

Digital Electronics Project 3 – Ngspice Installation

Due: 23:59, Mar. 17, 2022

There are many methods to install software programs on a Linux system [1]. Typically, package managers and software repositories are provided in many different Linux distributions. The software repository works pretty much like the App Store for our mobile (Apple/Android) devices. For some proprietary software programs such as Microsoft Skype, Google Chrome, etc., Linux users require to download installation packages via corresponding official websites. Besides of software repositories, some Linux software programs are in pre-compiled form, which are designed to run on any Linux distribution without installation process (i.e., just unpack the archive to a folder). Occasionally, there are still some software programs required to be configured and compiled from source code form. Compiling from source code is not a daily task Linux users should do, but it is also not as hard as it may sound [2].

Ngspice is an open-source circuit simulator [3]. We will use the simulator to analyze circuit behaviors in later lectures. In this project, you are going to install ngspice software on your system. If you are using WSL environment, you have to install an X-server for Microsoft Windows so that the Graphical User Interface (GUI) could be properly displayed. We recommend **MobaXterm** which includes both X-server and remote terminal connections.

In order to verify the correctness of the installed program, we provide a simple regression flow. Please download the "test.sp" SPICE circuit netlist in your system. Use the following steps to launch SPICE simulation and examine the result.

- (1) shell command: `ngspice test.sp`
- (2) in ngspice command line: `run`
- (3) in ngspice command line: `plot v(in) v(out)`
- (4) in ngspice command line: `quit`

[BONUS] Extra bonus points (up to 50) will be granted for students who compile the ngspice from its source code form rather than using "sudo apt install ..." directly. However, you might encounter some errors (e.g., missing compilation toolchain and dependent packages) when you try to compile the software. To tackle these problems, you might need to carefully examine the error messages and try to figure out the solutions via Google and Moodle Discussion Forum.

Please submit your learning report according to the following rules:

- 1- The font size of your report is 12 in PDF format.
- 2- The filename is your student ID (e.g., B12345678.pdf).
- 3- 200 – 500 words to describe how you install the program.
- 4- One screenshot which shows the ngspice execution result of "test.sp".
- 5- More different screenshots of your system are encouraged.
- 6- A list of your reference is a must.

Reference:

[1] How to Install Software on Linux,

URL: <https://www.howtogeek.com/191245/beginner-geek-how-to-install-software-on-linux/>

[2] How to Compile and Install from Source on Ubuntu,

URL: <https://www.howtogeek.com/105413/how-to-compile-and-install-from-source-on-ubuntu/>

[3] Ngspice official website,

URL: <http://ngspice.sourceforge.net/>