

Educational package for PSM projects

Installation

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

1.1 Download the executable setup program from:

<https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html#>

1.2 Execute the setup (.exe) program.

1.3 For Linux operative systems, it is possible to use LTspice using the Wine environment. See for example: http://ltwiki.org/index.php5?title=Running_Under_Linux

2. Install Glade Layout Editor:

2.1 Download the compressed glade folder from the site

<https://peardrop.co.uk/>

choosing the latest version available for your platform (windows 32 and 64 bit, linux and MAC OS are supported)

Choose a version starting with number “5”. For example, “5.0.22”. Versions that starts with “4” are based on python “2” instead of python “3” and are not compatible with the design kit.

2.2 Unzip the downloaded file into an empty directory. Check that the full path of the chosen empty directory does not include spaces. Best option: create the directory as a sub-dir of c:\ A sub-dir should be created (named “glade4_win64” for Windows 64 bit). The glade executable (glade.exe) is located into this directory.

2.3 Windows users: Run the vcredist_x64 application, included in the glade_win64 directory, to install the required VC++ redistributables. You can skip this step and try next step (launch of the glade executable). If it runs with no error messages, your operating system does not require the “vcredist” execution. If glade does not start, then launch vcredist_x64.

2.4 Launch glade (no set-up is required, simply execute the glade.exe executable) to check that the installation is complete.

2.5 Windows users: do not follow the README.txt instructions included into the glade directory, since most operation are already performed by a script of the Design Kit. (see next section). Other platform users (different from Windows) should follow the README instructions.

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

3.1. Download the .zip work directory (psm_work_v4.zip). Note that the version number may change depending on the last update.

3.2 Uncompress it into a known location. Again, create a directory where to uncompress the zip file with an as simple as possible path. Two sub-dirs “LTSpice” and “glade” should have been created.

3.3 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.

- 3.4 Set-up the glade environment by entering the sub.dir glade (it is inside “psm_work”). Right click on the file “layout.bat” and choose “modify”, The batch file should be opened by a text editor (typically notepad.exe).
- 3.5 Modify the line where the “set GLADE_HOME=C:\Program Files\glade4_win64” line appear, by substituting your actual path to the glade install directory. Note that lines beginning with “rem” are comments.
- 3.6 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.
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. Expand the psm library and check if a cell (inverter) with a view (layout) is present and shows regularly

To start a new project:

- 1) Open Glade by launching the layout.bat script
- 2) Create a new library and choose the technology file: psm025.tch (located in psm_work\glade)
- 3) Create a new cell (view: layout)