

COMPUTER ENGINEERING DEPARTMENT

COE 360 SPICE3 Notes

Instructor: Dr. Alaaeldin Amin

03 October, 1997

1. SPICE3 runs interactively and can only be run in an X-windows environment for watching the output waveforms.
2. To be able to run spice3 you should have `/usr/local/bin` specified in your path as defined in your ``.cshrc`` file (most probably you already do).
3. To view the UNIX manual page for spice, type: `man -M /tools/da/man spice`
4. SPICE3 has a good hierarchical help facility which can be invoked from within spice3 by typing: `help`
5. To simulate the input file `sample.cir` using spice3, type the following command in an X-terminal window::

```
spice3 sample.cir
```

6. It is recommended that the following line be included in your input simulation file:

```
.OPTINS LIST RELTOL=0.005 VNTOL=1E-4 ABSTOL=1E-10  
+ DEFL=0.8U
```

7. YOU should first watch the output waveforms to make sure that the outputs are correct before printing.
8. You should produce a printout of the input simulation file together with an ascii (or postscript) plot of the required node voltages and/or branch currents. To produce such an output you should run.
9. When running spice3 interactively, you may need to repeat the following commands from within spice3 before you get a correctly working circuit:

```
reset /* This command should precede each run command */  
run /* runs the simulation */  
plot v(1) v(11) I(VDD) /* Displays plots of the listed voltages and currents */  
edit /* Brings up the input simulation file in the editor window for the required edits */  
reset  
run  
plot <list of voltages and/or currents>
```
10. You can ZOOM in on various regions of your plots by specifying a window over this region using the right mouse button.
11. The output plots can be obtained in a normal ASCII file or you can store it into an eps file to be inserted in WORD document, LATEX document, etc.:

Commands for Getting an Encapsulated PostScript Picture of your PLOTS

- `set hcopydevtype=postscript`
- `Set curplottitle=`` Specify Tile ```
- `hardcopy filename.eps V(1) V(11),`
- `filename.eps` is the file name where the plots of `v(1) v(11)` waveforms will be stored

Commands for Getting an ASCII printout of your PLOTS

- `Set nobreak`
- `Set width=80`
- `Set curplottitle= `` Specify Tile ```
- `Asciiplot v(1) v(11)> filename`
- `filename` is the ASCII file name where the plots of `v(1) v(11)` waveforms will be stored.

12. In the directory where you will run simulations, you may use the a `.spiceinit` initialization file to define some of the parameters or command aliases.