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 form 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)
.   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 subdir 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.
      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 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.zip)
      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 shoud
   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. A dialog window should open like that shown in the figure:

Open Library

Library Name _PSM/student_pack/glade

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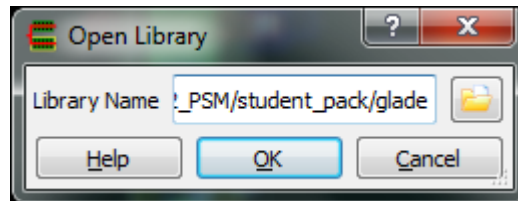
Click on the directory icon on the right to choose the library. Navigate up to your psm_work\glade directory and then select the "psm" directory.
Note that a library is a directory. After selecting the library, press OK in the original window. The library psm should be added on the left window of the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