

MI-SUGAR

Analog Electronic Circuit Design Environment for Mac OS X

Version: 0.5.6

Author: Berk Özer

Release date: May 5, 2004

Description

MI-SUGAR (pronounced "my sugar") is a lightweight environment for analog electrical circuit analysis. It uses the free simulators SPICE and GnuCap. You can create circuits using either the schematic entry tool or the netlist editor, then run the simulation and analyze the results in the graph plotter. MI-SUGAR supports subcircuits with which users can build their own library of elements. You can export the analysis results to various file formats.

Installation

To install MI-SUGAR simply drag the application icon into a folder of your choice.

Please refer to the help files for usage instructions!

Software Requirements

- Mac OS X 10.3

Contact Information

E-mail: info@macinit.com

WWW: <http://www.macinit.com>

Changes from 0.5.5 to 0.5.6

- * Elements now snap to alignment, making for tidier schematics.
- * Introduction of Schematic Variants: You can now have up to four different schematics ("variants") in one window. Only one is active at a time. Select one with the keys 1, 2, 3 and 4. Replace the content of a variant with that of the current variant by pressing Command plus the number of the target variant.
- * The keyboard shortcuts for panning have changed. Use the Option key in combination with the arrow keys.
- * Many bug fixes and minor changes and usability enhancements:
 - * The text element fonts and colors display correctly now.
 - * Element labels rotate and flip with the element now.
 - * The right side of the netlist editor does not disappear anymore when resizing it after having minimized it in a previous window.
 - * Copy-pasting elements now includes the connection lines.
 - * You can not move elements beyond the work area's border anymore.

- * Node elements can be copy-dragged with the Option key now.
- * Plot windows and circuit windows are not coupled anymore.

The complete development history can be found at the end of this file.

DISCLAIMER

By using this software you accept that the author is not responsible of any damage or loss of intellectual or material property, caused directly or indirectly, through the use of this software.

MI-SPICE

SPICE (Simulation Program with Integrated Circuit Emphasis) for Mac OS X

Version: 1.3

Compiled by: Berk Özer

Build date: April 15, 2004

Changes from 1.2 to 1.3:

- * Applied third party patches, collected and partly written by Charles Williams. Most notably: The command for transient analysis does correctly use the given step size now.
<http://newton.ex.ac.uk/teaching/CDHW/Electronics2/userguide/secD.html>
- * Updated the BSIM4 MOSFET model to version 4.4.0.
- * Updated the BSIMSOI model to version 3.2.0.

Changes from 1.1 to 1.2:

- * Updated the BSIMSOI model (MOS level 9) to version 3.1.1.
- * Updated the BSIM4 MOSFET model (MOS level 14) to version 4.3.0.
- * Added the Parker-Skellern MESFET-JFET model (JFET level 2).
- * Added the EPFL-EKV 2.6 MOSFET model (MOS level 44).
- * Added the HiSIM 1.2 MOSFET model (MOS level 60).
- * Removed the 'init file' warning message and made the BSIM models dump their warning messages to the terminal instead of a log file.

Changes from 1.0 to 1.1:

- * Updated MOSFET model BSIM3 from version 3.1 to version 3.2.4
- * Added MOSFET models BSIM4 (version 4.2.1) and BSIMSOI (version 3.1)

Copyright

MI-SPICE is the ported version of SPICE 3f5 to Mac OS X by Berk Özer. MI-SPICE is free for personal, academic and commercial use.

In case of redistribution the source and author must be mentioned.

- **SPICE 3f5** was developed at the University of California, Berkeley. Copyright 1993 The Regents of the University of California.
- The **BSIM3**, **BSIM4** and **BSIMSOI** extensions were developed by the UC Berkeley BSIM Research Group. Copyrights 2000, 2001, 2003.

- The **Parker-Skellern Model** was implemented by the Collaborative Nonlinear Electronics Research Facility, Macquarie University, Australia
- The **EKV MOSFET Model** is developed at the Swiss Federal Institute of Technology (EPFL) Electronics Laboratory
- The **HiSIM** model is developed at the Semiconductor Technology Academic Research Center, Hiroshima

DISCLAIMER

By using this software you accept that the author is not responsible of any damage or loss of intellectual or material property caused directly or indirectly through the use of this software.

IN NO EVENT SHALL THE UNIVERSITY OF CALIFORNIA BE LIABLE TO ANY PARTY FOR DIRECT, INDIRECT, SPECIAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES, INCLUDING LOST PROFITS, ARISING OUT OF THE USE OF THIS SOFTWARE AND ITS DOCUMENTATION, EVEN IF THE UNIVERSITY OF CALIFORNIA HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGE. THE UNIVERSITY OF CALIFORNIA SPECIFICALLY DISCLAIMS ANY WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. THE SOFTWARE PROVIDED HEREUNDER IS ON AN "AS IS" BASIS, AND THE UNIVERSITY OF CALIFORNIA HAS NO OBLIGATIONS TO PROVIDE MAINTENANCE, SUPPORT, UPDATES, ENHANCEMENTS, OR MODIFICATIONS.

Gnucap

The Gnu Circuit Analysis Package by Albert Davis

Version: 0.34

Author: Albert Davis

Release date: February 1, 2004

Description

"Gnucap is a general purpose circuit simulator. It performs nonlinear dc and transient analyses, fourier analysis, and ac analysis. It can also be run in batch mode or as a server. Spice compatible models for the MOSFET (level 1-7), BJT, and diode are included in this release. Gnucap is not based on Berkeley Spice, but some of the models have been derived from the Berkeley models."

Notes

Gnucap is free software and is distributed under the GNU General Public License (GPL). You may extract the Gnucap binary from the package and use it independently from MI-SUGAR.

Gnucap homepage: <http://www.gnucap.org/html/>

MI-SUGAR Development History

Changes from 0.5.5 to 0.5.6

- * Elements now snap to alignment, making for tidier schematics.
- * Introduction of Schematic Variants: You can now have up to four different schematics ("variants") in one window. Only one is active at a time. Select one with the keys 1, 2, 3 and 4. Replace the content of a variant with that of the current variant by pressing Command plus the number of the target variant.
- * The keyboard shortcuts for panning have changed. Use the Option key in combination with the arrow keys.
- * Many bug fixes and minor usability enhancements:
 - * The text element fonts and colors display correctly now.
 - * Element labels rotate and flip with the element now.
 - * The right side of the netlist editor does not disappear anymore when resizing it after having minimized it in a previous window.
 - * Copy-pasting elements now includes the connection lines.
 - * You can not move elements beyond the work area's border anymore.
 - * Node elements can be copy-dragged with the Option key now.
 - * Plot windows and circuit windows are not coupled anymore.

Changes from 0.5.4 to 0.5.5

- * Significant user interface modifications:
 - * The right mouse button is no longer used in the schematic tool,
 - * There are no scroll bars for navigating the work area anymore. Navigation is done by panning (Option key + drag) and zooming.
 - * Buttons have moved from the elements panel to the info panel.
 - * Icons have been renewed or made darker so they don't distract.
 - * More shortcut key combinations have been added.
- * Added unlimited undo for schematic tool.
- * Added text elements for putting notes on the schematic.
- * Added option to export results to tab-delimited text file.
- * Non-linear dependent power source.
- * Updated SPICE engine.
- * Plus many small enhancements and bug fixes.

Changes from 0.5.3 to 0.5.4

- * Improved device model management.
- * Added printing of schematics.
- * Added MOSFET elements with bulk connectors.
- * Updated the GnuCap binary. See below.
- * Many small fixes.

Changes from 0.5.2 to 0.5.3

- * Added support for subcircuits.
- * Node names are used now when converting schematic to netlist.
- * Added pulse current source element.
- * Many small enhancements and bug fixes.

Changes from 0.5.1 to 0.5.2

- * The .sugar file format has changed! Files saved with previous versions (0.5 & 0.5.1) can not be opened with this version.
- * This version does not work on Mac OS X 10.2 (Jaguar).
- * Connections between elements have changed from 2-point (single straight line) to 3-point (two lines, one horizontal and one vertical)
- * Pressing the space bar after capturing will display assigned node numbers and device information (for some elements) directly on the schematic.
- * Added dependent voltage and current source schematic elements.
- * If a schematic uses elements with non-default device models then these models are saved together with the schematic now. When a file is opened new models are imported automatically.
- * Minor behavioral fixes.
- * Updated Gnuicap binary.

Changes from 0.5 to 0.5.1

- * From this version on the right mouse button is used to connect elements (instead of the Command key + left mouse button).
- * Added choice of vertical and horizontal window layout.
- * Fixed numerous bugs.
- * Added pulse voltage, sinusoidal voltage and alternative node element.
- * Added version recognition of the .sugar file format.
- * Added copy-paste functionality between windows.
- * Added a toolbar button which moves the schematic to the lower left corner.

Changes from 0.4.4 to 0.5

- * Added a schematic entry tool.
- * Added basic capturing capability (converting schematic to netlist)
- * Added an additional file format for saving the schematic
- * Added simple device model management for use with the schematic tool

Changes from 0.4.3 to 0.4.4

- * Fixed the bug which prevented plotting on Mac OS X 10.3.
- * Added new device models to the built-in SPICE.
(See the MI-SPICE section below.)
- * Added checking of the plotting region limits before turning on logarithmic scaling.
- * Now allows plotting of results when using a custom simulator.
(Input/output of the simulator must be compatible with SPICE.)

Changes from 0.4.2 to 0.4.3

- * The grid color and background color of the plotter are customizable now.
- * Added line numbering to the netlist editor.
- * Lone CR line ending characters are automatically filtered out.
- * Added option to automatically show the 'Guides' section of the plotter controls when a guide is dragged.
- * Added option to remember the settings in the 'Plot' section of the plotter controls.
- * Added option to close all previous plot windows when a new plot window is shown.

Changes from 0.4.1 to 0.4.2

- * First commercial version.
- * Window size is stored between sessions now.
- * Better support for printing
- * Graph color is customizable now.
- * Exporting analysis results via AppleScript.
- * Other minor improvements.

Changes from 0.4 to 0.4.1

- * Added drawing of Nyquist plots for complex-valued simulation variables.
- * A donation key is required to access the plotter controls panel now.

Changes from 0.3.3 to 0.4

- * Added support for GnuCap in addition to SPICE.
- * Added zooming into and out of subregions in the plotter.
- * Added highlighting of the circuit description (by coloring lines).
- * All output is shown in a single text area now.
- * SPICE output is not in raw format anymore. Use the .PRINT command to display the results.
- * Scripting support with the **analyze** and **plot** commands.

Changes from 0.3.2 to 0.3.3

- * Complex-valued analysis results are supported now.
- * The preferences panel has been renewed.
- * SPICE comes now contained within the application bundle so you don't have to set the path to it anymore.

Changes from 0.3.1 to 0.3.2

- * Added exporting of analysis results to MathML and Matlab formats
- * Fixed an important bug in the circuit source parser.
- * Fixed a bug that would show wrong values for the ordinate of the current mouse position when the ordinate is logarithmically scaled.
- * Changed label positions in the graph plotter.
- * Modified grid lines for logarithmic scales.
- * Fixed the "Save", "Save As..." and "Open Recent" behaviour.

Changes from 0.3 to 0.3.1

- * Fixed CRLF bug, which made the application freeze when analyzing source files that were edited under Windows (or any editor that uses 'carriage return' in addition to 'line feed' to make a new line).
- * More options in the preferences panel: Automatic checking for updates, plotter background color, plotter line widths, and plotter labels font size.
- * The plots can now be copied to the clipboard.
- * Immature support for printing of plots.
- * Did some interface beautification work and changed the location of the shell output area.

Changes from 0.2.1 to 0.3

- * Fixed the mismatching of the graph visibility indicators in the plotter options.
- * Added scaling of graph variables.
- * The plotter draws a uniform grid now instead of the sweep lines.
- * Added labels to the plotter.
- * Fixed a bug which prevented plotting graphs of variables with negative sweep stepping.
- * Changed the signature of the application from 'com.macinitiative.sugar' to 'com.macinit.sugar'. Users of previous versions will have to set the path to the SPICE tool and the preferred fonts again. Sorry. You can delete the old preferences file 'com.macinitiative.sugar.plist' (in the "Library/Preferences" sub-directory of your home directory).

Changes from 0.2 to 0.2.1

- * Now correctly plots DC analyses with two sweep parameters.
- * Basic AppleScript capabilities are enabled now.
- * A scrollbar is added to the variables table in plotter.
- * To improve stability, input files larger than 256KB are rejected.
- * The fonts of the circuit source text area and of the raw analysis output text area can now be set in the 'Preferences' panel.
- * The application's icon has been modified.

END