Hints and Explanations for Using Viewdraw to Prepare SPICE files and for Running SmartSpice™ on the Engineering Server

1. You have two choices for generating a SPICE deck for this project. The first is to download and edit the baseline schematic from the class web site and use Viewdraw to generate the file. This is recommended for anyone who has had Engineering 163 and therefore is familiar with the Viewdraw tool. The other way is to download the SPICE file for the differential stage that is also on the class web site. Edit that by hand to add the necessary functionality.

2. The general procedure for schematic entry is:

   - Download the base schematic to your U:\Wv\sch directory (or other working directory if have changed the default Viewdraw directories).
   - Edit the schematic with Viewdraw to add the additional transistors and current source; to correct all transistor sizes; and to edit the "AC Analysis" symbol to set the frequency parameters for that calculation. Device sizes, areas, etc. are symbol attributes that you change by double clicking on the symbol and entering values in a dialog screen.
   - From the Viewdraw menu run "Tools/Create Analog Netlist". This opens a dialog screen. On the Advanced tab, choose "Use net names" and see that the "PROBE Statements" option is selected. Hit the "Create Netlist" button and watch for errors that need correcting.
   - You will find a ".cir" file in your working directory if this program runs correctly. Open that with WORDPAD or another editor and check two things: 1. if necessary delete any "$" symbols – Viewdraw prefixes global signals such as VDD with the dollar sign but SPICE will not accept that symbol; 2. check that the line ".PROBE/CSDF" appears in the file. If it does not, edit it in. (This line causes SPICE to store all nodal voltages and branch currents. Makes using the data easier.)

3. If when you open the differential amplifier schematic in Viewdraw, there are no MOSFET symbols on the schematic, it probably means that your Viewdraw library list does not link to the right symbol library. Add the following line to the list of library files at the end of the Viewdraw.ini file in your Viewdraw working directory (probably U:\Wv\Viewdraw.ini).

   ```
   DIR [r] P:\ePD\Mossets (Mossets)
   ```

4. Should you choose to create the SPICE file by editing the differential amplifier deck that is on the class web sit, then use the general structure of a SPICE file that looks like:

```
* SPICE Assignment for EN162.
* THE FIRST LINE MUST BE A COMMENT!!

* (DECK WITH TRANSISTOR INTERCONNECTIONS GOES HERE)

* Differential Gain Input Excitation:
* VC is the common mode DC voltage; VCAC is for the AC common mode gain
* VINAC is the AC differential input voltage
* VOFF is a DC offset at inputs that may be needed to get the right quiescent
```

```
* output
* EV1 and EV2 are voltage controlled sources that make the differential signals
* for the non-inverting (VNON) and inverting (VINV) inputs

EV1 VNON  VC1 VD 0 .5
EV2 VINV VC VD 0 .5
VC VCACLOW 0 DC 1.0
VCAC VC VCACLOW AC 0.0
VINAC VD 0 AC 1
VOFF VC VC1 DC 0.0

* These lines specify the operations done by SPICE, AC analysis, operating
* point determination, and saving all node and current information
.OP
.AC DEC 10 10HZ 100MHZ
.PROBE/CSDF

* Transistor models from the assignment base file.  Be sure the model names
* match the names in the interconnection list.

.MODEL …..

* SPICE files must end with an “.END” statement
.END


* To measure Common-Mode Gain in a hand calculation, edit the AC source lines
VC VCACLOW 0 DC 0 AC 1.0
VINAC VD 0 AC 0.0
* This should do it with the same stuff as before,
* just with the input changed by superposition.



* Offset Voltage: Sweep "VOFF" if needed to find a value for which the quiescent
* output is within a reasonable range (1.0 to 2.0 volts)  Add these lines:
.DC VOFF –0.002 0.002 .00005
.MEASURE DC VOFFSET WHEN V(OUT)=1.55
.PRINT DC VOFF
```

5. The basic steps for running SmartSpice are:

    - From the Windows start menu choose “Start/Programs/Electrical/Silvaco/SmartSpice”
    - The SmartSpice program will open and from the menu “Files/Open” browse to your “.cir” file
    - From menu Analysis/ choose “Run”. (If you wish to see the small signal parameters from the “.OP” analysis, use “Run Batchprint” instead.)
    - From menu Display/ choose “Vectors” to open a dialog in which you can do printing and plotting of results. The procedure is fairly obvious: you pick a result – “id(m1)” or “vout” – from the vectors list and hit a softbutton to list, print, or plot the value.

6. Plotting or printing functions of outputs in SmartSpice:

    - If you want to print or plot a function of one of the outputs, you have a choice of two methods. You can include a “.LET” command in your

SPICE file defining the quantity or you can use the Vectors/Display dialog to set up such a calculation. Once the calculation is made, the result appears as an entry in the results list and can be plotted, listed, etc. the same way as any other result.

- There is some advantage to using the Vectors/Display dialog for entry since SmartSpice does not always execute two ".LET" commands properly. At the bottom of the Vectors/Display dialog there is a function window in which you can type a definition for a function. Two examples: "gain=db(vout)" will form a result called "gain" that forms the magnitude of vout in decibels relative to a 1 volt reference; and "phase = 57.3*ph(vout)" which will report the phase angle of vout. (The factor 57.3 converts radians to degrees.) After typing an equation, hit "ADD" to put it in the result list. Thereafter you can print/plot by picking that result and the appropriate softbutton.