIRE	X-Green	Color of wires being drawn.

5.6 Command-Line Options

These are all the command-line options log currently understands:

`-x displayname`: This option specifies the screen where the newcrt and mylib windows will appear. For example, the command analog -x foo.school.edu:0.0 results in the newcrt and mylib windows appearing on the console of the machine foo.school.edu. An alternative to setting the DISPLAY environment variable in the launch shell.

`-c filename`: Specifies a .cnf configuration file.

5.7 X Display Preferences

AnaLOG looks in your .Xresources file for geometry information. If set properly, this allows windows of a certain size to be automatically placed at a particular place at every invocation of AnaLOG; manual placement of windows is no longer necessary. This feature was added by Jim Clark at Harvard. Adding these lines to your .Xresources file:

```
mylib.geometry:  =1150x550+0+0
```

```
newcrt.geometry:  +510+586
```

will result in good placement and sizing for the color monitors shipped with SPARC IPC's; for other screens, change the numbers after “+” signs for new positions, and change the numbers between the “x” to modify the size of the mylib window. The newcrt window cannot be modified; AnaLOG expects it to be a fixed size.

AnaLOG now senses if a display screen is black and white, and tries to use stipple instead of colors. However, there are times where you want to force the tools to display in black and white, even if the display screen is color. These times include using an OS/2 based color PC as an X terminal for the analog tools, because of present incompatabilities between PC graphics cards and analog. To force a black and white display, add this line to your .Xresources file:

```
mylib.color:  black_and_white
```

and then restart your X server. Black and white support was provided by Tor Sverre Lande (Oslo), and the .Xresources flag was added by Mike Godfrey (Stanford).