

## Educational package for PSM projects

### Installation

#### 1. Install Linear Technology “LTspice” (Schematic Editor and Simulator)

- 1.1 Download the executable setup program LTSpiceIV.exe from:  
<http://www.linear.com/designtools/software/#LTspice>

#### 2. Install Glade Layout Editor:

- 2.1 Download the compressed glade tree from <http://www.peardrop.co.uk/>, choosing the correct version for the operating system in use.
- 2.2 Unzip (for example with “jzip program”) the downloaded file into an empty directory. A sub-dir named “glade4\_win32” or “glade4\_win64” should be created. The glade executable (glade.exe) is located into this directory. There is no need to launch a setup, just launch glade.exe to execute the program.
- 2.3 Launch glade (no set-up is required, simply execute the glade.exe executable) to check that the installation is complete. If an error is generated, probably the computer does not still include the required DLLs. In that case, launch the “vcredist\_x32.exe” or “vcredist\_x64.exe” file, as suggested in the Readme file included into the “glade4\_win32” or “glade4\_win64 directory.

#### 3. Install the python interpreter. Python is an extremely versatile language that can be used to create simple or very complex programs in a relatively short time. It is an interpreted program (that is, it is not compiled) so that it needs a program (the interpreter) to execute the code. Two different python main versions exist: python 2 and python 3. Python 2 (current sub-version 2.7.6 ) is the older one, but it represents the best option in terms of available modules, additional libraries and compatibility with other programs. Python is required by Glade for automating DRC and EXTRACT tasks.

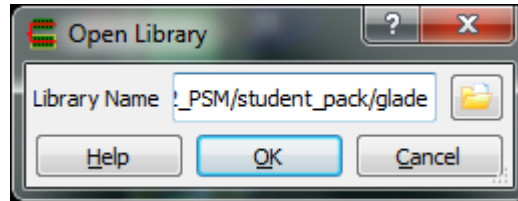
- 3.1 Download the python setup (version 2.7.6) from: <https://www.python.org/> and execute the installer (.msi file).

#### 4. Download the minimal design kit (created by the teacher) and customize it

- 4.1. Download the .zip work directory (psm\_work.zip)
- 4.2 Uncompress it into a known location. Two sub-dirs “LTSpice” and “glade” should have been created.
- 4.3 Get into LTSpice dir and test whether LTSpice is correctly installed by clicking (twice) on work\_bench.asc. LTSpice should start and open schematic diagram with a single subcircuit and two voltage sources. Launch simulate->run and a dc-sweep simulation should be executed with no errors.
- 4.4 Set-up the glade environment by entering the sub.dir glade (within “psm\_work”). Right click onto the file “layout.bat” and choose “modify”, The batch file should be opened by a text editor (typically notepad.exe).
- 4.5 Modify the line where the  
“set GLADE\_HOME=C:\Program Files\glade4\_win32” line appear, by substituting your

actual path to the glade install directory. Note that lines beginning with “rem” are comments.

- 4.6 Test glade by executing the layout.bat batch file. Glade should open normally. After that open the example library by the menu command: file->Open Lib. A dialog window should open like that shown in the figure:



Click on the directory icon on the right to choose the library. Navigate up to your psm\_work\glade directory and then select the “psm” directory. Note that a library is a directory. After selecting the library, press OK in the original window. The library psm should be added on the left window of the glade program.