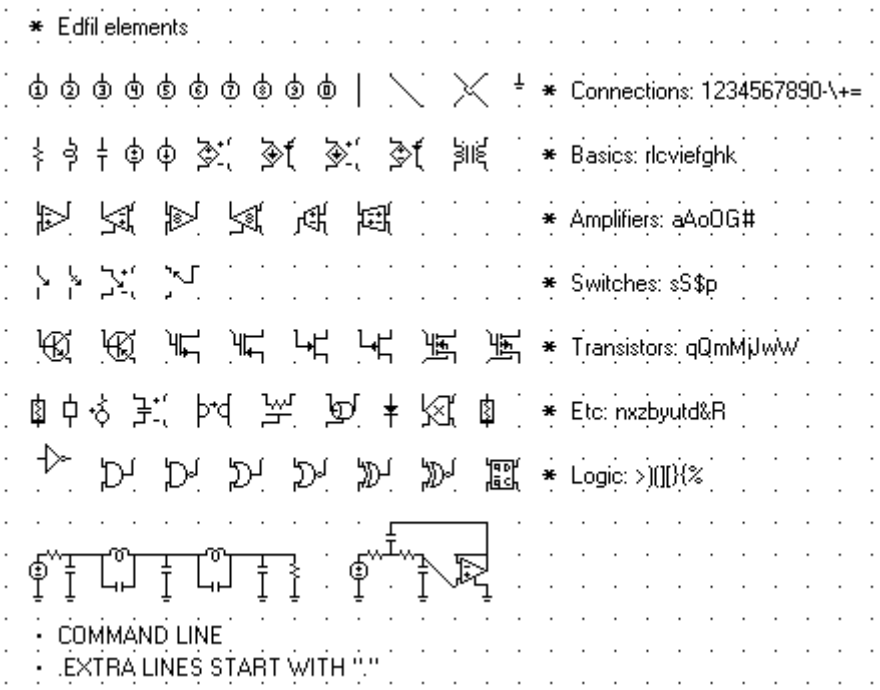
The EdFil editor program

Windows version



All elements

The EdFil editor can be used to generate the input net-list file for the ASIZ program, other circuit simulators developed by the author, 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 the schematic capture programs found in other packages, 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 and touching the Return key or the left mouse button, or by Ctrl-X, what causes the program to ask parameters for all the 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 in parenthesis, that are evaluated.

Interconnections are made with the elements "-" and "\", crossings with "+". Them ground connection is "=". Labels "0…9" can be used for long or difficult connections. If more labels are needed, just change 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.

The visualization can be adjusted by clicking the right mouse button to center the drawing on that point. There are three levels of zoom, selected with Ctrl-S. Ctrl-A centers the schematic in the screen.

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. 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. Use comments (or resistors with both nodes grounded, obsolete method) to create sets of values in this way, and operate the program 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. Always use parenthesis after "=" if what follows is a formula. The operators accepted are sin(), cos(), tan(), log(), exp(), *, /, |(//) ,^ ,= ,+ ,-, and (). The log is natural.

For formulas in comments, create values and reference them in uppercase, because the program converts everything to uppercase, except comments.

Blocks can be created by pressing Ctrl-B 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:
Don't use values as E0, E1, etc., because they will be evaluated as zero (0×10^0, 0×10^1 etc.).
The ordering of the elements is a bit strange. Be careful with formulas in different comments aligned horizontally, as they will be evaluated from right to left. Formulas in the same comment will be 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.
Wrong formulas are listed in the netlist with a question mark added, and are not evaluated.
Values created in lowercase in comments are not recognized in elements, because the texts there are converted to uppercase.

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 r with nonlinear drawing.
Version 1.8b, 18/10/2014 – Changed multiplier drawing, corrected bug in the ordering.

Contacting the author:

Dr. Antonio Carlos M. de Queiroz
Programa de Engenharia Eletrica 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: 19/10/2014