Engineering 1620 – Spring 2011
SPICE Assignment Support Materials

NOTE: To retrieve the files referred to in this list, go to the Engineering 1620 web page and select "Course Materials". Right click the mouse on the hyperlink for an item and choose the "Save As" option, making your own copy of the material.

Sedra and Smith on SPICE Syntax: This is a scanned copy of an appendix from the 4th edition of Sedra and Smith, *Microelectronic Circuits* that discusses the syntax and construction of a SPICE file. It should be read in conjunction with the examples of such files on the CD that accompanies the present edition of the text.

Engn1620_base11.cir File: This is the complete SPICE file for the circuit fragment in the handout. It contains the models for the two types of MOSFETs included in the circuit. You can edit this file in WordPad or other text editor to add whatever you want in the way of additional circuitry or simulation commands. If you choose to use *Viewdraw* to augment the schematic, then you do not need this file as the extraction software, *SpiceLink*, will generate a new file with models included.

***Viewdraw* Schematic File**: This is the file (called En162_base04.1) that I used to draw the schematic in the handout. To use it, save a copy in the "sch" subdirectory of your Viewdraw working directory, which by default is normally U:\wv. You will need to link to the P:\Programs\MentorGraphics\MOSSETs library with the *Dashboard* tool (I think this library was included in the setup of *Viewdraw* for Engineering 1630 this past fall. If the MOSFET transistors are not there when you open the file in Viewdraw, follow the instructions under the "Viewdraw/SPICE Info".) All the parts you would want to put on the schematic are in that directory. You wire and assign attributes in the usual way. To generate a new SPICE file, use the pull-down menu "Tools/Create Analog Netlist". Your diagram can include symbols for the simulation commands too, although you may want to modify those by hand during runs rather than always going back to the schematic. There is even a way in the netlist tool by supplying the name of a configuration file on the "Advanced" tab of *SpiceLink* to tell the system to compute the AD, AS, … attributes automatically.

WinnowSPICE Program: This is a self-contained executable that opens a file dialog in which you locate the file with operating point data from a SPICE run. (You have to save an output log file from within SmartSpice beforehand.) The program reads that file and creates a new, shorter file with only a subset of the most useful transistor small signal parameters. That file is stored in the original directory of the Log file with the same base name and with the extension ".wnw." This file is particularly handy to paste into EXCEL for sorting and tabulating small signal device paramters.

***Viewdraw*/SPICE Info Document:** This is a short set of notes and hints on setting up a SPICE deck both by hand and from the Viewdraw output. It has more explicit information on fixing problems with making these two programs talk to each other. It also has some information on the peculiarities of the version of SPICE that we use, namely SmartSpice from Silvaco Corp.