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

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 the path to the 22nm_HP.pm file:

        /Users/taniak/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 technicallity.

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 mocmos and set its "Technology scale':" (at bottom) to 22 (for 22 nm).

Download and Install for Mac OS X

- First make sure you have XQuartz installed.
- Get it at: http://xquartz.macosforge.org/landing/
- Current release is: XQuartz 2.7.11
- Download XQuartz-2.7.11.dmg and install via drag and drop.

There are many ways to install ngspice, but it is highly dependent on your OS version. The methods can be found below as well as to the links for resources for help with installation. Method 1 has been tested on High Sierra with ngspice-27, and Method 2 has been tested on Mojave with ngspice-32 (latest version). For specific assistance use Piazza and office hours with instructor.

Three methods to install ngspice:

1. (Easiest way, binary package, might take some library setup)
- Go to: http://sourceforge.net/projects/ngspice/files/ng-spice-rework/
- Download and install: ngspice.pkg ← for latest version. You might need an older version

2. (Less easier way, compile from source)
- Install macports, see: http://www.macports.org/install.php
- Using the Mac OS X Package (.pkg) Installer is recommended.
- Once installed, in a terminal window, type: `sudo port install ngspice`
- This will install ngspice into the /opt directory.
- After this, ngspice will be available at the command prompt.
- Type `ngspice` to launch

3. (Allows very latest version install)
- Install XCode Tools if you haven't already.
- Go to: http://sourceforge.net/projects/ngspice/files/ng-spice-rework/
- Download ng-spice-rework-32.tar.gz to a working area.
- In your working area type: `tar zxvf ng-spice-rework-25.tar`
- Go into the newly created ngspice directory and type the following commands hitting enter after each command:
- `./configure`
- `./make`
- `sudo make install`
- when finished, ngspice will be installed in: /usr/local/bin/ngspice