The EdFil editor program

Windows version

✳ **Edfil elements**

✳ Connections: 1234567890-\+=

✳ Basics: rILcCviefghk

✳ Amplifiers: aAoOG#

✳ Switches: sS$pP

✳ Transistors: qQmMjJwW

✳ Etc: nxzbyutd&/R

✳ Logic: >)(][}{%@
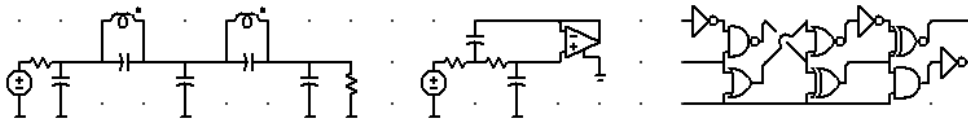
· COMMAND LINE

· .EXTRA LINES START WITH "."

✳ Formulas: (A=1) (B=10) (C=1) (X=((-B+(B^2-4*A*C)^0.5)/(2*A)))

All elements

The EdFil editor can be used to generate the input netlist file for the ASIZ program, other circuit simulators developed by the author, as SENSI, IFFT, MNAE, MNAV, etc., and also for other programs, even SPICE. It was written (by 1983) after the conclusion that it is virtually impossible to write a textual netlist for a nontrivial circuit without errors. This program is rather different from other schematic capture programs, but it is very easy to use.

The circuit is drawn on the screen using the mouse and some keyboard keys, which cause the insertion of a corresponding component between the two nodes closest to the mouse position. The orientation of the component can be changed by pressing Space or Backspace. Names and parameters for the elements are given by moving the cursor to the component, in the same area used for insertion, and touching the Return key or the left mouse button, or by pressing Ctrl-X, what causes the program to ask parameters for all visible elements. The best method is usually to firstly draw all the circuit and later give the parameters. There are default names and parameters for all elements, corresponding to usual values in the author's simulators. The editor accepts any text in the parameters, to be given accordingly to what the simulator to be used requires, that are reproduced in the netlist exactly as given, in uppercase, unless for formulas, that must be enclosed between parenthesis, that are evaluated. The parameters can be visualized by pressing Esc with the mouse cursor over an element.

Interconnections are made with the wire elements "-" and "\", crossings with "+". The ground connection is "=". Labels "0…9" can be used for long or difficult connections. If more labels are needed, a second character can be added to their names.

Comments and commands can be added to the schematic diagram with the use of the "*" and "." commands. Comments only appear in the netlist if the corresponding option is set. If a command starts with ".", it appears in

the netlist as a normal netlist line. This can be useful to insert text at the end of the netlist, usually describing other elements (as coupling coefficients for inductors).

The visualization can be adjusted by clicking the right mouse button to center the drawing on that point. There are several levels of zoom, selected with Ctrl-S or the mouse wheel. Ctrl-A centers the schematic in the screen. Vector drawings are normally used, except in the lowest zoom level. Bitmap drawing can also be selected.

Once drawn the circuit, the netlist is generated by pressing Ctrl-Q. The program then asks the names of the files where to save the netlist (.net file) and the circuit file (.cir file). The program will ask if the circuit file (.cir file) shall be saved if you try to close the program without saving it. At any moment, Ctrl-P shows the node numbers as they will appear in the netlist. Ctrl-K redraws the schematic.

An important feature of the program is the optional use of formulas in the parameters. Any mathematical expression using constants and values given in a value list can be used for a numerical parameter, listed as a formula between parentheses, separated from the remaining text by spaces, without internal spaces. The expressions are evaluated in the netlist generation. This allows easy use of component value lists generated by synthesis programs, or even the use of EdFil as a synthesis program, with components parameters given by design formulas. The "=" sign can be used in formulas to create new values in the internal value list. Comments (any text, actually) can be used to create sets of values in this way, and the program can be used as a simple spreadsheet. Evaluation is first from top to bottom for comments and commands and then from bottom to top for other elements, always from right to left for different elements, and from left to right in the same element. Note that "=" has high precedence, not low, so always use parenthesis after "=" if what follows is a formula. The operators accepted are sin(), cos(), tan(), log(), exp(), *, /, |(//) ,^ ,= ,+ ,-, and (). The log is natural. Formulas are case-insensitive.

Blocks can be created by pressing Ctrl-B with the mouse cursor at one corner and moving the mouse. They can then be copied with Ctrl-C, moved with Ctrl-V, or deleted, with confirmation, with Ctrl-Y. Pressing Ctrl-B again unmarks the block.

Known bugs and features that eventually may be changed:
The ordering of the elements is a bit strange. The program tries to assign low node numbers to nodes connected to resistors, then capacitors, inductors, etc., from bottom to top, right to left. A resistor at the bottom right would be connected to node 1.
Formulas in different comments aligned horizontally are evaluated from right to left. Formulas in the same comment are evaluated from left to right.
The elements added with lowercase keys are stored in the .cir file with uppercase first letters in the name, and vice-versa.
An element with a connection placed at the lower left corner of the drawing space will be grounded, connected to node 0.
The bitmap drawings are quite crude, due to the low resolution, 16x16, of the elements. From version 2.0, better vector drawings are used when zoom is used, but the elements are still restricted to the same square area. When vector graphics are used, the grid dots are not redrawn when an element is deleted.
The elements can have a maximum of 4 terminals. It is possible, however, to use other elements as extensions, depending on implementation in the simulator program. An example are the set/reset pins for flip-flops.
There is no "undo", so always remember to save the work before making significant changes.
Although there are copy and move block commands, the clipboard is not used, so it is not possible to directly copy between instances of the program. As a resource, the .cir files can be edited, so a part of a circuit can be copied to another. It is enough to place a block to be copied at the bottom of the screen, and then copy the corresponding lines in the .cir file, which will be the first after ordering, to another .cir file. Leave enough space to avoid overlaps.
Formulas cannot have internal spaces. Actually, spaces before values are allowed, but not after.

Recent changes:
Version 1.0k, 18/5/2005 - Added command to center the circuit in the screen (Ctrl-A). The parameters of the elements can have up to 100 characters (50 in the previous versions). Commands (".") are always sent to the netlist.
Version 1.1, 30/10/2008 - Added operational amplifier with 4 terminals and connections as the other controlled sources ("o"). The analysis programs will eventually be modified to accept it.
Version 1.1a, 06/11/2008 - Bug with the centering function corrected. Added amplifier with 4 terminals ("a").

Version 1.2, 28/10/2009 – Added elements B, N, K, and W.
Version 1.3, 25/01/2010 – Added element $, voltage-controlled switch.
Version 1.4, 7/9/2011 – Added element # (translates to G), balanced operational amplifier. Added Exp function. Log is Ln.
Version 1.5, 6/12/2011 – Added element &, voltage multiplier.
Version 1.6, 28/12/2011 – Elements W and w (MOS transistors with 4 terminals) are listed as M in the netlist. Formulas can be used in the comments. Larger parameter text (200 characters).
Version 1.6a, 19/1/2012 – Corrected bug in the "." command.
Version 1.7, 12/12/2012 – Added logic gates.
Version 1.7a, 14/3/2013 – A command line without "." is generated if the text starts with ".". Useful for coupling coefficients between inductors.
Version 1.7b, 23/11/2013 – Exponentiations work for negative arguments when the exponent is 2, 3, or 4.
Version 1.8, 12/9/2014 – Element "%", flip-flop (provisory).
Version 1.8a, 4/10/2014 – Element "R", identical to "r" with nonlinear resistor drawing.
Version 1.8b, 18/10/2014 – Changed multiplier drawing, corrected bug in the ordering.
Version 1.9, 7/11/2014 – Added divider "/".
Version 1.9a, 6/4/2015 – Corrected bug causing some variables starting with E being interpreted as numbers, and correct treatment of expressions with sums of numbers in exponential notation.
Version 1.9b, 28/7/2015 – Zoom with the mouse wheel. Ctrl-I inverts the colors.
Version 1.9c, 21/8/2015 – Errors in formulas cause a popup warning. Center command in menu. Esc lists parameters of the element under the cursor. F1 shows the control screen. Ctrl-W shows connections in different colors for different labels.
Version 1.9d, 26/9/2015 – Formulas are case-insensitive.
Version 1.9e, 2/10/2015 – Corrected error in the detection of invalid formulas.
Version 1.9f, 10/12/2015 – Values must start with letters. Corrected bug in the number of values.
Version 1.9g, 13/4/2016 – The program always saves the drawing file when saving the netlist. Parameter Ld added for MOS transistors.
Version 1.9h, 29/6/2016 – Added double switch with control node "P".
Version 1.9i, 19/11/2016 – Modified some default values. Multiple spaces allowed between formulas.
Version 1.9j, 3/11/2017 – Bug allowing changing the first letter of a component name fixed.
Version 2.0, 5/1/2018 – Vector drawings when zoom is used. Small changes in some drawings, as polarities indicated in capacitors and inductors. Using file association with .cir files now opens the file with focus in the editor screen. Larger fonts in the dialog boxes and selectable font for the texts. Extension element "@" for flip-flops. Two kinds of inductors and capacitors, with and without polarity indication. Ideal and real diodes.

Contacting the author:

Dr. Antonio Carlos M. de Queiroz
Programa de Engenharia Elétrica COPPE
Universidade Federal do Rio de Janeiro
CP 68504 21941-972 Rio de Janeiro, RJ, Brazil
E-mail: acmq@ufrj.br
http://www.coe.ufrj.br/~acmq

Last update: 24/1/2018