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 mocmos and set its “Technology scale’:” (at bottom) to 22 (for 22 nm).

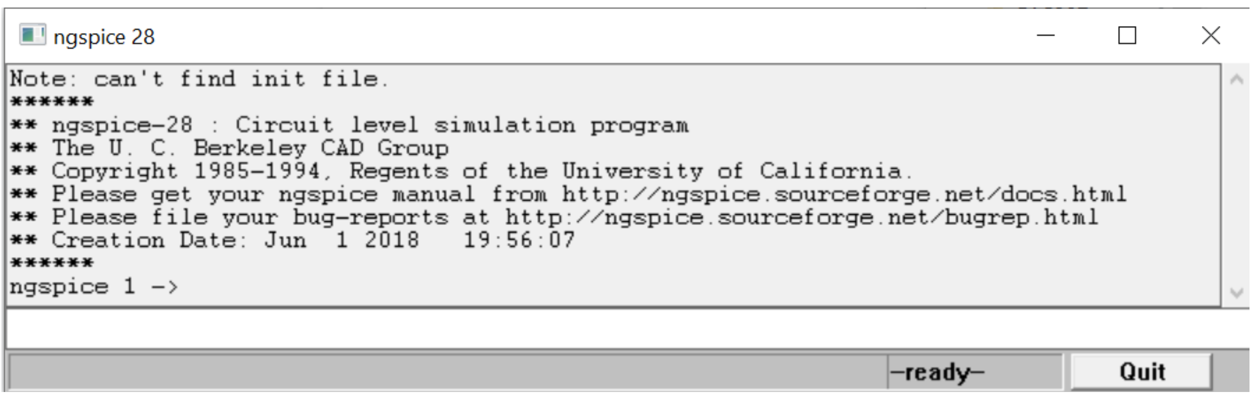
Download SPICE from http://ngspice.sourceforge.net/

Unzip the files.

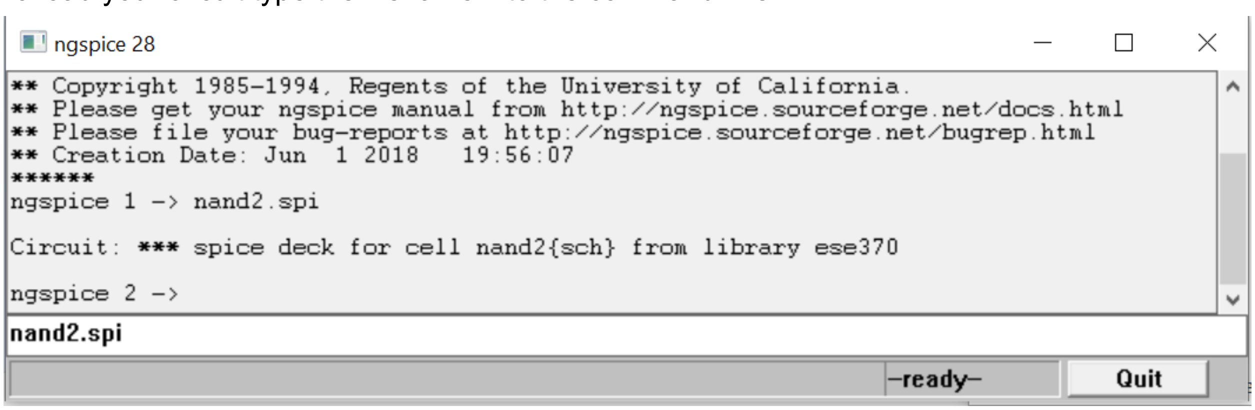
Main difference from using eniac
Save SPICE decks (.SPI) files into the **bin** folder from your extraction

- bin
- doc
- examples
- lib
- share

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



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



Now you can run your analysis on the circuit.