ELEC 2210 EXPERIMENT 9
MOSFETs in PSPICE

Objectives:

The experiments in this laboratory exercise will provide an introduction to simulating MOSFET circuits using PSPICE. The objectives of this experiment include:

- Review basic principles of MOSFETs from ELEC 2210
- Become familiar with PSPICE for circuit simulation
- Continue to develop professional lab skills and written communication skills.

Introduction

A thorough treatment of MOSFETs can be found in Chapter 4 of the ELEC 2210 textbook, Microelectronics Circuit Design by R.C. Jaeger. This text and the associated website also provide some PSPICE-based examples. Further PSPICE support and examples are provided at this link:

http://www.eng.auburn.edu/~troppel/pspice_links.html

In a previous experiment, we learned that a MOSFET is often used as a voltage-controlled switch, as illustrated in Fig. 1.

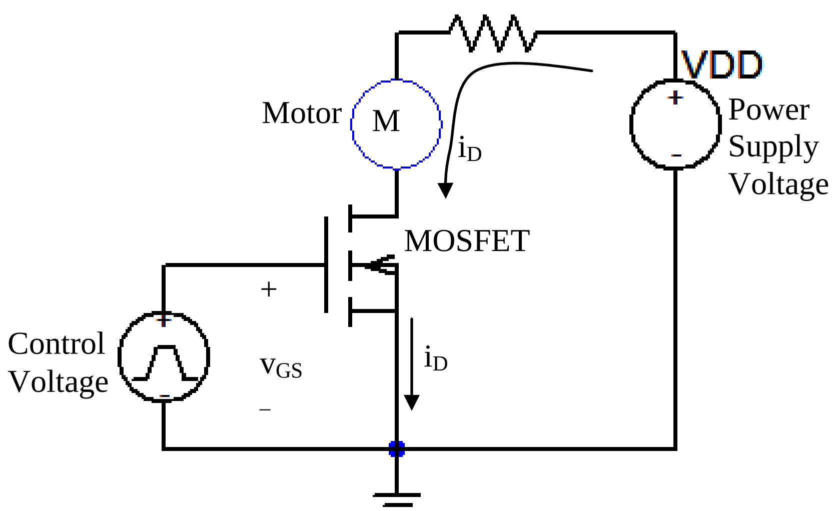

Figure 1. MOSFET switching circuit. When the control voltage exceeds the threshold voltage, the MOSFET is “ON” and current flows through the motor. Otherwise, the MOSFET is “OFF,” and no current flows.

The amount of current which flows is determined by the control voltage. For most switching applications, the MOSFET is operated in the triode region when it is conducting current. In this region, the MOSFET channel presents a small resistance in series with the load, as desired. In order to turn off, the MOSFET is operated in cutoff.

Pre-Lab (not for submission):

- Review Experiment 7, especially the MOSFET regions of operation.
- Download and install PSPICE if you will be using your own computer. http://www.eng.auburn.edu/~troppel/pspice_links.html
- Obtain the data sheet for the IRF150 power MOSFET. You can get this from the class website or the web.

Lab Exercise: (See submission instructions at the end of the writeup.)

(1) *Threshold Voltage*

Use the circuit shown in Fig. 2 to determine the threshold voltage of the IRF150. On the data sheet, the threshold voltage is defined to be the value of V_{GS} when I_D = 250 μA, with the drain connected to the gate.

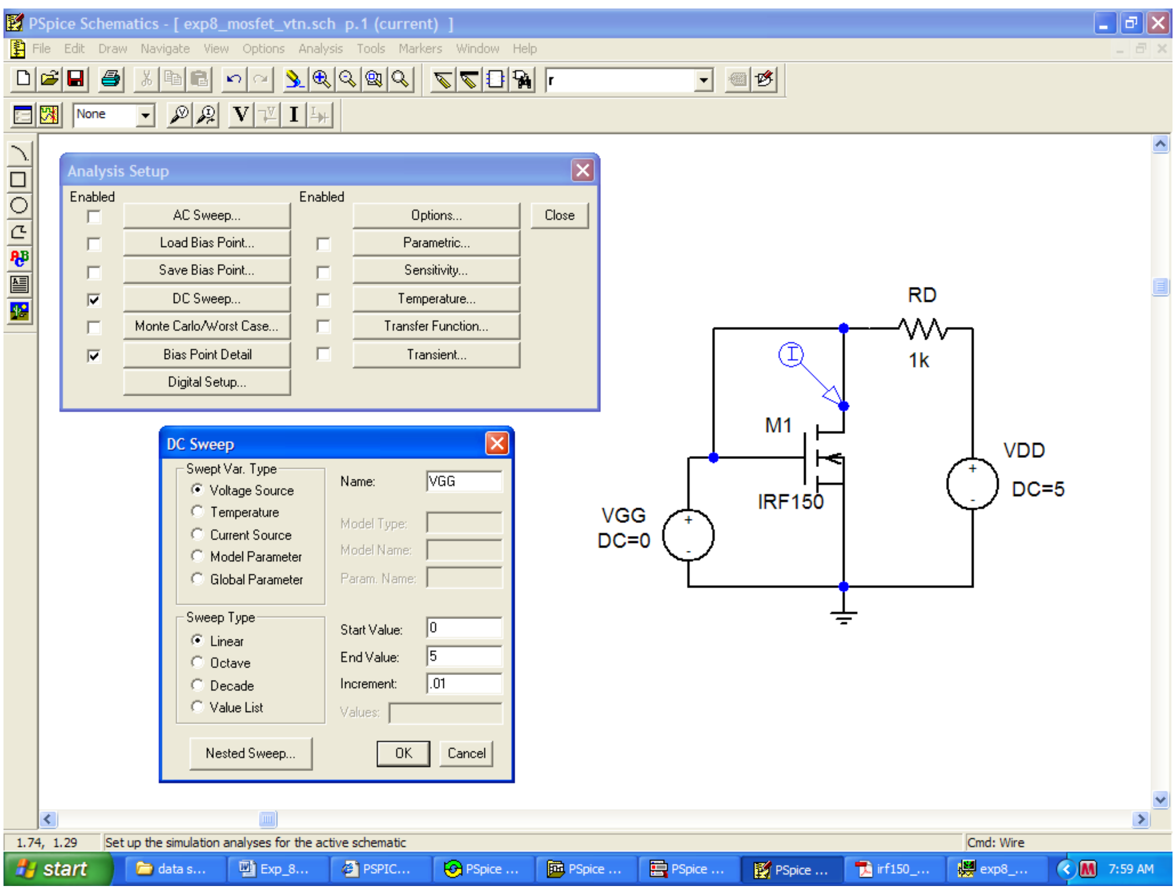

Figure 2. Simulation setup for measuring the threshold voltage with PSPICE.

Measure the value of V_{TN} as accurately as possible, rescaling the plot as necessary and using the cursor. Does your measured value of V_{TN} fall within the range specified on the data sheet? Also compare with the value listed in the model parameters for the IRF150. The procedure for finding this is described in Fig. 3.

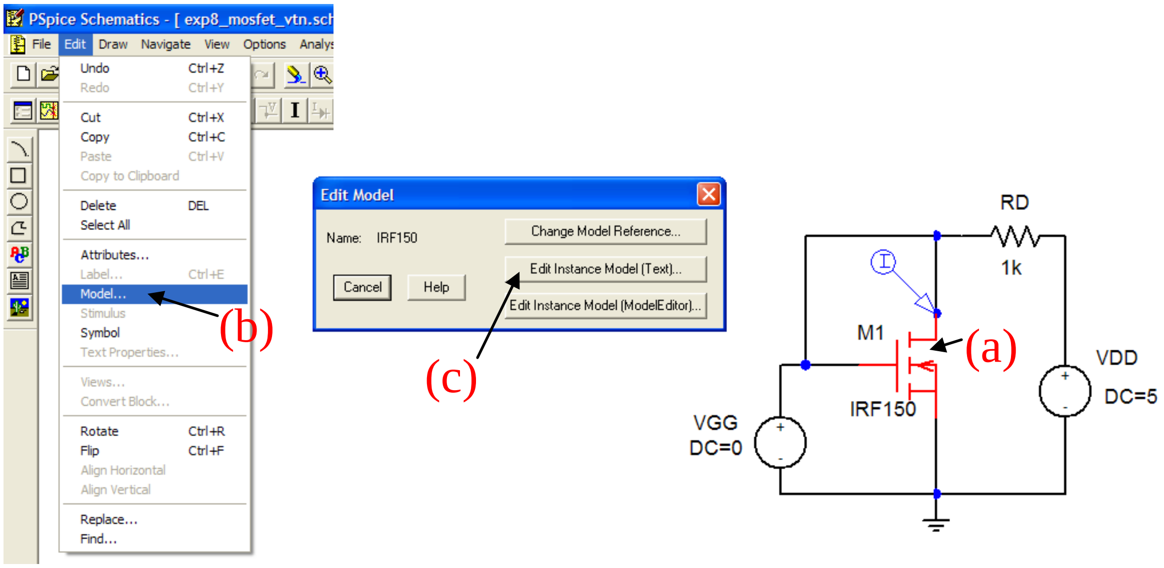

Figure 3. Showing the MOSFET parameters. (a) Click on the part, (b) select Edit / Model to get the dialog box shown, (c) click on the middle button "Edit Instance Model (Text)", (d) find the line which gives the value of Vto. This is the threshold voltage parameter.

(2) *I_D vs. V_{DS} and Estimating K_n*

In this part, you will use the PSPICE to trace I_D as a function of V_{DS} for several values of V_{GS}. From your observations, you will estimate the value of K_n for your MOSFET.

Construct the circuit shown in Fig. 4. Use the nested sweep capability of PSPICE to sweep VDD from 0 to 20 V in .01 V steps (main sweep) and VGS from 0 to 10 V in 1 V steps (nested sweep). To set up the nested sweep, click on the "Nested Sweep" button in the DC Sweep dialog box.

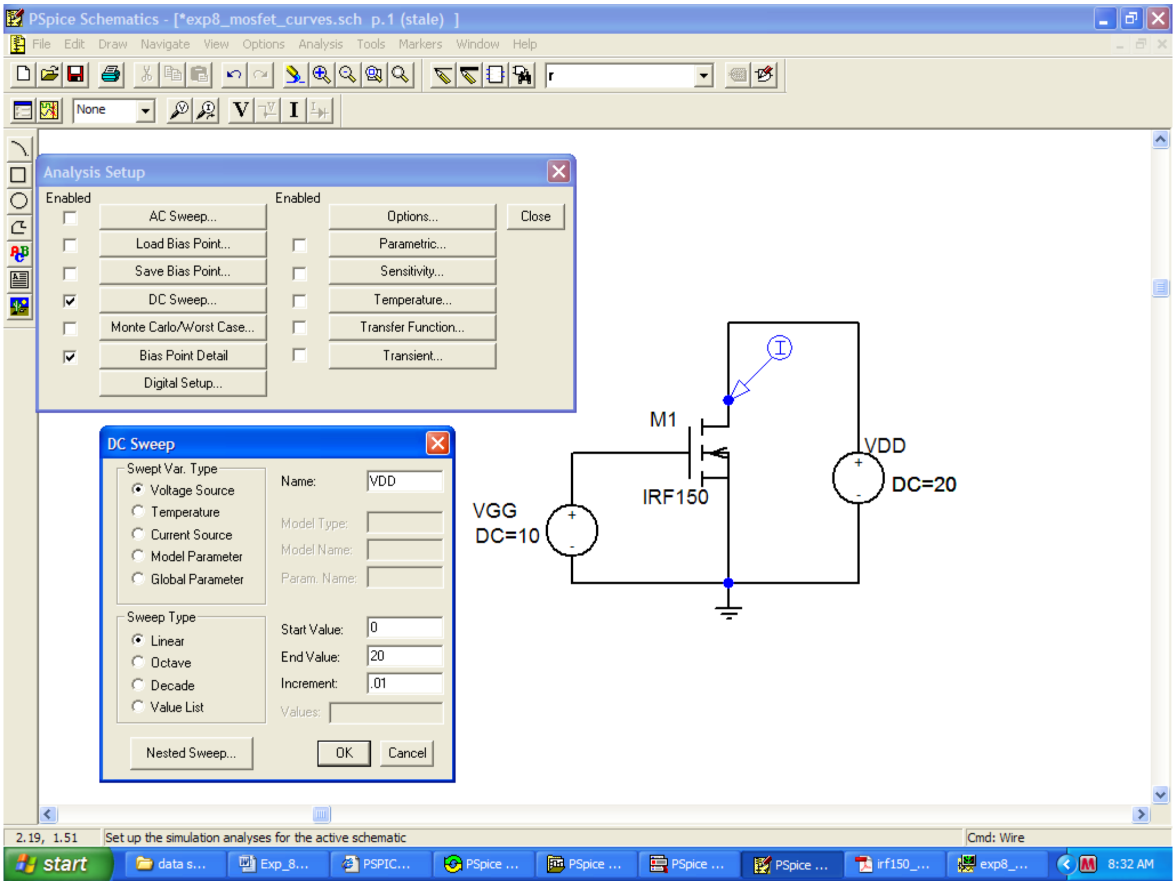

Figure 4. Circuit for plotting I_D vs. V_{DS} and determining K_n. To set up the nested sweep, click on the "Nested Sweep" button in the DC Sweep dialog box.

The resulting family of curves, called the output characteristics, is shown in Fig. 5. Use this result to determine the value of K_n. Calculate the values of K_n based on each of the curves for V_{GS} = 5, 6, 7, 8, 9, and 10 V. Use a similar procedure to the one you used in lab, namely, measure the current in saturation using the cursor, and use the saturation region equation for the MOSFET together with the threshold voltage you found in Part (1).

Make a table of the values of K_n. Are they consistent? What is the average value? What is the percent difference between the lowest & highest values? Calculate this as:

$$\frac{highest - lowest}{average} \times 100\%$$

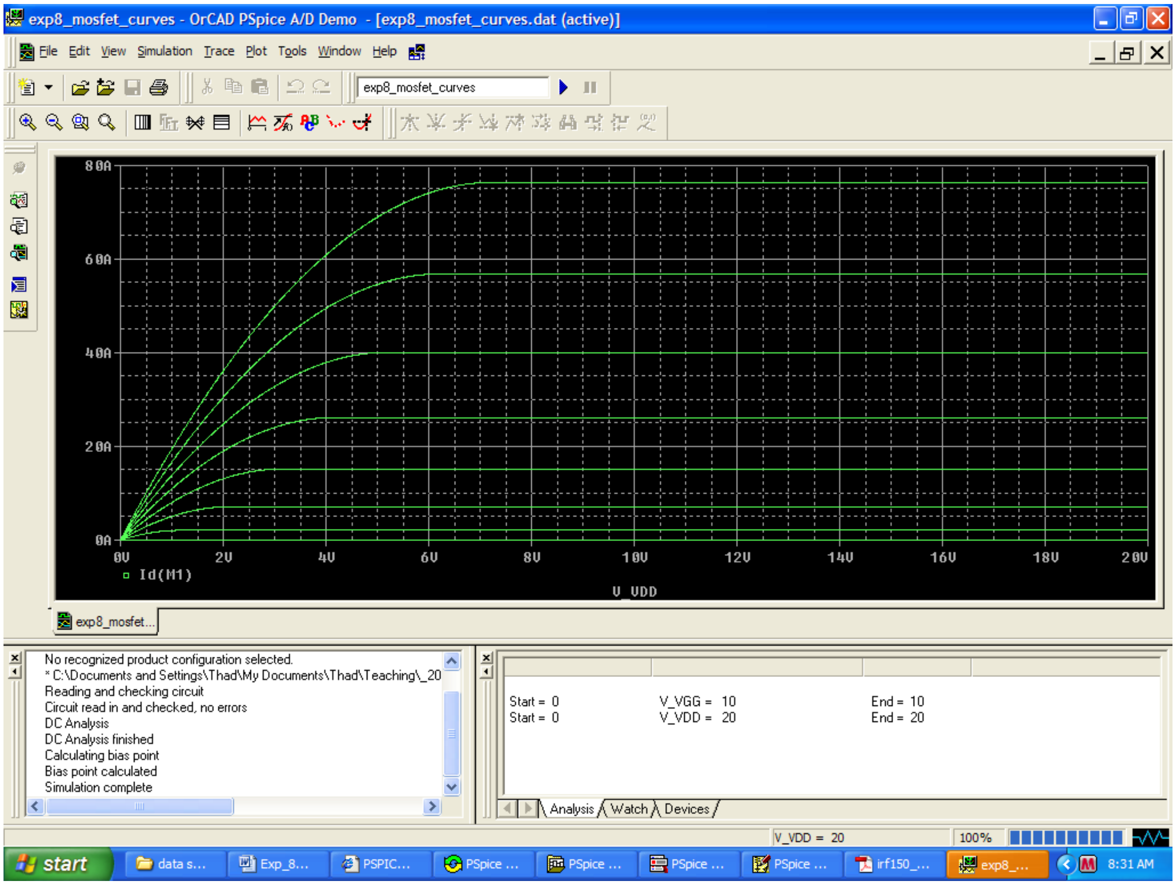

Figure 5. Plot of the output characteristics, I_D vs. V_{DS}, for the IRF150. The topmost curve corresponds to the largest value of V_{GS}, which is 10 V in this case. The next curve down is for V_{GS} = 9 V, etc.

(3) *MOSFET Switching Circuits and ON Resistance (R_{DS}).*

Construct the circuit shown in Fig. 6.

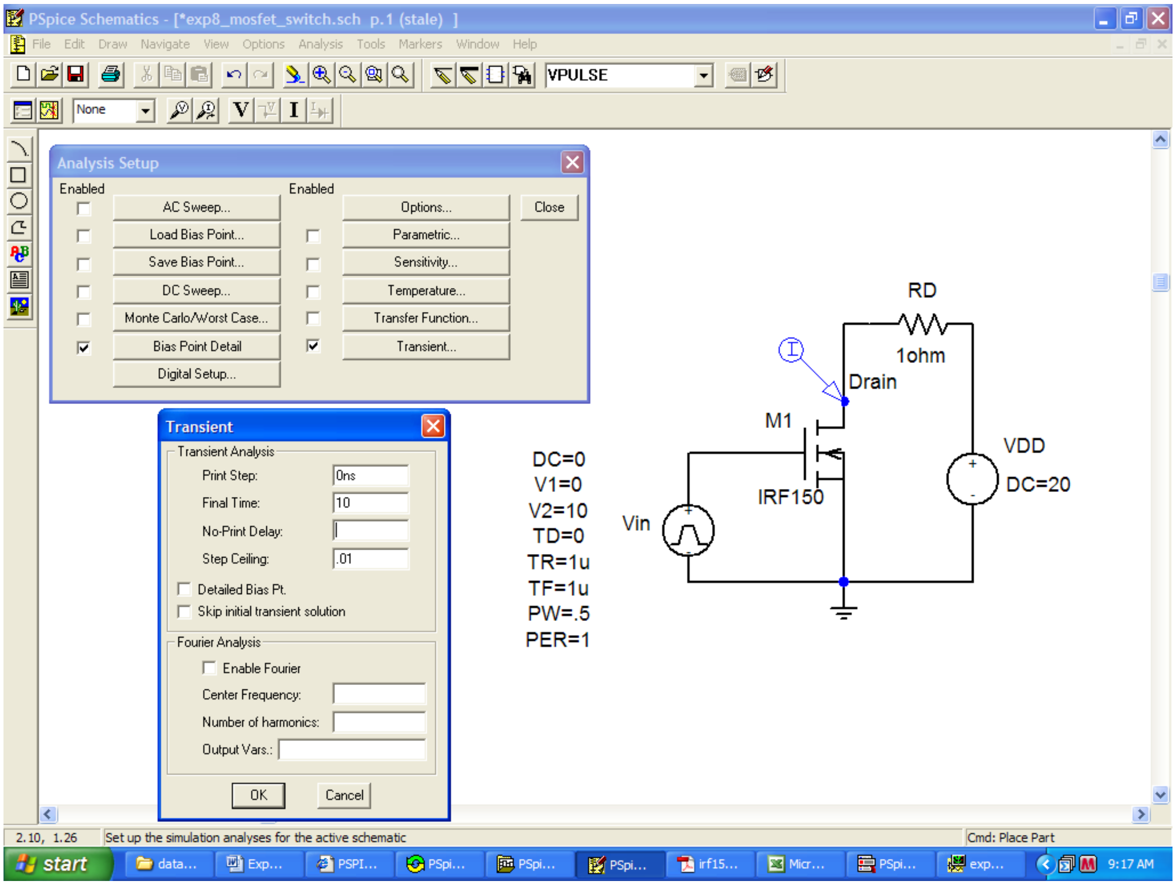

Figure 6. Switching circuit. The gate voltage is provided by a VPULSE part. The parameters are shown. The pulse width (PW) and period (PER) are 0.5 and 1 second, respectively. The rise and fall times (TR and TF) are 1 microsecond. The pulse goes between V1 and V2 (0 and 10 V).

When we run this simulation, we want to display the current on one plot. The voltages Vin and V_{DS} will be shown together on a separate plot. In PROBE, you can add a plot to the window as shown in Fig. 7.

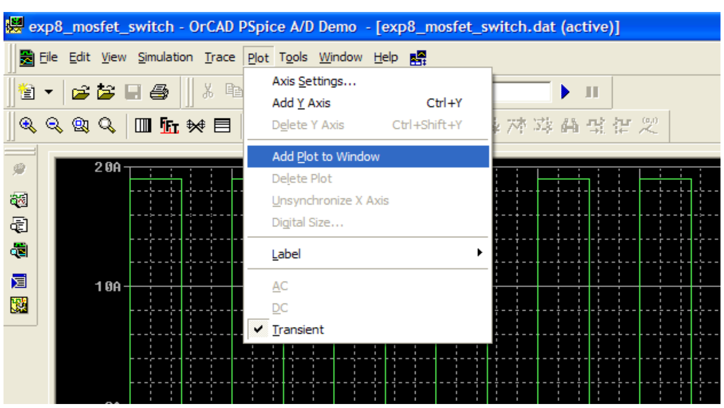

Figure 7. Adding a plot to the PROBE window. It is usually best to show currents on a separate plot from voltages.

If you use this technique together with some of the annotation capabilities in PROBE, you can get a window like the one shown in Fig. 8.

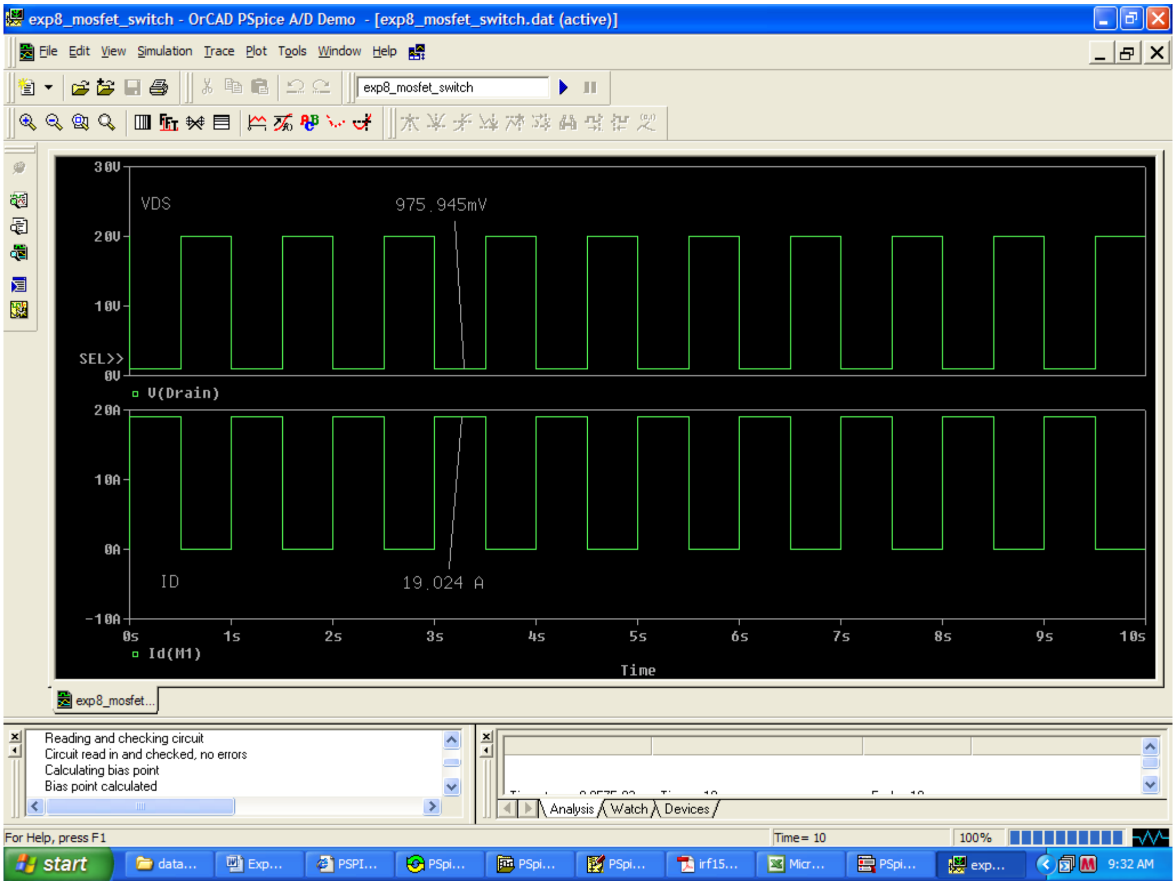

Figure 8. Window showing V_{DS} and I_D for the MOSFET. The *x* and *y* gridlines have been turned off to unclutter the view. The cursor and Plot/Label/Mark features have been used to indicate the minimum value of V_{DS} and the maximum value of current. These values can be used to calculate R_{DS}, the on resistance of the MOSFET.

When the MOSFET is on, it is like a closed switch with a certain amount of resistance. This on resistance is called R_{DS} on the data sheet. During the time that the MOSFET is on, the equivalent circuit shown in Fig. 9 pertains.

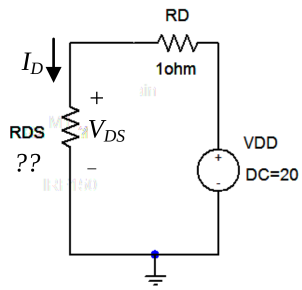

Figure 9. Equivalent circuit for Fig. 6 when the MOSFET is in the on state.

When the MOSFET is on, maximum current will flow, and V_{DS} will be at its minimum value.

Using this knowledge, calculate R_{DS} from your simulation results (Fig. 8). What value of R_{DS} is specified on the data sheet? Does your calculated value fall within the specified range?

Preparing and submitting your report:

Prepare your report in electronic form and save it for the remainder of this semester, but submit it in hardcopy. Your GTA or professor may ask you at a later time to email or upload your electronic version if there are questions about the hardcopy submission.

For each part, provide the following, in this order:

(a) Circuit diagram from Schematics showing parameter values, similar to those shown in this writeup.
(b) Output file (.out), condensed with extraneous blank lines and duplicate headers removed. Be sure to keep the original formatting (Courier font or equivalent) so that table columns line up, etc.
(c) Probe plot(s), suitably annotated to convey your measurements clearly. Part of your job is to make it as easy as possible for the reader to understand your results.
(d) Hand calculations and discussion as requested in this writeup.

Separate your report into three parts, corresponding to the three parts of this writeup. Start each part on a new page.