SIMLAB User's Guide

M. Silveira, A. Lumsdaine, J. White

RLE Technical Report No. 588

December 1994

The Research Laboratory of Electronics
MASSACHUSETTS INSTITUTE OF TECHNOLOGY
CAMBRIDGE, MASSACHUSETTS 02139-4307

This work was supported in part by the National Science Foundation under Grant MIP 91-17724.

SIMLAB^{TM} USER'S GUIDE

M. Silveira, A. Lumsdaine, J. White

Research Laboratory of Electronics
Department of Electrical Engineering and Computer Science
Massachusetts Institute of Technology
Cambridge, MA 02139

Draft of December 12, 1994

Abstract

This manual describes how to use the SIMLAB circuit simulator for verifying the electrical behavior of circuits.

SIMLAB is a circuit simulation environment consisting of a flexible, user-friendly front-end operating in conjunction with a sophisticated and versatile simulation engine. The program is specifically designed to be used as an educational tool and as a research platform. SIMLAB can be operated in either batch or interactive mode. The user is allowed to separately specify algorithms for the various aspects of the simulation, including the simulation environment, ODE system solution type, numerical integration method, nonlinear algebraic and linear system solution methods.

Furthermore, SIMLAB is also designed to be easily customized for research purposes. Researchers can easily construct and test algorithms by inserting them into the existing SIMLAB framework. For information on how to add algorithms to SIMLAB, see *The* SIMLAB *Programmer's Guide.*

Contents

1 Introduction

SIMLAB is a circuit simulation environment consisting of a flexible, user-friendly front-end operating in conjunction with a sophisticated and versatile simulation engine. The program is specifically designed to be used as an educational tool and as a research platform. SIMLAB can be operated in either batch or interactive mode.

The user is allowed to separately specify algorithms for the various aspects of the simulation. These include:

- Simulation environment (e.g. serial or parallel depending on the underlying hardware).
- ODE system solution (e.g. point)
- ODE system time integration (e.g. backward-Euler, trapezoidal, second-order Gear),
- Nonlinear algebraic system solution (e.g. multidimensional Newton's method, nonlinear relaxation),
- Linear system solution (e.g. sparse Gaussian elimination, Gauss-Jacobi relaxation, conjugate gradient, conjugate gradient squared),

Furthermore, SIMLAB has a notion of simulation mode and different methods can be specified for different modes. At present, supported modes are DC for the calculation of operating points, and Transient for the calculation of the time response of a circuit. For instance, assuming that the user has specified the multidimensional Newton's method for solving the nonlinear system of equations, the linear solver associated could be different depending of what type of simulation is being performed.

The user can also interactively tune the following simulation parameters:

- `stop` – final simulation time,
- `cmin` – minimum capacitance to ground at each node,
- `gmin` – minimum conductance to ground at each node,
- `nrvabs`, `nrvrel` – Newton-Raphson absolute and relative voltage convergence criteria,
- `nrcabs`, `nrcrel` – Newton-Raphson absolute and relative current convergence criteria,
- `lterel`, `lteabs` – absolute and relative local truncation error,
- `nralpha` – Newton-Raphson ΔV step limit,

- `newjacob` – Number of Newton iterations before updating the Jacobian,
- `maxtrnr`, `maxdcnr` – Maximum number of *transient* and *dc* Newton-Raphson iterations allowed,
- `dodc`, `dotran` – flags indicating *dc* or *transient* solution is desired,
- `verbose` – flag indicating verbose mode,
- `simdebug` – flag indicating debug mode.

Simulation statistics are also reported for all phases of the simulation process.

In the bibliography, references to standard textbooks and papers on circuit simulation and related topics are given.

In its basic form, SIMLAB is a powerful circuit simulator, but it is also designed to be easily customized for research purposes. For example, SIMLAB forms the core of special-purpose simulation programs, such as a switched capacitor filter simulator and a simulator for vision circuits. The program code is highly modular, so that researchers can easily construct and test algorithms by inserting them into the existing SIMLAB framework. For information on how to add algorithms to SIMLAB see *The* SIMLAB *Programmer's Guide.*

The SIMLAB program was developed at MIT under support from Analog Devices, the NSF PYI program, and DARPA contract N00014-87-K-825. Miguel Silveira was partially supported by the Portuguese INVOTAN committee and Andrew Lumsdaine was also supported by an AEA/Dynatech faculty development fellowship. Installing SIMLAB in your computer system is done with the aid of **makelinks**, a tool for avoiding code duplication developed at MIT by G. Adams, D. Siegel and S. Narasimhan. The program uses a circuit parser written by P. Moore and a sparse matrix solver based on work by K. Kundert, both of whom were students in computer-aided design at U.C. Berkeley. The front-end interpreter for the program was inspired by the **xlisp** program written by David Michael Betz and uses the readline library from the Free Software Foundation's **bash** program. The plotting package included with the SIMLAB distribution is **xgraph**, a two-dimensional plotting package developed by D. Harrison at U.C. Berkeley. Section 3 of this user's manual was derived in part from R. Saleh's and A.R. Newton's SPLICE User's Manual, available from U.C. Berkeley.

Question or problems related to the installation or usage of the SIMLAB circuit simulator should be addressed to any one of the authors by U.S. mail, or by electronic mail to `simlab@rle-vlsi.mit.edu` (18.62.0.214). Any bugs should be reported by electronic mail to `simlab-bug@rle-vlsi.mit.edu` so that they can be solved for later releases. Furthermore, the authors welcome any comments, suggestions, or enhancements that will come from other users experience with the program.

2 A SIMLAB Primer

SIMLAB is invoked from the command line as follows:

```
simlab [-c circuit_file [-r]] [-f diary_file] [-qvV] [config_file]
```

SIMLAB can be used in interactive mode, in batch mode, or both.

The command line arguments are:

`-c circuit` specifies a circuit file to be read initially. See the `circuit` command in Section A

`-f diary_file` specifies a diary file to be used initially. See the `diary` command in Section A

`-r` specifies that a run is to be executed using the known data. This option requires that a circuit file be specified and tells simlab to use the default methods to simulate the given circuit.

`-q` turns off interactive mode. When this command line argument is given, SIMLAB will exit after executing the command contained in the specified configuration file. If no configuration file is given this line argument is ignored.

`-v` turns on verbose mode, in which case SIMLAB emits certain diagnostic messages. See the description of the `verbose` environment variable in Section B.

`-V` turns on debug mode, in which case SIMLAB emits many diagnostic messages. See the description of the `simdebug` environment variable in Section B.

`config_file` is an initial configuration file read by SIMLAB . If a circuit file is also specified, the configuration file is read *after* the circuit file. SIMLAB will read and execute each line of the configuration file just as if the commands were interactively entered. If a `quit` command is not given in the configuration file, SIMLAB will enter interactive mode after reading it (and executing any commands therein) unless the command line argument `-q` is given, in which case SIMLAB exits after executing the command contained in the configuration file.

In general, an interactive simulation session will proceed as follows:

1. Start SIMLAB (perhaps with a circuit file, configuration file, or both);
2. Configure the simulation environment (interactively, with a configuration file, or both);

3. Run simulation;
4. Plot simulation results;
5. If not done, change circuit parameters and go to 2

When SIMLAB is first started it attempts to read a startup file. SIMLAB first looks for the file specified by the environment variable SIMLAB_RC. If this variable is not set, SIMLAB tries to read, in order, "./.simlabrc" and " /.simlabrc".

2.1 The SIMLAB Front-End

The primary interface to SIMLAB is through an interpretive front-end. The front-end reads commands and expressions (either interactively or from a configuration file) and evaluates them. See Appendix A for a description of the available SIMLAB commands.

SIMLAB will also evaluate expressions. For example, typing in:

```
=> a = 5 * pi
```

will produce the result

```
a = 15.708
```

Notice that the variable a has been created and now exists in the SIMLAB environment. If you now type

```
=> a
```

SIMLAB will return the value of a:

```
a = 15.708
```

SIMLAB has predefined variables (which correspond to certain simulation parameters) that the user can set via the front-end. These variables include:

- stop – final simulation time,
- cmin – minimum capacitance to ground at each node,
- gmin – minimum conductance to ground at each node,
- nrvabs, nrvrel – Newton-Raphson absolute and relative voltage convergence criteria,
- nrcabs, nrcrel – Newton-Raphson absolute and relative current convergence criteria,

- `lterel`, `lteabs` – absolute and relative local truncation error,
- `nralpha` – Newton-Raphson ΔV step limit,
- `newjacob` – Number of Newton iterations before updating the Jacobian,
- `maxtrnr`, `maxdcnr` – Maximum number of *transient* and *dc* Newton-Raphson iterations allowed,
- `dodc`, `dotran` – flags indicating *dc* or *transient* solution is desired,
- `verbose` – flag indicating verbose mode,
- `simdebug` – flag indicating debug mode.

See Appendix B for a complete description of these variables.

The values of these variables can be examined and changed in the obvious way. For instance, if you type

```
=> cmin
```

SIMLAB will return with the value of `cmin`, e.g.,

```
cmin = 1.e-15
```

To change the value of `cmin` to 1.0e-12, type in

```
=> cmin = 1.0e-12
```

SIMLAB will then reply

```
cmin = 1.e-12
```

The value of `cmin` used during simulation will now be 1.0e-12. Note that the predefined system variables can be used in algebraic expressions. For example, an input line like

```
=> cmin = stop / 1.0e-6
```

is perfectly acceptable. See Appendix B for a complete description of SIMLAB 's variables and the rules that guide their usage and the changes in their values.

2.1.1 Command-Line Editing

One handy feature of SIMLAB in interactive mode is its command-line editor, which is implemented with the `readline` library of the Free Software Foundation's `bash` program.

The default key-bindings are essentially patterned after `gnu-emacs` (or `tcsh`) and allow the user to recall and edit previous lines.

Table 1 shows the most primary key-bindings for the SIMLAB command-line editor. For more information about `readline`, refer to the `bash` documentation from the Free Software Foundation.

The command-line editor saves each line that is entered into a history list. The history list can be accessed via the C-n and C-p keys (down and up history, respectively). The entire history list can be displayed with the `history` command. The run-time length of the history list is controlled by the SIMLAB variable `histlength`. Upon exiting, SIMLAB saves the history list on a file. The size of the saved history list is controlled by the SIMLAB variable `savehist` and cannot exceed the value of `histlength`. By default upon starting, SIMLAB looks for the environment variable `SIMLAB_HISTFILE` for the location of the history file. If this variable is not set, SIMLAB will use the default location which is defined at compile time (normally "`./.simlab_hist`"; see `config.h` if you desire to change this value).

2.1.2 Plotting

Currently, viewing the results of a simulation run is done using a modified version of `xgraph`, a two-dimensional plotting package developed by D. Harrison at U.C. Berkeley. However, if some local viewing program is available, the user can override the use of `xgraph` by setting the environment variable `SIMLAB_PLOTTOOLS` to point to the preferred plotting tool. The environment variable `SIMLAB_PLOTARGS` should also be set, possibly with command line arguments for the plotting tool. If no arguments are necessary, the variable should still be set (this can be acomplished from inside SIMLAB). Note that the plotting tools must be able to read the simlab output format which consists of lines of the form:

⟨ `node_name` ⟩ ⟨ `time` ⟩ ⟨ `voltage` ⟩

2.2 Specifying algorithms in SIMLAB

One of the most powerful features of SIMLAB (when used for algorithm development)is the ability to specify which numerical algorithms are used at different stages of the simulation. For this purpose, there are five basic predefined *functional controls*:

solve_linear: Specifies function to be used for linear system solution during simulation.

solve_nonlinear: Specifies function to be used for nonlinear system solution during simulation.

Key	Action
C-a	beginning-of-line
C-b	backward-char
C-d	delete-char
C-e	end-of-line
C-f	forward-char
C-g	keyboard-quit
TAB	complete word
C-k	kill-line
C-l	refresh
C-n	down-history
C-p	up-history
C-q	quoted-insert
C-r	isearch-backward
C-s	isearch-forward
C-t	transpose-chars
C-u	universal-argument
C-w	kill-region
C-y	yank
SPC .. ~	self-insert-command
ESC 0 .. ESC 9	digit-argument
ESC b	backward-word
ESC c	capitalize-word
ESC d	kill-word
ESC f	forward-word
ESC g	goto-line
ESC l	downcase-word
ESC t	transpose-words
ESC u	upcase-word
ESC y	yank-pop
ESC ?	list completions
ESC DEL	backward-kill-word

Table 1: SIMLAB Key Bindings.

integrate: Specifies which integration method to use.

simulate: Specifies what type of simulation structure to use.

environment: Specifies the environment in which SIMLAB is being used. This will normally be `serial` but functionality to take advantage of parallel computer architectures can be added.

Furthermore with the notion of simulation mode, some of the above functional controls can be given different meanings depending on the current mode of simulation. Section B contains a detailed description of these functional controls. The functional controls are modified with the `set` command, and displayed with the `show` command (see Appendix A).

For example, if you are testing out a new linear system solver called "baz" (and have properly compiled it into SIMLAB — see the *SIMLAB Programmer's Guide*), you would set the transient linear system solver to `baz` with

```
=> set trans solve_linear baz
```

Or, if `baz` was a more generic function, you could set it to be the linear solver for all modes of simulation (namely DC and transient) with

```
=> set solve_linear baz
```

You can examine the settings of the functional controls with `show`. Typing

```
=> show solve_linear
```

will return the current settings for `solve_linear` for all known modes of simulation. Just typing

```
=> show
```

will return the current settings for all functional controls and simulation modes.

If a functional control is set to a non-existent function, SIMLAB will output a warning and dismiss the change. However if a change is made to a functional control such that the new value is incompatible with the current values of other functional controls, SIMLAB will output an error message at run-time. For instance, suppose that a new simulation method, called `simfoo` is compiled into SIMLAB and is such that `simfoo` uses different data-structures and therefore requires an integration method called `simfooint`. If the user merely specifies

```
=> set simulate simfoo
```

and then request a run

```
=> run
```

SIMLAB will try to use the default integration algorithm and will output an error saying that that algorithm is not a valid setting for the current simulation control.

2.3 Warnings about the Algorithms used in the SIMLAB Program

One limitation of some popular simulation methods is that they sometimes do not converge unless there is a capacitor to ground at every node in the circuit. This is a mild assumption for most realistic circuits, although users of circuit simulators often don't include these capacitances. Therefore the SIMLAB program will insert a minimum capacitance to ground at every node in the circuit. This value of the minimum capacitance is set by the parameter `cmin` and is a user-alterable, but should not be set to zero.

Finally, the SIMLAB program was developed at MIT as a research project. It is not intended to be a production system, although serious effort has been made to insure the correctness of the program (it works for at least one test case). Reporting problems to the authors about SIMLAB would be appreciated, provided it is done gently.

2.4 Hints on Convergence

Obtaining the DC solution can be a difficult problem, although SIMLAB seems to do a good job of it. However, there are still examples which will not work. If you have trouble, try playing with the options `gmin` and `nralpha`. `gmin` is a conductance placed to ground at each node, so it should be kept to a reasonable (small) value or your DC solution will not be accurate. `nralpha` controls the maximum step a voltage can take in any one iteration of the DC solution.

SIMLAB does a very good job of converging during the transient solution. In fact, nonconvergence during the transient solution usually means that there is an error in the circuit or a bug in the program. If you have problems during the transient solution, it may help to set the option `newjacob` equal to 1, although this may slow the program down somewhat. Other options include increasing the value of `cmin`, reducing the absolute and relative convergence criteria or increasing the maximum number of iterations on iterative solvers.

3 The Circuit File

Although the user interacts with SIMLAB in batch or interactive mode, circuits are read from circuit description files (specified with the `circuit` command). A circuit description file contains element models, circuit elements, subcircuit definitions, and selected simulation parameters. Example circuit files can be found in Appendix F.

3.1 Fundamentals

3.1.1 Models and Elements

The following elements can be used to construct circuits for SIMLAB: linear resistors, linear capacitors, diodes, voltage-controlled current sources, bipolar and MOS transistors. The BJT transistor model in SIMLAB is an adaption of the integral charge control model of Gummel and Pool with extensions that include several effects of high bias levels. For a more detailed description see [8]. The MOS transistors in SIMLAB are modeled by the Schichmann-Hodges equations [7] (SPICE2 level=1) or by those of the semi-empirical model described in [8] (SPICE2 level=3). SIMLAB also supports the following types of independent sources: constant current sources, constant grounded voltage sources, piecewise-linear time-dependent grounded voltage sources, and sinusoidal time-dependent grounded voltage sources. Floating voltage sources are not yet supported.

The `model` statement is used to describe general parameters for the SIMLAB elements, and many elements of the same type may refer to a single `model` statement. Each model definition must include an external reference name, the name of a primitive model type, and the appropriate model parameters. For example:

model *BusMos* **nmos** *vto=0.8v kp=20u, gamma=0.8 phi=0.6*

defines the model for an enhancement NMOS transistor with 0.8V threshold.

The circuit topology is specified using element statements. Each SIMLAB element statement must include a name for the element, a list of node names, and a reference to a model. So a *BusMos* transistor named *dWrite2* whose substrate is grounded, is clocked by signal *PHI2*, and connects nodes *latchOut* to *DBus* would be defined by:

dWrite2 latchOut PHI2 DBus 0 BusMos

3.1.2 Subcircuits and Parameters

The SIMLAB circuit parser also supports hierarchical circuit descriptions through the use of subcircuits. A subcircuit is like a program macro in the sense that the subcircuit is first defined, and then each time it is referred to, the definition replaces the reference.

The general form for a subcircuit definition is the keyword `define` followed by a subcircuit name, and then a list of nodes enclosed in parentheses. The element statements that describe the subcircuit follow the define statement, and the subcircuit is terminated with an end statement. For example, an inverter subcircuit could be described as follows:

```
define inverter (input, output)
  m1  output  input  0  0  pulldn w=10u l=5u
  m2 vdd  output  output  0  pullup w=5u l=10u
end inverter
```

and then a circuit that is a string of inverters could be described by

```
inverter1  in1  out1  inverter
inverter2  out1  out2  inverter
inverter3  out2  out3  inverter
```

Node names are not the only parameters that can be passed into subcircuits. It is also possible to pass number parameters to subcircuits. For example, if one wanted to scale the width of the above inverter, keeping the same width/length ratios, the inverter subcircuit could be defined as follows:

```
define inverter (input, output)
parameters w=10u
  m1 output input  0  0  pulldn w=w l=0.5* w
  m2 vdd  output  output  0  pullup w=w l=2.0*w
end inverter
```

and then a circuit that is a string of inverters could be described by

```
inverter1  in1  out1  inverter
inverter2  out1  out2  invertsr w=20u
inverter3  out2  out3  inverter w=400u
```

where inverter1 would be assigned the default value of 10 microns for the width.

3.1.3 Locals and Globals

It is possible to have local nodes in a subcircuit. In fact, if any node name that is not contained in the node list of the define statement for the subcircuit is used in the subcircuit, it will automatically be local to the subcircuit. This can be overridden by explicitly declaring certain nodes to be global using the `global` statement. This is particularly useful for supply voltage node names. Note, however, that the first node specified on the global list is always the ground or reference node.

3.1.4 Control Statements

Information such as the simulation period, the simulation accuracy criteria, and analysis requests, are specified with a single options statement as follows:

simlab **options** *stop=100ns dodc=1 dotran=1*

Default values are used whenever parameter values are not set explicitly by the user. The value of `stop` should always be set by the user, as it sets the simulation interval.

In order to request that the output of certain nodes in the circuit are recorded, the plot statement is used, and its format is just the keyword `plot` followed by a list of node names. A warning will be issued if the input file does not specify any nodes for plotting.

3.1.5 Initial Conditions and the DC Solution

The SIMLAB program allows the user two ways of specifying the initial conditions used to begin a transient analysis. The default is that all the node voltages begin at zero. This can be thought of as a "power-up" phase. However, the implication of this strategy is that the circuit should not be clocked until it has settled after powering up. If the circuit is clocked too early, the simulation will still be performed without error, but the circuit may not behave as expected.

If desired, it is possible to force a node to be initialized to any voltage by using the `ic` statement. Its format is the keyword `ic` followed by a list of node name, initial value pairs with the two in each pair connected by an equal sign. For example,

```
ic input=5.0
```

forces the initial condition for node *input* to be 5 volts.

Finally, the user can request that SIMLAB perform an explicit dc solution, as available in SPICE2, and begin the transient analysis with that solution. The nodes that have been specified by an `ic` statement are forced to their respective values for the dc solution.

3.1.6 Comments

Any line in the circuit description file that begins with a semicolon (;) is assumed to be a comment line and is ignored.

3.2 Basic Language Rules

Field separators Delimiters may be one of more blanks, or a comma. These delimiters must be placed where required by the syntax of the circuit description language.

Continuation Character A statement may be continued by ending it with a backslash (\) and continuing on the next line.

Name Field A name field can have any number letters or numbers in it and must not contain any delimiters (only alphanumeric characters). In all cases, the name must start with a letter. For example, *alu*, or *fadder7* are acceptable names, but *7alu* is not.

Lower and Upper Case All keywords (**ic, plot, define, model, end, parameters, global**) must be in lower case. Otherwise, no distinction is made between upper and lower case letters; the names 'signalOne' and 'SignalOne' are considered to be identical by SIMLAB. Case may be used for ease of comprehension, however, provided it is used with care!

Number representations A number field may be replaced by an integer (i.e. 12, -44), a floating point number (3.14159), or an integer or floating point number followed by an integer exponent (1E-14, 2.65e3).

Scale factors An integer or a floating point number can be scaled if followed by one of these scale factors:

g=1E9	meg=1E6	k=1E3	m=1E-3
u=1E-6	n=1E-9	p=1E-12	f=1E-15

Letters immediately following a number that are not scale factors are ignored, and letters immediately following a scale factor are ignored. Hence, 10, 10v, 10volts, and 10Hz all represent the same number, and m, ma, msec, and mmhos all represent the same scale factor. Note that 1000, 1000.0, 1000Hz, 1E3, 1.0E3, 1kHz, and 1k all represent the same number.

3.3 Choosing Node Names

Node names A node name can have any number of letters or numbers in it and must not contain any delimiters (only alphanumeric characters). If the node name is not all numbers, it must start with a letter. For example, *a7*, or *l7* are acceptable names, but *7a* is not.

Ground Node and Global nodes Global names can be used whenever you want to define a node name that is to be used in the main circuit as well as subcircuits. They are generally used for supply voltage names. Any number of global nodes can be specified using the global statement. Its format is the keyword **global**, followed by a list of node names. The global statement also performs another function, that of indicating to the SIMLAB program which node to use as ground. The first node in the global node list is assumed by SIMLAB to be the ground node, and should be so specified. There is no other constraint on the order of the nodes in the list.

3.4 General Form for Model and Element Statements

Notation In the following description, data fields that are enclosed in brackets, '[' and ']' are optional. Names enclosed in '⟨' and '⟩' should be replaced by alphanumeric strings and value fields should be replaced with numbers. All indicated punctuation is required. Associated reference directions are used (positive current flows in the direction of most positive to most negative).

3.4.1 Model Statements

Model Statement

General Form:
model ⟨*name*⟩ ⟨*type*⟩ [[⟨*name1*⟩ = ⟨*value*⟩] [⟨*name2*⟩ = ⟨*value*⟩]...]

Examples:
model nmos1 **nmos** vto=2v kp=15u gamma=0
model cstray **c** c=0.025pf

This statement describes a model in the circuit. Element statements refer to this statement to obtain parameter values associated with the element.

The model statement must begin with the word **model**. The next field is the model name, which is used to refer to the model in the rest of the circuit description. The third field is the model type. This should be the name of an internal SIMLAB model (r, c, nmos, pmos, nmos3, pmos3, d, etc).

If there are parameters, they should be listed at the end of the model statement. All unlisted parameters will be set to the default value for the model type (See next section). To specify a parameter, include the parameter name and the value, separated by an equal sign (=). The model parameters list can be in any order.

3.4.2 Element Statements

Element Statements

General Form:

⟨ *name* ⟩ ⟨ *node*1 ⟩ ... ⟨ *nodeN* ⟩ ⟨ *model or subcircuit name* ⟩ ⟨ *params* ⟩

Examples:

nmos1 store input phase 1 0 mostran w=25u 1=1.5u
gcap1 input output **c** c=0.07p
gate7 output outbar inverter

The element definition statement consists of an element name, the nodes connected to the element, a model to which the element refers, and optional parameters. The element names can be the same as the model name (or even as a node name), but no two elements may have the same element name. The number and order of the nodes must correspond to the model being referenced. The model name can be either a user defined model, a subcircuit, or a SIMLAB low-level model (c, r, dc, pwl, i, etc.). An element statement can also include parameters, and will override the values given in the **model** statement. MOS transistor elements can include the width (w) and length (l) parameters, diodes can include the area factor, and resistor and capacitor elements can include resistance and capacitance values. As with the **model** statement, parameters should be listed as a name/value pair, separated by "=".

3.4.3 Subcircuit Definitions

The subcircuit defines a macro which can be expanded in the circuit any number of times. When an element statement refers to a subcircuit definition, the elements defined within the subcircuit are inserted into the circuit. The node names specified in the element statement are substituted for the node names in the subcircuit definition statement. This substitution is by order, the first node in the element statement is substituted for the first node in the subcircuit definition, and so on. Note that the node names in the subcircuit definition must be enclosed in parentheses, and that it is an error for the element statement to have a different number of nodes than the subcircuit definition it refers to.

Any nodes used within the subcircuit which are not global (contained on the global node list), and are not listed in the subcircuit definition statement, are local to that subcircuit (i.e. these names are not known outside the subcircuit definition). Since a global node cannot be replaced during a subcircuit expansion, it is an error for a global name to appear as a node on a subcircuit definition statement.

The subcircuit can optionally include a parameters statement. To specify subcircuit parameters, the keyword **parameters** is followed by a list of parameter names and default values, separated by equals (=). A named parameter can be used as if it were a number in the subcircuit description, and in addition, can be multiplied by a floating point number scale factor. An element statement which refers to the subcircuit can set the values of any or all of the names parameters. Any parameters that are not set on the element statement will be given its default value.

The most recently begun subcircuit is terminated by the **end** statement. An optional name may be specified after the **end** statement to indicate which subcircuit definition is being completed. This name is not used by the program but is useful to include for readability.

Internal subcircuit nodes may be referenced for plotting or for assigning initial conditions, by using a name derived from its subcircuit node name and the subcircuit instance name separated by a dot ".". For example, "adder1.i1" refers to internal node "i1" in a subcircuit instance called "adder1".

Subcircuit Definitions

General Form:

define ⟨ *subcircuit name* ⟩ (⟨ *node* 1 ⟩ [. . . ⟨ *node N* ⟩])
[**parameters** ⟨ *param*1 ⟩ = ⟨ *default* ⟩ [. . . ⟨ *paramN* ⟩ = ⟨ *default* ⟩]]
. . .
⟨ *element statements* ⟩
end [⟨ *subcircuit name* ⟩]

Example:

define invert (output input)
parameters w=5u
m1 vdd output output 0 depload w=w 1=1.0* w
m2 output input 0 0 pulldn w=5.0* w l=w
end invert

3.5 Electrical Elements

3.5.1 MOS Models

There are two MOSFET models that can be used to simulate circuits with the SIMLAB program. There is a first order model based on the Schichmann-Hodges equations[4] for the dc drain current, and a more complicated one whose equations are the same as for SPICE2 MOS Level 3. An MOS model may be either N-channel or P-channel, enhancement or depletion.

First Order MOS Model and Element Definitions

First Order MOS Model and Element Definitions

MOS Model Definition:

model ⟨ *model name* ⟩ ⟨ *model type* ⟩ [⟨ *parameter* – *list* ⟩]

MOS Element Definitions:

⟨ *name* ⟩ ⟨ *drain* ⟩ ⟨ *gate* ⟩ ⟨ *source* ⟩ ⟨ *bulk* ⟩ ⟨ *model name* ⟩ ⟨ *params* ⟩

Examples:

model pullup **nmos** vto=-5.2 gamma=0.914 kp=69u phi=0.6 lambda=0.02
model pulldn **pmos** vto=1.12 gamma=0.266 kp=79u phi=0.6 lambda=0.02
load1 vdd output output 0 pullup w=5u 1=20u
pulldn 1 output outbar 0 0 pulldn w=10u 1=5u

The model parameters used in SIMLAB Schichmann-Hodges MOSFET model are as given in Table 2. Any parameter not specified will be set to its default value. The element parameters are listed in Table 3.

MOS Level 3 Model and Element Definitions

The MOS level 3 model is based on the model available in the SPICE2 program [6], although numerous modifications have been made to enhance the speed and insure the accuracy of the model. This model is suitable for small geometry MOSFETs and it accounts for second-order effects such as threshold voltage sensitivity to the length and width of the device and lowered saturation voltage and current due to velocity saturation. The details of this model may be found in [8]

MOS Level 3 Model and Element Definitions

MOS Model Definition:

model $\langle$ *model name* $\rangle$ $\langle$ *model type* $\rangle$ [$\langle$ *parameter – list* $\rangle$]

MOS Element Definitions:

$\langle$ *name* $\rangle$ $\langle$ *drain* $\rangle$ $\langle$ *gate* $\rangle$ $\langle$ *source* $\rangle$ $\langle$ *bulk* $\rangle$ $\langle$ *model name* $\rangle$ $\langle$ *params* $\rangle$

Examples:

model pullup **nmos3** vto=-5.2 gamma=0.914 kp=69u phi=0.6 lambda=0.02
model pulldn **pmos3** vto=1.12 gamma=0.266 kp=79u phi=0.6 lambda=0.02
load1 vdd output output 0 pullup w=5u 1=20u
pulldn 1 output outbar 0 0 pulldn w=10u 1=5u

The model parameters used in MOS Level 3 are listed in table 4. The element parmeters for the MOS 3 model are listed in table 6.

3.5.2 Bipolar Models

There is a BJT model that can be used to simulate circuits with bipolar transistors with the SIMLAB program. This model is an adaption of the integral charge control model of Gummel and Pool with extensions that include several effects of high bias levels.

name	parameter	units	default	typical
w	default width for elements	m	1	1μ
l	default length for elements	m	1	2μ
as	default source area for elements	m^2	0	0
ad	default drain area for elements	m^2	0	0
ps	default source perimeter for elements	m	0	0
pd	default drain perimeter for elements	m	0	0
gamma	bulk threshold parameter	$V^{1/2}$	0	0.37
phi	surface potential	V	0.6	0.65
lambda	channel-length modulation	V^{-1}	0	20m
vto	zero-bias threshold voltage	V	0.0	1
kp	transconductance parameter	AV^{-2}	20μ	10μ
tox	gate oxide Thickness	m	0	0.1μ
cgso	gate-source overlap capacitance per unit channel width	Fm^{-1}	0	40p
cgdo	gate-drain overlap capacitance per unit channel width	Fm^{-1}	0	40p
cgbo	gate-bulk overlap capacitance per unit channel width	Fm^{-1}	0	200p
fc	coefficient for forward-bias depletion capacitance formula		0.5	0.5
js	bulk junction saturation current per sq-meter of junction area	Am^{-2}	0	10n
mj	bulk junction bottom grading coef.		0.5	0.5
mjsw	bulk junction sidewall grading coef.		0.5	0.33
cj	zero bias bulk junction bottom cap. per sq-meter of junction area	Fm^{-2}	0	0.2m
cjsw	zero bias bulk junction sidewall cap. per meter of junction perimeter		0	1n
pb	bulk junction potential	V	0.8	0.87
ld	lateral diffusion	m	0	0.8μ
is	bulk junction saturation current	A	0	1f
cbd	zero-bias B-D junc. cap	F	0	20f
cbs	zero-bias B-S junc. cap	F	0	20f

Table 2: Schichmann-Hodges MOSFET model parameters.

name	parameter	units	examples
w	channel width	m	10μ
l	channel length	m	1.5μ
as	area of source region	m^2	100p
ad	area of drain region	m^2	100p
ps	perimeter of source region	m	40μ
pd	perimeter of drain region	m	40μ

Table 3: Schichmann-Hodges MOSFET element parameters.

Bipolar Transistor Models

BJT Model Definition:
model ⟨ *model name* ⟩ ⟨ *model type* ⟩ [⟨ *parameter – list* ⟩]

BJT Element Definitions:
⟨ *name* ⟩ ⟨ *collector* ⟩ ⟨ *base* ⟩ ⟨ *emitter* ⟩ ⟨ *substrate* ⟩ ⟨ *model name* ⟩ ⟨ *params* ⟩

Examples:
model nbjt **npn** bf=200 vaf=250 tr=10n
model pbjt **pnp**
load1 vdd 3 5 0 nbjt
pulldn 1 output outbar 0 0 pbjt bf=100

The model parameters used in SIMLAB BJT model are as given in Table 7. Any parameter not specified will be set to its default value. The element parameters are listed in Table 9.

3.5.3 Diodes

Diodes

Model Definition:
model ⟨ *name* ⟩ **d** [⟨ *parameters* ⟩]

Element Definition:
⟨ *element name* ⟩ ⟨ *anode* ⟩ ⟨ *cathode* ⟩ ⟨ *model name* ⟩ [*area* = ⟨ *value* ⟩]

Example:
model pnj **d** cj0=8e-12
d1 drain pnj area=2p
d2 0 source pnj area=2p

name	**parameter**	**units**	**defaults**	**examples**
w	default width for elements	m	1	1μ
l	default length for elements	m	1	2μ
as	default source area for elements	m^2	0	0
ad	default drain area for elements	m^2	0	0
ass	area of the fet source sidewall	m^2	0	0
asd	area of the fet drain	m^2	0	0
nrs	number of squares (source res. calc.)		0	0
nrd	number of squares (source res. calc.)		0	0
vto	zero-bias threshold voltage	V	0	1.0
kp	transconductance parameter	AV^2	0	10μ
gamma	bulk threshold parameter	$V^{1/2}$	0	0.37
phi	surface potential	V	0	0.65
cgso	G-S overlap cap/m of width	Fm^{-1}	0	40p
cgdo	G-D overlap cap/m of width	Fm^{-1}	0	40p
cgbo	G-B overlap cap/m of length	Fm^{-1}	0	0.2n
tox	oxide thickness	m	10μ	10μ
nsub	substrate doping	cm^{-3}	0	4.0e15
nfs	fast surface state density	cm^{-2}	0	10G
xj	metallurgical junction depth	m	0	1μ
ld	lateral diffusion	m	0	0.8μ
uo	surface mobility	$cm^2(Vs)^{-1}$	600	700
vmax	maximum drift velocity	ms^{-1}	0	50K
delta	width effect on thres-hold voltage		0	1
theta	mobility modulation		0	0.1
eta	static feedback		0	1
kappa	staturation field factor		0.2	0.5
fc	coefficient for forward bias delp. cap. formula		0.2	1.2
js	bulk junc. sat. current per m^2 of area	Am^{-2}	0	1f
mj	bulk junction bottom grad. coefficient	Fm^{-2}	0.5	0.2m
mjsw	bulk junction sidewall grad. coefficient	Fm^{-2}	0.5	0.2m

Table 4: Level 3 MOSFET model parameters.

name	parameter	units	defaults	examples
cj	zero-bias junc. bottom capacitance/m^2 of area	Fm^{-2}	0	0.2m
cjsw	zero-bias junc. sidewall capacitance/m^2 of area	Fm^{-2}	0	0.2m
pb	bulk junc. potential	V	0.8	0.87
rs	source resistance (per square)	Ω	0	0
rd	drain resistance (per square)	Ω	0	0
nss	surface state density	cm^{-2}	0.1m	10G
is	bulk junc. sat. current	A	10f	1f

Table 5: Level 3 MOSFET model parameters (cont.).

name	parameter	units	examples
w	channel width	m	10μ
l	channel length	m	1.5μ
as	area of source region	m^2	100p
ad	area of drain region	m^2	100p
ps	perimeter of source region	m	40μ
pd	perimeter of drain region	m	40μ
nrs	number of squares (source res. calc.)	0	10
nrd	number of squares (source res. calc.)	0	10

Table 6: Level 3 MOSFET element parameters.

name	parameter	units	default	typical
is	transport saturation current	A	$10f$	$1f$
bf	ideal maximum forward beta		100	200
nf	forward current emission coefficient		1	1
vaf	forward Early voltage	V	∞	200
ikf	corner for forward beta high current roll-off	A	∞	0.01
ise	B-E leakage saturation current	A	0	0.1n
ne	B-E leakage emission coefficient		1.5	2
br	ideal maximum reverse beta		1	0.1
nr	reverse current emission coefficient		1	1
var	reverse Early voltage	V	∞	200
ikr	corner for reverse beta high current roll-off	A	∞	0.01
isc	B-C leakage saturation current	A	0	0.1n
nc	B-C leakage emission coefficient		2	1.5
rb	zero bias base resistance	Ω	0	100
irb	current where base resistance falls halfway to its min value	A	∞	0.1
rbm	minimum base resistance at high currents	Ω	**rb**	10
re	emitter resistance	Ω	0	1
rc	collector resistance	Ω	0	10
cje	B-E zero-bias depletion capacitance	F	0	2p
vje	B-E built-in potential	V	0.75	0.6
mje	B-E junction exponential factor		0.33	0.33
tf	ideal forward transit time	s	0	0.1n
xtf	coefficient for bias dependence of **tf**	s	0	0
vtf	voltage describing **vbc** dependence on **tf**	V	∞	
itf	high-current parameter for effect on **tf**	A	0	
ptf	excess phase at $f = \frac{1}{2\pi \mathbf{tf}}$Hz	deg	0	
cjc	B-C zero-bias depletion capacitance	F	0	2p
vjc	B-C built-in potential	V	0.75	0.6
mjc	B-C junction exponential factor		0.33	0.5
xcjc	fraction of B-C depletion capacitance connected to internal base node		1	

Table 7: Bipolar model parameters.

name	parameter	units	default	typical
tr	ideal reverse transit time	s	0	10n
cjs	zero-bias collector-substrate capacitance	F	0	2p
vjs	substrate junction built-in potential	V	0.75	0.6
mjs	substrate junction exponential factor		0	0.5
xtb	forward and reverse beta temperature exponent		0	
eg	energy gap for temperature effect on **is**	eV	1.11	
xti	temperature exponent for effect on **is**		3	
kf	flicker-noise coefficient		0	
af	flicker-noise exponent		1	
fc	coefficient for forward-bias depletion capacitance formula		0.5	
area	area factor		1	

Table 8: Bipolar model parameters (cont.).

name	parameter	units	examples
area	area factor	m^2	1

Table 9: Bipolar Element parameters.

name	**parameters**	**units**	**default**	**typical**
area	diode area	m^2	1	1μ
is	saturation current	A	10f	1.5p
imax	Forward explotion current	A	10	10
n	emission coefficient		1	1
tt	transit-time	s	0	0.1n
vt	thermal voltage	V	2.58n	2.58n
cjo	zero-bias junc. cap.	F	0	2p
phi	junction potential	V	0.6	0.6
m	grading coeff.		0.5	0.5
fc	coeff. of depletion cap. in forward bias		0.5	0.5
bv	rev. breakdown	V	∞	40

Table 10: Diode model parameters.

Diodes may be used anywhere, and are typically used to represent source and drain junction effects on MOS devices. In the element definition, *area* is the area of the junction. Model parameters are given in table 10. The parameter *imax* is used to bound the diode current to avoid numerical overflow. It should be set to a value that well above a reasonable current for the diode.

3.5.4 Resistors

Resistors

Model Definition:
model ⟨ *name* ⟩ **r** [*r* = ⟨ *value* ⟩]

Element Definition:
⟨ *element name* ⟩ ⟨ *node*1 ⟩ ⟨ *node*2 ⟩ ⟨ *model name* ⟩ *or* **r** [*r* = ⟨ *value* ⟩]

Example:
model rmod **r** r=15k
r1 input gate rmod
r2 1 2 **r** r=10k

⟨ *Node*1 ⟩ and ⟨ *node*2 ⟩ are the nodes to which the resistor is connected. ⟨ *Value* ⟩ is the resistance in ohms, and must not be set to zero.

3.5.5 Capacitors

Capacitors

Model Definition:
model ⟨ *name* ⟩ **c** [*c* = ⟨ *value* ⟩]

Element Definition:
⟨ *element name* ⟩ ⟨ *node*1 ⟩ ⟨ *node*2 ⟩ ⟨ *model name* ⟩ *or* **c** [*c* = ⟨ *value* ⟩]

Examples:
model cmod **c** c=5uf
c1 input gate cmod
c2 1 2 c c=15P

⟨ *Node*1 ⟩ and ⟨ *node*2 ⟩ are the nodes to which the capacitor is connected. ⟨ *Value* ⟩ is the capacitance in farads.

3.5.6 Constant Current Source

Constant Current Source

Element Definition:
⟨ *element name* ⟩ ⟨ *node*1 ⟩ ⟨ *node*2 ⟩ **i** *i* = ⟨ *value* ⟩

Example:
bias1 1 2 i i=20ua

The current ⟨ *value* ⟩ (in amperes) flows from ⟨ *node*1 ⟩ to ⟨ *node*2 ⟩.

3.5.7 Voltage Controlled Current Source

Voltage Controlled Current Source

Element Definition:
⟨ *element name* ⟩ ⟨ *node*1 ⟩ ⟨ *node*2 ⟩ ⟨ *node*3 ⟩ ⟨ *node*4 ⟩ **vccs** *gm* = ⟨ *value* ⟩

Example:
vccs1 1 2 3 4 **vccs** gm=3

The current ⟨ *value* ⟩ $\times$ ($v3 - v4$) flows from ⟨ *node*1 ⟩ to ⟨ *node*2 ⟩.

3.5.8 Constant Voltage Sources

Constant Voltage Sources

Element Definition:
⟨ *element name* ⟩ ⟨ *node*1 ⟩ ⟨ *node*2 ⟩ **dc** *v* = ⟨ *value* ⟩

Example:
vcc 1 0 dc v=5.0v

⟨ *Node*1 ⟩ is the positive node of the voltage source, and ⟨ *node*2 ⟩ must be the ground node. SIMLAB does not yet support floating voltage sources.

3.5.9 Piecewise-linear Voltage Sources

Piecewise-linear Voltage Sources

Element Definition:
⟨ *name* ⟩ ⟨ *node*1 ⟩ ⟨ *node*2 ⟩ **pwl** [*delay* = ⟨ *value* ⟩ ⟨ *period* ⟩ = ⟨ *value* ⟩]\
*t*0 = ⟨ *value* ⟩ *v*0 = ⟨ *value* ⟩ [... *t*24 = ⟨ *value* ⟩ *v*24 = ⟨ *value* ⟩]

Example:
vin 1 0 **pwl** delay=0ns period=100ns \
t0=0ns v0=5.0v t1=10ns v1=5.0v t2=15ns v2=0.0v t3=95ns v3=0ns

⟨ *Node*1 ⟩ is the positive node of the voltage source, and ⟨ *node*2 ⟩ must be the ground node. SIMLAB does not yet support floating voltage sources. There are three types of parameters for the **pwl**. There can be up to 20 (0 through 19) breakpoints in the piecewise linear waveform, which are pairs (ti=sec vi=volts). There is also a **delay** parameter, which causes each breakpoint is delayed by the amount specified. Finally, there is the **period** parameter, which, when specified, causes the waveform to repeat after the given number of seconds.

3.5.10 Sinusoidal Voltage Sources

Sinusoidal Voltage Sources

Element Definition:
$\langle$ *element name* $\rangle$ $\langle$ *node*1 $\rangle$ $\langle$ *node*2 $\rangle$ **sine** \
[*phase* = $\langle$ *value* $\rangle$ *freq* = $\langle$ *value* $\rangle$ *ampl* = $\langle$ *value* $\rangle$ *offset* = $\langle$ *value* $\rangle$]

Example:
vin 1 0 **sin** ampl=1v freq=1k

$\langle$ *Node*1 $\rangle$ is the positive node of the voltage source, and $\langle$ *node*2 $\rangle$ must be the ground node. SIMLAB does not yet support floating voltage sources.

3.6 Analysis and Control Statements

3.6.1 Plot Statement

The only way to specify desired output from a SIMLAB simulation is with the `plot` statement, and therefore every circuit description file should have this statement. All the nodes to be plotted must be listed after the `plot` keyword, and this keyword should appear only *once* in a circuit description file. There is no limit to the number of plotted nodes, but it is an error to try to plot an unused node. In the third example above, *latch1* and *latch2* are two subcircuit instances with the internal node store.

Plot Statement

General Form:
plot $\langle$ *node*1 $\rangle$ [$\langle$ *node*2 $\rangle$ [... $\langle$ *node N* $\rangle$]]

Examples:
plot clock j k q qbar
plot phase1 phase2 latch1.store latch2.store

For each node name following the **plot** keyword SIMLAB will produce an entry in the output file with the node name, time, and voltage. The file will have the same name as the circuit file, except with a ".trans" extension. The plot file contains a collection of time-voltage pairs. This format was chosen because it allows the end-user to write a post-processor to fit a given terminal with the minimum of effort (in fact, the `plot` command available from the front-end spawns a post-processor to read the plot file). Note that if accuracy is critical, the points should be interpolated with a 2-order Lagrange polynomial. However, linearly interpolating the points is, in most cases, good enough and produces reasonably good plots.

3.6.2 Initial Conditions

The **ic** statement forces the specified node to the given value for the calculation of the DC solution, after which, the node is allowed to take on whatever value is determined by the circuit. The DC solution uses zero for an initial guess for all nodes other than those specified by an **ic** statement or set by a voltage source.

Initial Conditions

General Form:
ic ⟨ *node1* ⟩ = ⟨ *value1* ⟩ ⟨ *node2* ⟩ = ⟨ *value2* ⟩ ... ⟨ *node N* ⟩ = ⟨ *value N* ⟩

Example:
ic a=5.0 b=0.2 c=0.3 d=5.0

3.6.3 Simulation Options

The options statement is used to specify the analyses to be performed, and the balues of simulation parameters to be used. These options, their defaults and an explanation of each is given in Table 11. With the exception of **dodc**, **dotran**, and **stop**, these parameters should only be adjusted by an informed user. The defaults should work well for most cases.

When an option is set from within a circuit file, that value of the option will only apply for that simulation and until another circuit is read in. When another circuit is read in, all simulation options are first reset to their default values. The new circuit may then possibly modify some of the options, but those modifications would only apply to that particular circuit. The default values of these options can be modified from the front-end. See Section B.2 for a more detailed explanation of the behavior of the simulation options.

Simulation Options

General Form:
simlab options [⟨ *param1* ⟩ = ⟨ *value1* ⟩ ... ⟨ *param N* ⟩ = ⟨ *value N* ⟩]

Examples:
simlab options stop=100ns dodc=0
simlab options stop=100ns cmin=1e-16

name	default	definition
dodc	1	If equal to 1, performs dc solution. If zero, no dc solution is performed.
dotran	1	If equal to 1, performs transient analysis. If zero, no transient analysis is performed.
cmin	1e-18F	Sets the minimum capacitance to ground. Only used in the dc solution. Should not be set to zero.
gmin	1n *mho*	Sets the minimum conductance to ground. Only used in the dc solution. Should not be set to zero.
nrvabs	1mV	sets the Newton-Raphson absolute voltage tolerance.
nrvrel	0.001	Sets the Newton-Raphson relative voltage tolerance.
nrcabs	1e-9	Sets the Newton-Raphson absolute current tolerance.
nrcrel	0.001	Sets the Newton-Raphson relative current tolerance.
maxdcnr	200	Sets the number of newton iterations used before giving up in the dc solution.
maxnr	10	Sets the number of newton iterations used before giving up in the transient solution.
nralpha	0.3V	Sets the maximum voltage step to use in newton iteration.
newjacob	3	Number of newton iterations used before reevaluating the jacobian in the transient analysis.
lteabs	5m	Sets the absolute local truncation error.
lterel	10m	Sets the relative local truncation error.

Table 11: SIMLAB program simulation options and default values.

A SIMLAB Commands

---

cd　　　　　　　　　　　　　　　　　　　　cd

Purpose:
Change current working directory

Synopsis:
`cd` [pathname]

Description:
`cd` changes the current working directory, i.e., the starting point for pathnames not beginning with '/'. Without an argument, `cd` changes to the user's home directory. The `cd` command understands the `csh` style of ~ shorthand for pathnames.

---

circuit　　　　　　　　　　　　　　　　　　circuit

Purpose:
Load a circuit file for simulation

Synopsis:
`circuit` filename

Description:
`circuit` loads a circuit into SIMLAB for simulation. "filename" must be a valid file name. Some application variables will be affected by the circuit file.

Diagnostics:
The input circuit file must be specified in SIMLAB format. See Section 3 for more information.

Purpose:

Continue simulation run.

Synopsis:

`continue`

Description:

`continue` allows the user to continue a simulation run from the present time until `stop`. This is useful when a run is made and the stop time increased, or when a run is interrupted. Transient simulations started with `continue` begin at time t = `PresentTime` and end at time t = `stop`.

Diagnostics:

If the present time equals the stop time, `continue` does nothing. If `continue` is issued without a previous `run`, a run is executed.

diary diary

Purpose:
Record the session in a file.

Synopsis:
`diary` [`on` / `off`] [filename]

Description:
The **`diary`** command saves the entire SIMLAB session (in ascii format) in file filename. If no "filename" argument is specified, the session is saved (in ascii format) in the file "diary.doc." The argument **`off`** will temporarily suspend diary operation until **`diary`** `on` is issued.

The **`diary`** command will also save output generated by programs run with escapes to the shell.

Diagnostics:
Diary uses `tee` to capture output produced by programs run in an inferior shell. Some programs (in particular, `ls`) have a different format when directing output to a pipe instead of to the standard output.

help help

Purpose:
Provide help.

Synopsis:
`help` [item]

Description:
`help` provides help. When invoked without an argument, `help` provides a list of possible help topics. When invoked with an argument, `help` provides information on that specific topic.

Diagnostics:
At present, help can only list possible help topics. On-line help will be enhanced in future releases.

history history

Purpose:
List previous command lines.

Synopsis:
`history`

Description:
`history` lists the lines entered into SIMLAB for review by the user. Previous lines can be accessed with successive applications of the C-p key (see Section 2.1.1).

plot

Purpose:
Plot the circuit output.

Synopsis:
`plot`

Description:
`plot` causes all the nodes specified in the circuit input file (using the circuit description file `plot` directive, which is distinct from the SIMLAB `plot` command) to be plotted. See Sections 2.1.2 and 3.6.1.

Diagnostics:
SIMLAB effects this command by forking and calling an appropriate plot program for its host environment. Presently, only the X-windows version 11 environment is supported. At this time, all the nodes specified by the `plot` line in the circuit file will be plotted — individual nodes cannot be selected from that list. This should change in a future release.

printenv

Purpose:
Print the value of an environment variable.

Synopsis:
`printenv` [env_var]

Description:
`printenv` prints out the values of the variables in the UNIX environment of the SIMLAB process. If a variable is specified, only its value is printed.

quit

Purpose:
Leave SIMLAB

Synopsis:
`quit`

Description:
The quit command terminates the current SIMLAB session. Any open diary file is closed.

run

Purpose:
Perform a simulation.

Synopsis:
`run`

Description:
The **run** command begins a simulation run. If the `dodc` variable is set TRUE, a DC solution is initially attempted. If the `dotran` variable is set, a transient simulation is attempted. Transient simulations started with **run** begin at time $t = 0$ and end at time $t =$ `stop`. The user can stop a simulation in progress by typing ctrl-c.

set

Purpose:
Set simulation parameter.

Synopsis:
`set` [mode] param val

Description:
The `set` command assigns value "val" to parameter "param" in mode "mode". If no mode is specified the parameter is assigned value "val" irrespective of the simulation mode. "val" must be a valid value for "param". See Section B for a description of available parameters and values.

setenv

Purpose:
Set environment variable.

Synopsis:
`setenv` [env_var [val]]

Description:
With no arguments, the `setenv` command displays all UNIX environment variables. With the `env_val` argument it sets the environment variable `env_var` to have an empty (null) value. (By convention, environment variables are normally given upper-case names.) With both `env_var` and `val` arguments `setenv` sets the environment variable `env_var` to the value `val`, which must be either a single word or a quoted string. When SIMLAB is started all the environment variables currently set are automatically imported to the SIMLAB UNIX environment. Setting a new variable or reseting the value of an existing variable affectes the UNIX environment of SIMLAB but not that of its father process.

Purpose:
Execute shell command

Synopsis:
sh [command]

Description:
sh executes its arguments using an inferior shell. If nor arguments are given, sh will start up the shell specified by the SHELL environment variable, or /bin/sh if the SHELL environment variable is not set.

For example:

```
=> sh ls
```

will execute the ls command in the current working directory.

Diagnostics:
The inferior shell when sh is invoked with arguments is sh not csh, so certain csh features will not work (in particular the ˜ for pathnames).

## show	show

Purpose:
Show SIMLAB parameters and values.

Synopsis:
`show` [[mode]] param]

Description:
The show command displays SIMLAB parameters and values. With an argument, show displays the specified parameter and values for the various modes. If a mode is also specified along with a parameter then only the value of that parameter for the mode in question is shown. Without an argument, show displays all the parameters and their values for the various modes. See Section B for a description of available parameters and values.

## source	source

Purpose:
Load and execute a SIMLAB configuration file.

Synopsis:
`source` filename

Description:
The `source` command causes SIMLAB to read and execute a SIMLAB configuration file.

Diagnostics:
SIMLAB attempts to interpret everything in the specified file. Funny things can happen if it is given a file which is not a SIMLAB script.

unsetenv unsetenv

Purpose:
Unset environment variable.

Synopsis:
`unsetenv` env_var

Description:
`unsetenv` removes the variable "env_var" from the UNIX environment.

who who

Purpose:
Display variables existing in the SIMLAB environment.

Synopsis:
`who`

Description:
The `who` command displays the variables existing in the SIMLAB environment. The variables are grouped into three categories, system constants, application variables, and user-defined variables. See Section B for a description of SIMLAB environment variables.

B SIMLAB Variables

B.1 System Constants

True
Description: Boolean value True. Can be used with boolean simulation variables, e.g., `dodc = True`. `True` is equivalent to integer value one.

False
Description: Boolean value False. Can be used with boolean simulation variables, e.g., `dodc = False`. `False` is equivalent to integer value zero.

pi
Description: 3.14159 . . . .

B.2 Simulation Variables

These variables are used for controlling various aspects of the simulation. You will most likely only need to change those variables which are necessary to specify the type of simulation desired, e.g., `stop`, `dodc`, or `dotrans`. In general, the default values of the other variables will work well. If convergence problems are encountered, you may wish to adjust some (see Sections 2.3 and 2.4).

In the current release of SIMLAB all variables can be changed by the user when interacting with the front-end. Some of them, however, can also be set from the circuit file. SIMLAB has the notion of default value of a variable and current value of a variable. Changes in the default or current values of SIMLAB's variables adhere to the following conventions:

1. all changes to values of SIMLAB's variables, affecting either their default or current values, are shown to the user;
2. before any circuit is read into SIMLAB, the current values of all SIMLAB variables are reset to their default values;

3. if the value of a SIMLAB variable is set at the command line that change affects both the default and current values of that variable; furthermore the new value becomes the new default value for the entire session;
4. if the value of a SIMLAB variable is set from the circuit file, that change affects only the current value, although it remains in effect until a new value is given at the command line or a new circuit is read.

This means that reading in circuits that explicitely contain commands to change the value of some of SIMLAB's variables will not affect subsequent runs of other circuit, since the variables are reset to their default values before a new circuit is read into SIMLAB. However setting the value of a variable at the command line changes the actual default value of the variable for the entire SIMLAB session.

time

Description:
The current value of time in the simulation run. `time` is set to zero when `run` is issued, and should be set to `stop` when a simulation run is completed normally.

cmin

Description:
Minimum value of capacitance to ground at each node. Should not be set to zero.

Default:
1.0e-18 F

dodc

Description:
Boolean variable specifying whether or not to conduct a dc simulation. If `dodc` and `dotran` are both set to `True`, the dc simulation is conducted first and the solution is used as the initial condition for the transient simulation.

Default:
`True`

dotran

Description:
Boolean variable specifying whether or not to conduct a transient simulation. If `dodc` and `dotran` are both set to `True`, the dc simulation is conducted first and the solution is used as the initial condition for the transient simulation.

Default:
`True`

gmin

Description:
Minimum condutance to ground at each node. Should not be set to zero.

Default:
1p mhos

histlength

Description:
Size of the history list to be kept by SIMLAB at run-time.

Default:
100

lteabs

Description:
Absolute local truncation error tolerance.

Default:
5m

lterel

Description:
Relative local truncation error tolerance.

Default:
10m

maxdcnr

Description:
Maximum number of newton iterations in the dc simulation to be attempted before indicating failure.

Default:
200

maxnr

Description:
Maximum number of newton iterations in the transient simulation to be attempted before indicating failure.

Default:
10

maxsteps

Description:
The simulation interval divided by maxsteps yields the smallest allowable timestep.

Default:
1.0e8

minsteps

Description:
Minimum allowable number of steps in simulation interval.

Default:
50

newjacob

Description:
Number of newton iterations used before re-evaluating Jacobian.

Default:
3

nralpha

Description:
Maximum allowable voltage step in Newton iteration update.

Default:
0.3

nrcabs

Description:
Absolute Newton current error tolerance.

Default:
1n

nrcrel

Description:
Relative Newton current error tolerance.

Default:
1m

nrvabs

Description:
Absolute Newton voltage error tolerance.

Default:
1m

nrvrel

Description:
Relative Newton voltage error tolerance.

Default:
1m

savehist

Description:
Size of the history list to be saved upon exiting SIMLAB.

Default:
100

simdebug

Description:
Flag indicating simdebug mode. This is a debugging mode and a large amount of diagnostic messages are printed. There should never be any reason to set `simdebug` to `True`.

Default:
`False`

stop

Description:
`stop` specifies the length of the transient simulation interval. Transient simulations started with `run` begin at time $t = 0$ and end at time $t =$ `stop`. Transient simulations started with `continue` begin at time $t =$ `PresentTime` and end at time $t =$ `stop`.

Default:
1 s

verbose

Description:
Flag indicating verbose mode. This is a debugging mode and a large amount of diagnostic messages are printed. There should never be any reason to set `verbose` to `True`.

Default:
`False`

B.3 Functional Controls

One of the most powerful features of SIMLAB is the ability to specify the algorithms to be used at different levels of the simulation for the various simulation modes. This feature is most useful for the individual who is developing and testing new algorithms. For most people using SIMLAB strictly as a circuit simulator, the default functions will be good enough.

In this section, the seven functional controls are described, along with valid values for them.

solve_linear

Description:
Specifies function to be used for linear system solution during simulation. Different functions can be specified for DC and Transient simulation.

Values:

`(dc)`	`direct`	(sparse Gaussian elimination)
`(tran)`	`direct`	(sparse Gaussian elimination)

Default:

`(dc)`	`direct`
`(tran)`	`direct`

solve_nonlinear

Description:
Specifies function to be used for nonlinear system solution during simulation. Different functions can be specified for DC and Transient simulation.

Values:

`(dc)`	`newton`	(Newton-Raphson iteration)
`(tran)`	`newton`	(Newton-Raphson iteration)

Default:

`(dc)`	`newton`
`(tran)`	`newton`

integrate

Description:
Specifies which integration method to use. Has no meaning except in transient mode.

Values:

`(tran)`	`trap`	(trapezoidal integration)
`(tran)`	`gear2`	(second-order backward-difference integration)
`(tran)`	`beuler`	(backward-Euler integration)
`(tran)`	`feuler`	(forward-Euler integration)

Default:

`(tran)`	`trap`

simulate

Description:
Specifies what type of simulation structure to use. Different functions can be specified for DC and Transient simulation.

Values:

`(dc)`	`pt`	(pointwise solution in time)
`(tran)`	`pt`	(pointwise solution in time)

Default:

`(dc)`	`pt`
`(tran)`	`pt`

environment

Description:
Specifies what type of environment SIMLAB was built for. Different environments cannot be specified for DC and Transient simulation. The value reflects the nature of the underlying hardware and is therefore likely to suffer little if any changes. The notion of environment is included to allow expandability to different computer architectures such as was done with CMVSIM[5, 4], the vision circuit simulator developed on the Connection Machine[3].

Values:

`(dc)`	`serial`	(sequential computer environment)
`(tran)`	`serial`	(sequential computer environment)

Default:

`(dc)`	`serial`
`(tran)`	`serial`

B.4 Environment Variables

SIMLAB_PLOTTOOL

Description:
Specifies which program to use for plotting node waveforms when the `plot` command is used. This program must be in the user's `PATH`. By default SIMLAB uses SIMGRAPH, a modified version of the XGRAPH program.

SIMLAB_PLOTARGS

Description:
Contains command line arguments to be used when invoking `SIMLAB_PLOTTOOL` for plotting node waveforms. It is disregarded if the default tool, SIMGRAPH is used.

SIMLAB_HISTFILE

Description:
Specifies the location and name of the file to use as history file. If not set, SIMLAB will use the default naming specified at compilation time (normally "`./.simlab_hist`").

SIMLAB_RC

Description:

Specifies the location and name of the default SIMLAB configuration file. This file is read by SIMLAB at startup time. If this variable is not set, SIMLAB tries to read, in order, "`./.simlabrc`" and " `/.simlabrc`".

C Non-Standard Supported Packages

Currently SIMLAB supports two packages not included under what is called the standard functionality. These are the `Iterative Linear System Solution` package and the `Waveform Newton Circuit Simulation` package. These packages are examples of the powerfulness of SIMLAB as a tool for research in circuit simulation algorithms and are provided together with the standard distribution. However to be able to use the functionality they provide, SIMLAB must be told to incorporate these packages when building an executable. This is accomplished by proper setup of the options provided in the `config.h` configuration file.

The mentioned packages were built on top of SIMLAB and have been tested perhaps less frequently than the SIMLAB standard package (they still run for at least one test case, however). The authors welcome any comments, bug-reports, suggestions or enhancements to these packages as well as to the rest of the SIMLAB circuit simulation.

C.1 Iterative Methods for Linear System Solution

This package is distributed under the `relax` subdirectory of the source code tree. To have SIMLAB incorporate the functionality provided in this package, you must define `USE_RELAX` in the `config.h` configuration file of SIMLAB before building and installing the program.

Currently this package contains a series of algorithms for the iterative solution of linear systems. Among these are:

- Gauss-Jacobi relaxation (`gj`)
- Gauss-Seidel Relaxation (`gs`)
- Kaczmark Relaxation (`km`)
- Banded Relaxation (`vbr`)
- Jacobi Over-Relaxation methods (`jor`)
- Seidel Over-Relaxation methods (`sor`)
- Conjugate Gradient (`cg`)
- Generalized Conjugate Residual (`gcr`)
- Conjugate Gradient Squared (`cgs`)
- Preconditioned Conjugate Gradient (`pcg`)

- Preconditioned Conjugate Gradient Squared (`pcg`)
- Conjugate Gradient applied to the normal equations (`cgnr`)
- Conjugate Gradient applied to the normal equations (`cgnr`)
- Conjugate Residual (`cr`)
- Generalized Conjugate Residual (`gcr`)
- Preconditioned Generalized Conjugate Residual (`pgcr`)
- Preconditioned Restarted Generalized Conjugate Residual (`pgcr_k`)
- Orthomin (`orthomin`)

These algorithms can be specified as linear system solvers (see Functional Controls in sectionB.3) and are added to the list of possible values for the `solve_linear` parameter (for both DC and Transient modes) whenever this package is linked into SIMLAB .

The addition of this package and further setting up the `solve_linear` parameter to use one of the functions provided therein, leads to the installation of some extra variables. Normally one can expect that a bound on the number of iterations allowed for the iterative method will be one such variable. Usually it is named after the algorithm being used, for instance `maxcg`. Increasing `maxcg` may cause an otherwise non-converging method to suddenly converge. At present each mode of simulation will have its own associated variable, whose name for DC simulation will be such as `maxdccg`. Using the `who` command will allow you to determine which new variables were added to your environment when one of the new algorithms is chosen for linear system solution.

If an iterative algorithm is chosen along with a preconditioner for inproved convergence rate, a parameter called `precondition` is added to your environment. At present this package supports the following types of preconditioners:

- Identity preconditioner (`none`)
- Diagonal Preconditioner (`diag`)
- Incomplete Choleski preconditioner (`ic`)
- Incomplete Choleski preconditioner (`mic`)
- Incomplete LU preconditioner (`il`)
- Incomplete LU preconditioner with Markowitz ordering (`mil`)

For information on the algorithms available in this package we refer the user to the appropriate documentation, such as [2, 1].

D Example Configuration File

```
bootinv.cfg

% SIMLAB Boot-strapped inverter example - Configuration file
%

% Save output in diary file
diary boot.doc

% Change to 'circuits' directory
cd circuits

% Load circuit
circuit boot.rel

% Make sure to do DC solution
dodc = 1

% Use direct methods for linear solution
set solve_linear direct

% Use 2nd order backwards-differentiation integration formula
set integrate gear2

% Run
run

% Plot the results on the display
plot

% Done
diary off

quit
```

E Example Session

Example Simlab Session

```
                              < SIMLAB 1.0 >
                  (c) 1990 Massachusetts Institute of Technology
                        Research Laboratory of Electronics
                        All Rights Reserved     10 Oct 90
=> % Change to 'circuits' directory
=> cd circuits

=> % Load circuit
=> circuit boot.rel

Reading boot.rel
stop = 1.250000e-07
cmin = 1.000000e-14
lterel = 5.000000e-04
lteabs = 5.000000e-06
dodc = 0

=> % Make sure to do DC solution
=> dodc = 1
dodc = True

=> % Use direct methods for linear solution
=> set solve_linear direct

=> % Use 2nd order backwards-differentiation integration formula
=> set integrate gear2

=> % Run
=> run
```

Example Simlab session (cont.)

```
Simulating circuit boot.rel

        DC Simulation Results
        Sun Oct 14 01:12:50 1990
-----------------------------------------
                       Host:  sobolev
          Simulation Method:  Pointwise Solution
           DC Solution Time:  0
  Nonlinear Solution Method:  Newton
       DC Newton Iterations:  23
     Linear Solution Method:  Direct
 Initial Decomposition Time:  0
       Linear Solution Time:  0

      Transient Simulation Results
        Sun Oct 14 01:12:51 1990
-----------------------------------------
                       Host:  sobolev
          Simulation Method:  Pointwise Solution
    Transient Solution Time:  .966667
         Integration Method:  2nd Order Gear
                  Timesteps:  231
  Nonlinear Solution Method:  Newton
          Newton Iterations:  362
     Linear Solution Method:  Direct
 Initial Decomposition Time:  0
       Linear Solution Time:  0.0666667

=> % Plot the results on the display
=> plot

=> % Done
=> diary off

=> quit
```

F Example Circuit Files

A simple boot-strapped inverter is illustrated below.

bootinv.rel

```
; SIMLAB Boot-strapped inverter example

; Required global statement to indicate ground node
global 0

; Transistor model parameters
model tran nmos vto = 1 kp = 20u gamma = 0.31 phi = 0.6 lambda = 0.02

; Voltage sources
vsrc vdd 0 dc v=5
input in 0 pwl period = 50ns t0=0n v0=0 t1 = 15ns v1 = 0 \
                  t2 = 20ns v2=5v t3=40ns v3=5v t4=45ns v4=5v

; Transistors
mo1 vdd boot out 0 tran w=5u l=5u
mo2 vdd vdd boot 0 tran w=5u l=5u
mo3 out in 0 0 tran w=5u l=5u

; Capacitors
c1 boot 0 c c=0.001pf
c2 out 0 c c=0.01pf
c3 boot out c c=0.01pf

; Analysis options.
; Do not do DC solution.  Do Transient analysis for 125ns.
simlab options dodc = 0 stop = 125ns cmin = 1.e-14 \
        lterel = 0.0005 lteabs = 0.000005

; Record the voltages of the following nodes
plot in out boot
```

A CMOS Domino Gate with complex models is given below:

domino.rel

```
; domino CMOS circuit
; Required global statement -- First in list indicates ground node.
global 0

; Record the voltages of the following nodes
plot 1 2 3 4 5 6 7 8 9 10

; Analysis options
simlab options stop=100ns dodc=1

; Input Sources
vdd 100 0 dc v=5
vphi 10 0 pwl period=40ns t0=0 v0=5 t1=1ns \
                v1=0 t2=16ns v2=0 t3=17ns v3=5
vala3 1 0 pwl period=40ns t0=0 v0=5 t1=2ns v1=5 t2=3ns v2=0 \
                t3=18ns v3=0 t4=19ns v4=5
va2a4 2 0 pwl t0=0 v0=5 t1=2ns v1=5 t2=3ns v2=0 t3=100ns v3=0
va5 9 0 pwl period=40ns t0=0 v0=5 t1=2ns v1=5 t2=3ns v2=0 \
                t3=18ns v3=0 t4=19ns v4=5

; Devices
mpphi 4 10 100 0 pqshare w=7u l=3u ad=56p as=56p pd=30u ps=30u
ma1 3 1 7 0 nqshare w=7u l=3u ad=10.5p as=10.5p pd=17u ps=17u
ma2 7 2 5 0 sqshare w=7u l=3u ad=10.5p as=10.5p pd=17u ps=17u
ma3 3 1 8 0 sqshare w=7u l-3u ad=10.5p as=10.5p pd=17u ps=17u
ma4 8 2 5 0 sqshare w=7u l=3u ad=10.5p as=10.5p pd=17u ps=17u
ma5 4 9 3 0 nqshare w=7u l=3u ad=56p as=10.5p pd=30u ps=17u
mnphi 5 10 0 0 sqshare w=7u l=3u ad=10.5p as=56p pd=17y ps=30u
```

domino.rel (cont.)

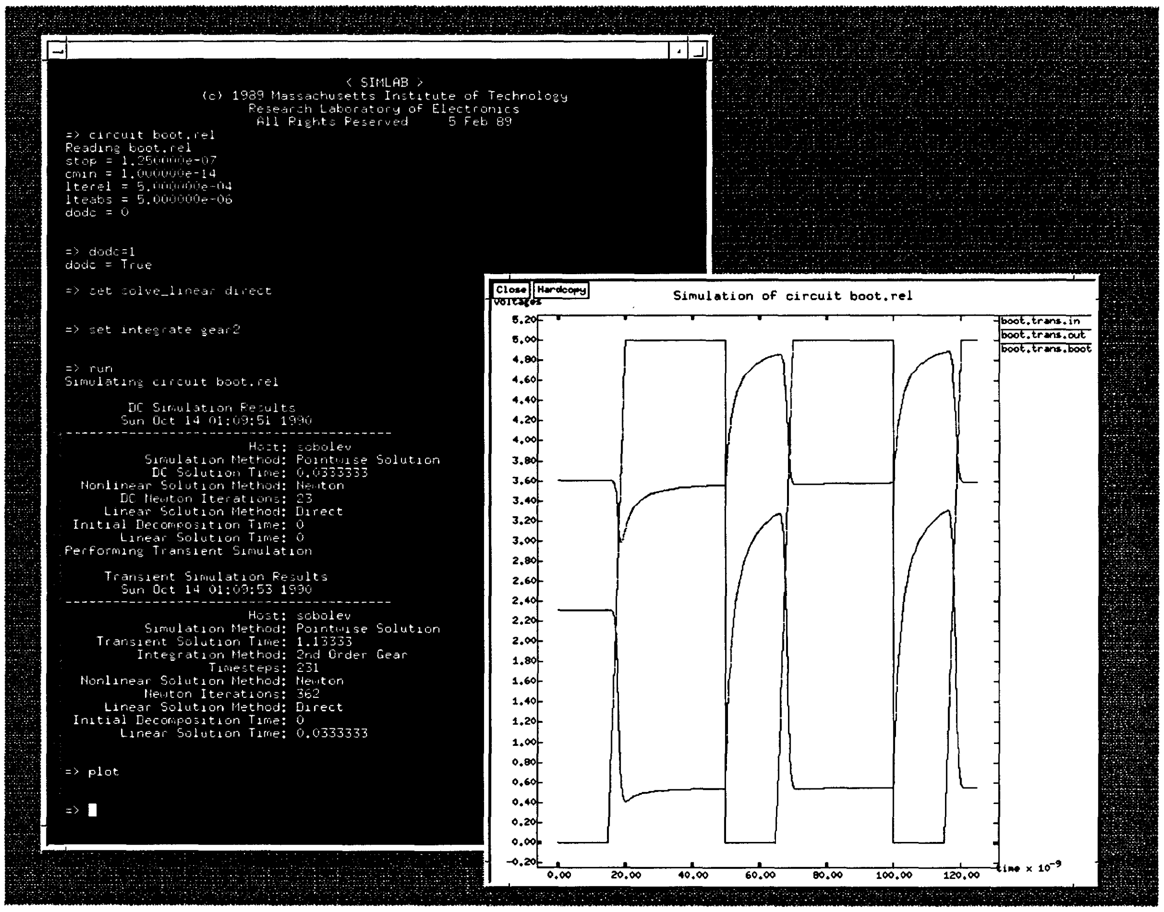
```
; Output buffer
m7 6 4 100 100 pqshare w=7u l=3u ad=56p as=56p pd=30u ps=30u
m8 6 4 0 0 sqshare w=7u l=3u ad=56p as=56p pd=30u ps=30u

; Constructed complex p and n channel models.
model nmod nmos vto=0.8 kp=35u gamma=0.2 phi=0.55 lambda=0.02 \
        cgso=1.4e-10 cgdo=1.4e-10 cgbo=2.4e-10 tox=500e-10
model pmod pmos vto=-0.8 kp=14e-6 gamma=0.37 phi=0.61 lambda=0.01 \
        cgso=2.1e-10 cgdo=2.1e-10 cgbo=2.4e-10 tox=500e-10

; Diode models
model npdmod d is=3e-9 cjo=0.73e-10 m=0.33
model nadmod d is=3e-9 cjo=0.77e-4 m=0.5
model ppdmod d is=1e-9 cjp=2.2e-10 m=0.33
model padmod d is=3e-9 cjo=1.4e-4 m=0.5

; Define nqshare and pqshare subcircuits
define nqshare (d g s b)
parameters w=10u l=5u as=1.0 ad=1.0 ps=.10 pd=.10
  device d g s b nmod w=w l=l
  dsp bs npdmod area=ps
  dpd bd npdmod area=pd
  dps bd nadmod area=as
  dpd bd nadmod area=ad
end nqshare

define pqshare (d g s b)
parameters w=10u l=5u as=1.0 ad=1.0 ps=1.0 pd=1.0
  device d g s b pmod w=w l=l
  dpd db ppdmod area=ps
  dps sb padmod area=as
  dpd db padmod area=ad
end pqshare
```



```
                        < SIMLAB >
        (c) 1989 Massachusetts Institute of Technology
             Research Laboratory of Electronics
              All Rights Reserved      5 Feb 89
=> circuit boot.rel
Reading boot.rel
stop = 1.250000e-07
cmin = 1.000000e-14
lterel = 5.000000e-04
lteabs = 5.000000e-06
dodc = 0


=> dodc=1
dodc = True

=> set solve_linear direct


=> set integrate gear2


=> run
Simulating circuit boot.rel

        DC Simulation Results
       Sun Oct 14 01:09:51 1990
-----------------------------------------
                      Host: sobolev
         Simulation Method: Pointwise Solution
          DC Solution Time: 0.0333333
  Nonlinear Solution Method: Newton
       DC Newton Iterations: 23
     Linear Solution Method: Direct
 Initial Decomposition Time: 0
        Linear Solution Time: 0
Performing Transient Simulation

     Transient Simulation Results
       Sun Oct 14 01:09:53 1990
-----------------------------------------
                      Host: sobolev
         Simulation Method: Pointwise Solution
    Transient Solution Time: 1.13333
        Integration Method: 2nd Order Gear
                 Timesteps: 231
  Nonlinear Solution Method: Newton
          Newton Iterations: 362
     Linear Solution Method: Direct
 Initial Decomposition Time: 0
        Linear Solution Time: 0.0333333


=> plot


=> █
```

Figure 1: Screen Dump of a SIMLAB session for circuit boot.rel.

G Example SIMLAB Session

References

[1] G. H. GOLUB AND C. F. VAN LOAN, *Matrix Computations*, The John Hopkins University Press, Baltimore, Maryland, 1983.

[2] M. R. HESTENES AND E. STIEFEL, *Methods of conjugate gradients for solving linear systems*, Journal of Research of the National Bureau of Standards, 49 (1952), pp. 409–436.

[3] W. D. HILLIS, *The Connection Machine*, MIT Press, New Haven, CT, 1985.

[4] A. LUMSDAINE, M. SILVEIRA, AND J. WHITE, *CMVSIM programmer's guide.* Research Laboratory of Electronics, Massachusetts Institute of Technology. Unpublished, 1990.

[5] ——, *CMVSIM users' guide.* Research Laboratory of Electronics, Massachusetts Institute of Technology, 1990.

[6] L. W. NAGEL, *SPICE2: A computer program to simulate semiconductor circuits*, Tech. Rep. ERL M520, Electronics Research Laboratory Report, University of California, Berkeley, Berkeley, California, May 1975.

[7] H. SCHICHMANN AND D. A. HODGES, *Modeling and simulation of MOS insulated-gate field effect transistor switching circuits*, IEEE Journal Solid-State Circuits, SC-3 (1968), pp. 285–289.

[8] A. VLADIMIRESCU AND S. LIU, *The simulation of MOS integrated circuits using SPICE2*, Tech. Rep. ERL-M80/7, Electronics Research Laboratory Report, University of California, Berkeley, Berkeley, California, June 1980.