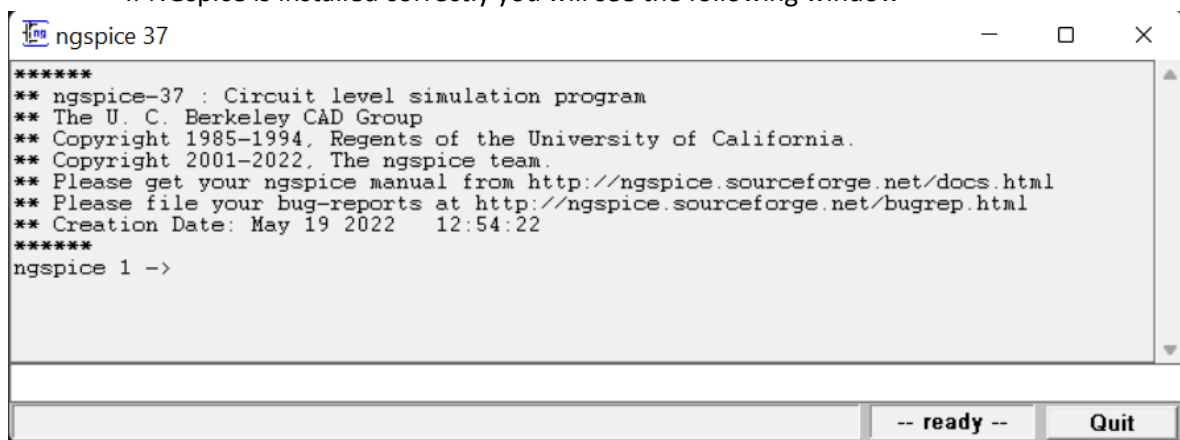


NGspice Installation

EE236

1. For Windows:

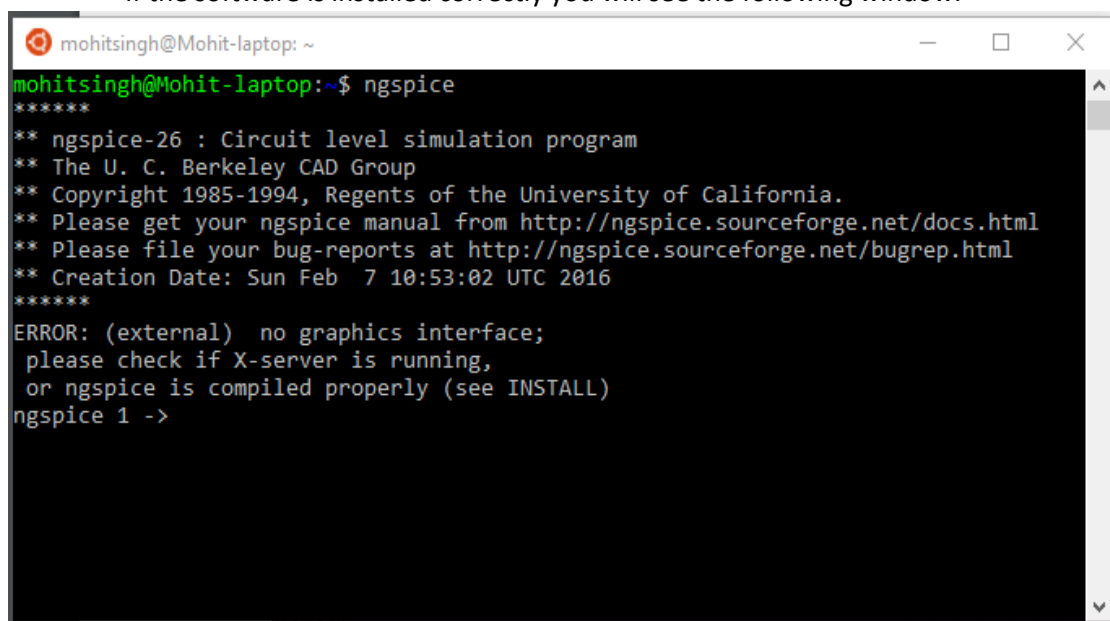
- Download the NGspice from the following link:
<https://sourceforge.net/projects/ngspice/files/>
Download Latest Version “ngspice-37_64.zip” (9.8MB)
- Unzip the downloaded file to obtain a folder ‘Spice64’.
- Copy the Spice64 folder to the working directory.
- To run the software, go to Spice64>bin and double click on ngspice.exe.
- If NGspice is installed correctly you will see the following window



```
ngspice 37
*****
** ngspice-37 : Circuit level simulation program
** The U. C. Berkeley CAD Group
** Copyright 1985-1994, Regents of the University of California.
** Copyright 2001-2022, The ngspice team.
** Please get your ngspice manual from http://ngspice.sourceforge.net/docs.html
** Please file your bug-reports at http://ngspice.sourceforge.net/bugrep.html
** Creation Date: May 19 2022 12:54:22
*****
ngspice 1 ->
```

2. For Ubuntu:

- Open the terminal and type the command ‘sudo apt-get install ngspice’ to install NGspice.
- To invoke NGspice type ‘ngspice’ in the terminal.
- If the software is installed correctly you will see the following window.



```
mohitsingh@Mohit-laptop: ~
mohitsingh@Mohit-laptop:~$ ngspice
*****
** ngspice-26 : Circuit level simulation program
** The U. C. Berkeley CAD Group
** Copyright 1985-1994, Regents of the University of California.
** Please get your ngspice manual from http://ngspice.sourceforge.net/docs.html
** Please file your bug-reports at http://ngspice.sourceforge.net/bugrep.html
** Creation Date: Sun Feb 7 10:53:02 UTC 2016
*****
ERROR: (external) no graphics interface;
please check if X-server is running,
or ngspice is compiled properly (see INSTALL)
ngspice 1 ->
```