

L2Spice user manual

Contents

L2Spice user manual	1
Application description	2
Quick guide	2
Conversion	2
Text Editor Interface	4
Conversion Parameters	5
Menu	6
Command Line Interface	9
CLI Options	9
Usage examples	10

Application description

L2Spice is a free, open source, cross-platform tool, which is designed to optimize the designing-simulating netlist workflow.

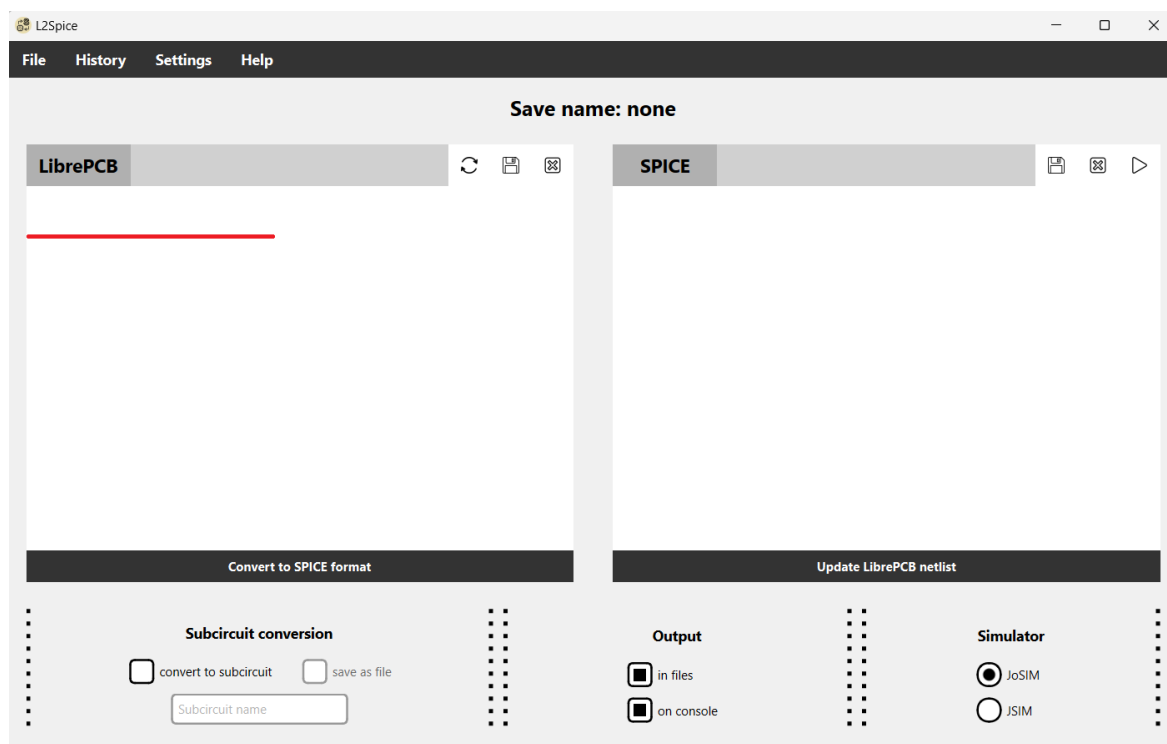
L2Spice requires a custom “Superconductors” library for LibrePCB with different independent sources, superconductors and special JoSIM/JSIM commands.

L2Spice provides not only a windowed application, but also a command line interface (CLI).

Quick guide

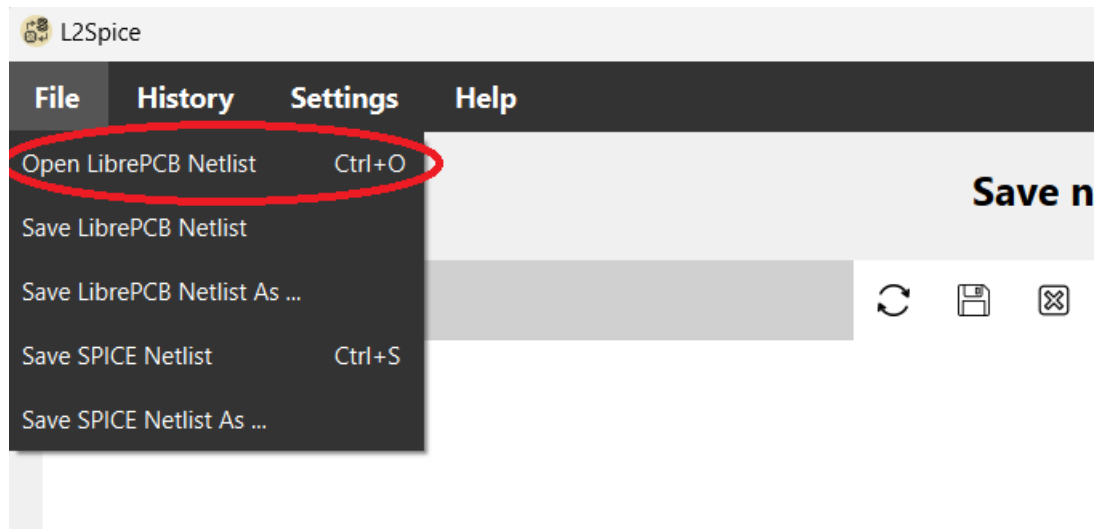
Conversion

The windowed version of L2Spice allows you to easily load the LibrePCB circuit, convert it to Spice netlist and update LibrePCB circuit back. To convert the LibrePCB circuit you need to load it first. You can do it by directly copy-pasting it into the text field on the left side of the application.



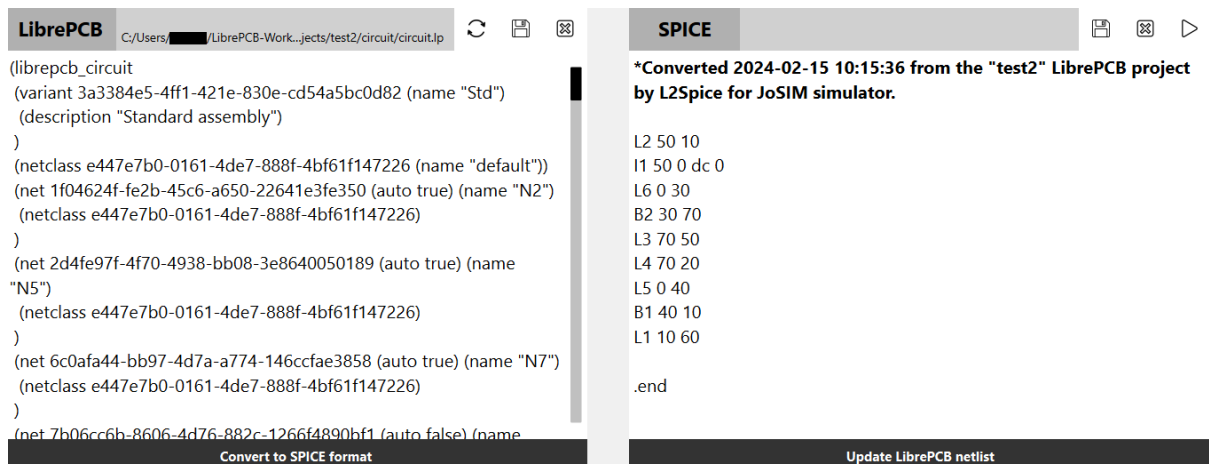
Picture 1 Main Window

You can also do it by pressing File > Open LibrePCB Netlist or just pressing Ctrl + O. You will see the common file dialog where you can choose the file to load, but it must be with “.lp” extension.



Picture 2 Open LibrePCB Netlist

After you have loaded the LibrePCB circuit you can simply press the “Convert to SPICE format” button and it will automatically create the SPICE netlist on window to the right.



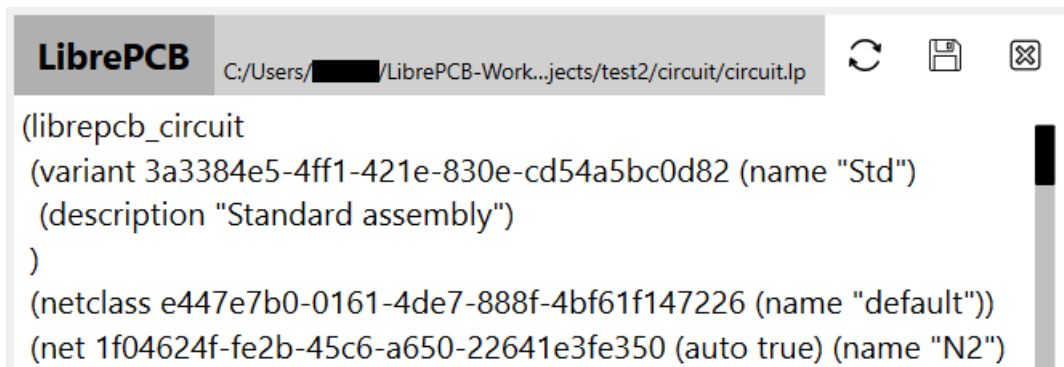
Picture 3 Text Editors

Also, you can change the parameters on the SPICE text editor and by pressing “Update LibrePCB netlist” it will update them into the LibrePCB circuit. (You cannot change the names of the components, the connections between them or the source types)

Text Editor Interface

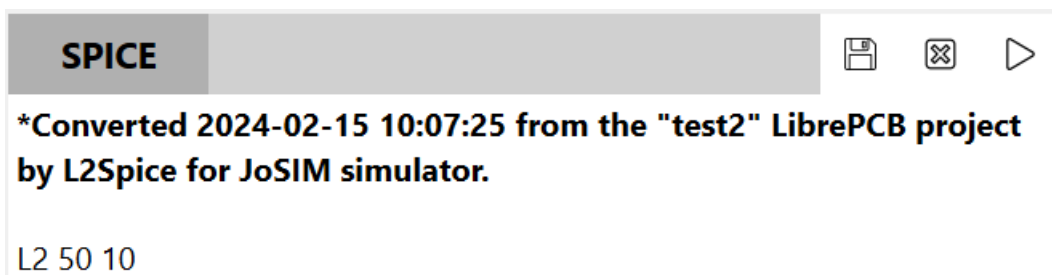
Above each text editor you can see the netlist format label, the file label and 3 buttons. The file label displays the current file where the circuit is saved or was opened from. It can also be empty.

As for LibrePCB circuit editor, you can have the “Reload”, “Save” and “Close” buttons. The “Reload” button allows you to pull up the last state of the file. It will help you when you’ve made the changes in LibrePCB circuit, saved them, and want to fast get those changes in the converter. The “Save” button allows you to save the changes into the linked file. And the “Close” button allows you to clear the text editor and remove the linked file.



Picture 4 LibrePCB Text Editor's Features

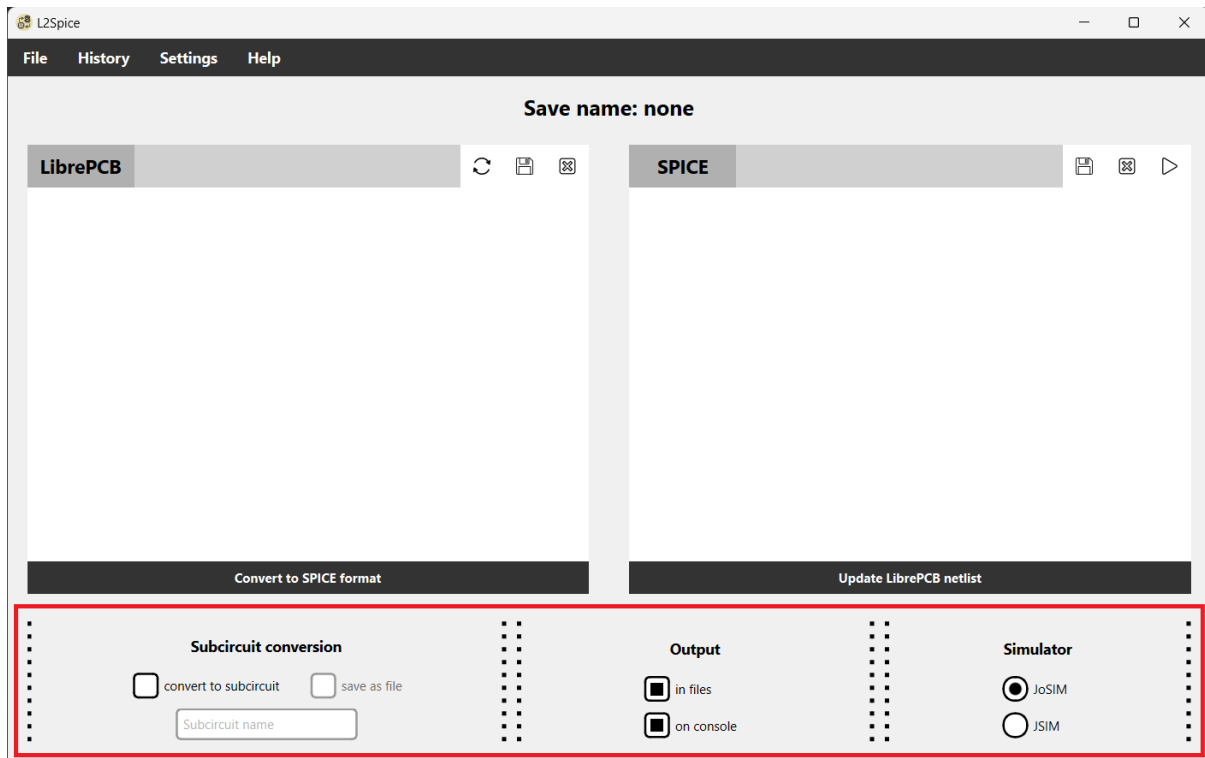
SPICE netlist editor has also 3 buttons, but with small changes. You can see the familiar “Save” and “Close” buttons which do the same functionality, but for SPICE netlist, and a new “Execute” button. This button allows you to directly simulate your netlist in JoSIM or JSIM simulators if you have downloaded one of them and specified the path to executables in the preferences.



Picture 5 SPICE Text Editor's Features

Conversion Parameters

L2Spice allows to customize the conversion process by specifying the conversion params on the bottom of the application.



Picture 6 Conversion Parameters

“Subcircuit conversion” parameters allow you to tell the application whether the given circuit is going to be used as a subcircuit or not. If it’s going to be subcircuit, you can give it a name and specify do you want to save it immediately after the conversion. If the name is not specified, the default name will be used.

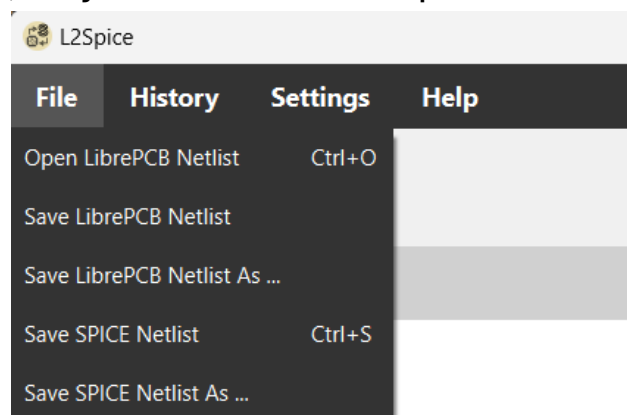
“Output” parameters allow you to specify what outputs of the simulation do you want to see. By outputs of the program, it is meant “.PRINT ...” statements. By default, both checkboxes are checked. If you remove some outputs from the program (if you want to see only and console part or only the file part or neither of them) or can uncheck them.

“Simulator” parameters allow you to switch between what version of the simulator you will use afterwards. The conversion between those simulators is a bit different, so you need to choose the simulator before running the conversion.

Menu

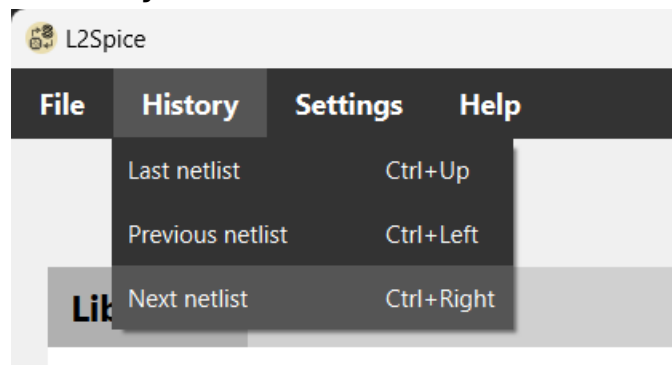
L2Spice has a menu with a wide range of options.

“File” menu allows you to open LibrePCB netlist, save LibrePCB netlist and save SPICE netlist. Some of the options also have shortcuts, as you can see in the picture.



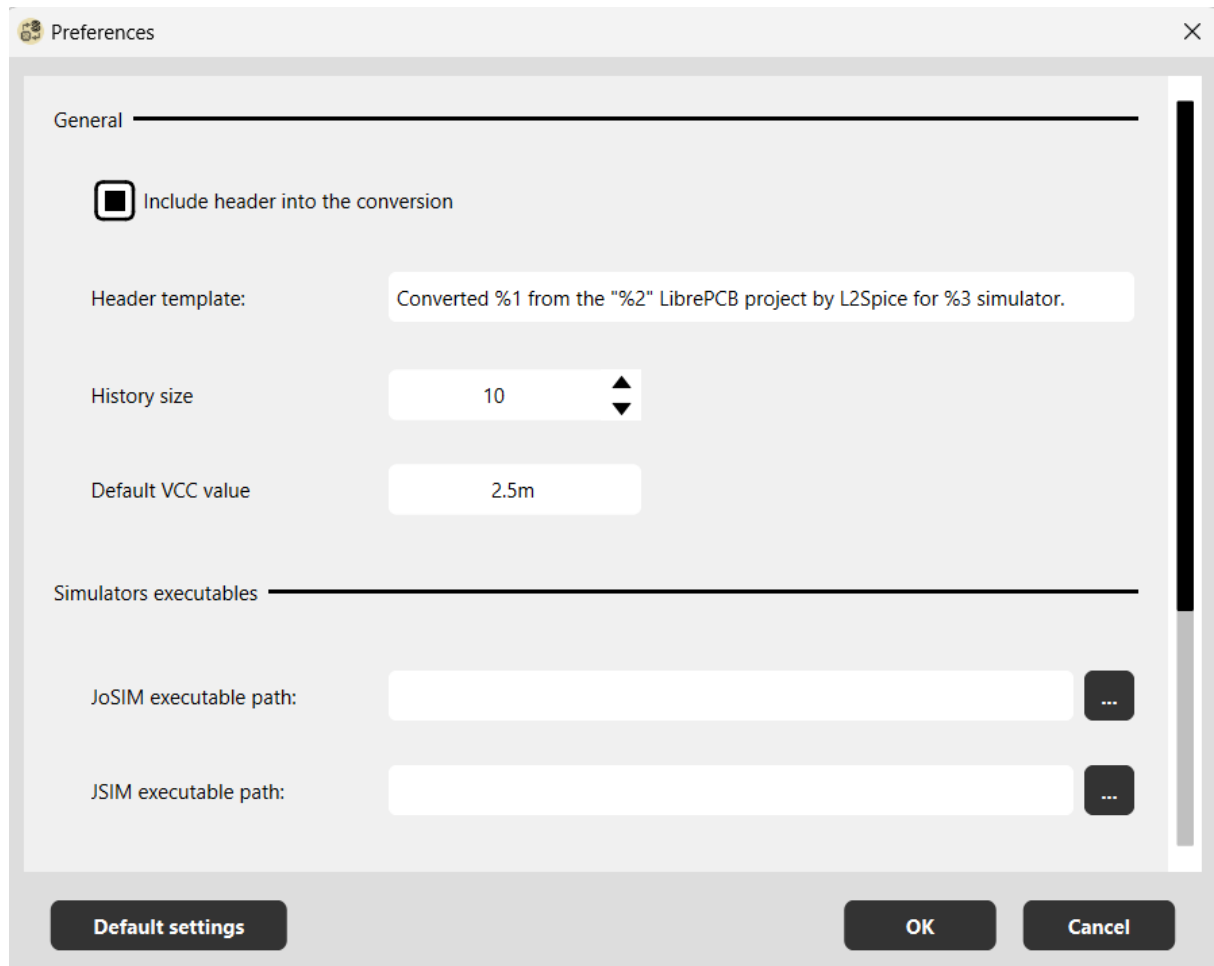
Picture 7 File Menu

The following option in the menu is labeled "History." This feature allows you to navigate through the application's history. Whenever you convert to SPICE or update the LibrePCB netlist, the application automatically saves the conversion data. This allows you to effortlessly switch between different states, toggling back and forth as needed. Convenient shortcuts are provided for each option in the “History” menu.



Picture 8 History Menu

L2Spice also has a customizable “Preferences” menu. You can open it by going to Settings > Preferences or just by pressing F8.



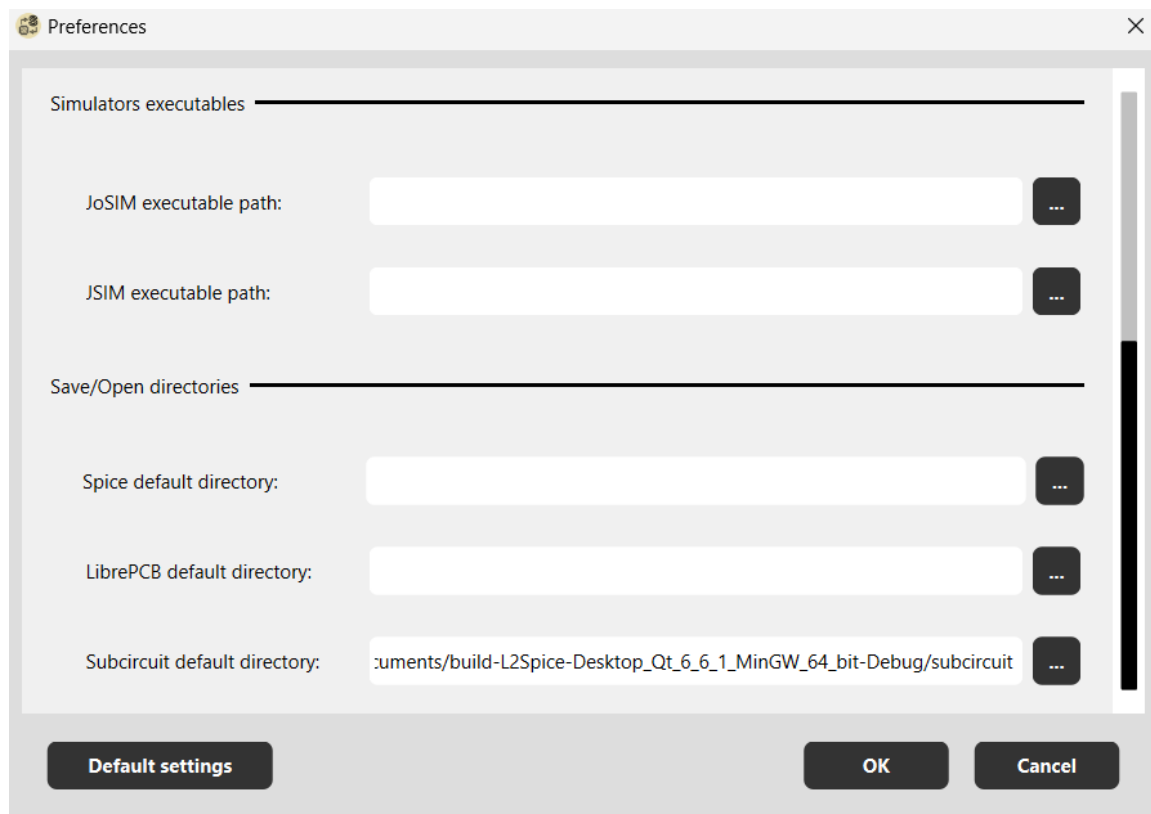
Picture 9 Preferences (1)

Here you can indicate do you want to include the header for the conversion or not, change such parameters as header template, history size, default VCC value or indicate the path to simulators executables.

As for header template, it can display next options:

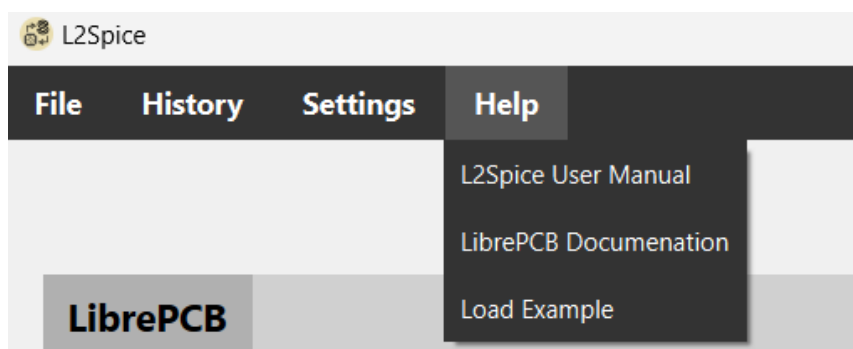
- %1 – time of the conversion
- %2 – LibrePCB project name
- %3 – simulator’s name (JoSIM/JSIM)

You can also customize the default directory where you want to open/save either SPICE netlist, LibrePCB circuit or subcircuit. That will allow you to indicate the exact path where the file dialog should be opened or subcircuit should be saved.



Picture 10 Preferences (2)

And the last but not least option in menu is called “Help”. There you can open L2Spice user manual, LibrePCB documentation or load the basic example of LibrePCB circuit.



Picture 11 Help Menu

Command Line Interface

You can use the application with command line arguments.

CLI Options

To execute the application, you need to call the L2Spice.exe with the next command line arguments:

- -h or --help – is the command-line option used to display the help message for the application. When invoked, the application will provide a summary of its usage, including a list of available command-line options, their descriptions, and examples of how to use them.
- -v or --version – is the command line option used to display the current version of the application.
- -i or --input – is a command-line option used to indicate the input file for the application. When using this flag, you need to provide the filename of the input file immediately after the flag. The application will then use this specified file as input for the conversion process. This flag is mandatory to use for the conversion.
- -o or --output – is a command-line option used to indicate the output file for the application. When using this flag, you need to provide the filename of the output file immediately after the flag. The application will write the conversion result into the given file. This flag is mandatory to use for the conversion.
- -s or --subcircuit – is an optional command-line flag used to indicate the application, that given circuit should be converted to the subcircuit.
- -j or --jsim – is an optional command-line flag used to indicate the application, that given circuit should be converted for the JSIM simulator. By default, the application converts the circuit for the JoSIM simulator.

- -wc or --without-console – is an optional command-line flag used to redirect output from the file to the console. It can be helpful for testing if you want to see the results of the conversion immediately. Usually, the output depends on what you have written in the circuit, but using this flag you will remove all “.FILE” statements from the conversion.
- -wf or --without-file – is an optional command-line flag used to remove the console output from the simulation. It can be used if you don't want to see any console output during the simulation.

Usage examples

L2Spice -h

Display the help menu.

L2Spice -i <libre_circuit>.lp -o <spice_netlist>.cir

Converts the input LibrePCB circuit to SPICE netlist.

L2Spice -i <spice_netlist>.cir -o <libre_circuit>.lp

Updated the LibrePCB circuit depending on SPICE netlist.

L2Spice -i <libre_circuit>.lp -o <spice_netlist>.cir -s -j

<>

*L2Spice --input <libre_circuit>.lp --output <spice_netlist>.cir -
-subcircuit --jsim*

Converts the input LibrePCB circuit to the SPICE subcircuit for the JSIM simulator.