

Using NGSpice on your PC (Guide written for Windows).

Setup electric for SPICE

Bring up File→Preferences.

Then select Categories→Tools→Spice.

- Set “Spice engine” to “Spice 3”.
- Set “Spice Level” to “3”.
- Set “Spice primitive set” to “SpicepartsS3”
- Select “User header cards from file:” and set the file to your file path to the 22nm_HP.pm file.

C:\Users\yourname\Desktop\ESE370\22nm_HP.pm (example)

Then click “Ok” to complete these changes.

The “header cards” in this case are the model for the transistors we’re simulating. In this case, we are using the 22nm high performance (HP) transistors from the Predictive Technology Models (PTM) that came from ASU: <http://ptm.asu.edu/>

Note: We made a slight modification to the name of the models in the version stashed in /home1/e/ese370/ptm/22nm_HP.pm so that it would work with Electric. If you grab the PTM version from the web, you will need to change the model names as well. We suggest you start with our version to avoid this technicality.

The model file is also available on the course syllabus for download if you install Ngspice locally on your own machine.

In future labs we may use different models (e.g. different device feature sizes) or compare the behavior of circuits between different technology models, so, at times, you will need to change the header files in use.

To finish setting this up for a 22nm technology, bring up File→Preferences again and go to: Categories→Technology→Scale. Select mocomos and set its “Technology scale:” (at bottom) to 22 (for 22 nm).

Download **SPICE** from <http://ngspice.sourceforge.net/>

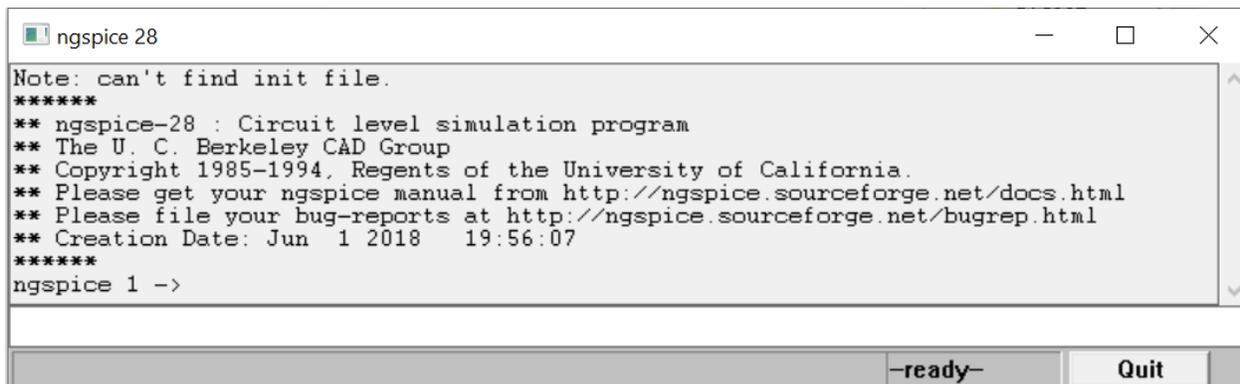
Unzip the files.

Main difference from using eniac

Save SPICE decks (.SPI) files into the **bin** folder from your extraction

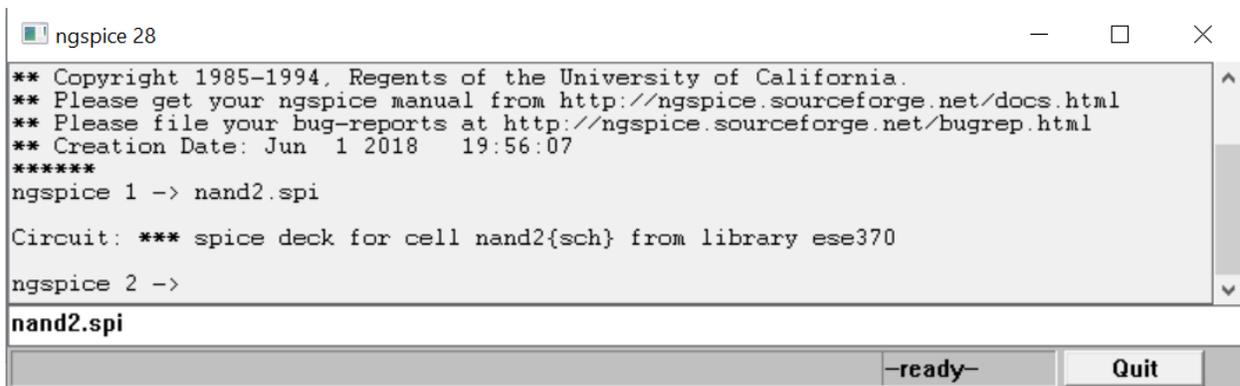
- bin
- doc
- examples
- lib
- share

Run the ngspice application also located in the **bin** folder



```
ngspice 28
Note: can't find init file.
*****
** ngspice-28 : 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: Jun  1 2018  19:56:07
*****
ngspice 1 ->
```

To load your circuit type the filename into the command line.



```
ngspice 28
** 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: Jun  1 2018  19:56:07
*****
ngspice 1 -> nand2.spi
Circuit: *** spice deck for cell nand2{sch} from library ese370
ngspice 2 ->
nand2.spi
```

Now you can run your analysis on the circuit.