Working with MOSFETs in ORCAD/PSpice (student edition)

This document has been written to help students in EE252 adequately simulate MOSFET devices in ORCAD/PSpice, one of the primary tools used for circuit simulation in the course. The problem is that the "Student edition" of the software (which students may be running on their own computers or in the lab) does not include the 2N7000 MOSFET transistor. The "EVAL" library includes an assortment of semiconductor products, including the 2N2222 and 2N2907 (electrically equivalent to the PN2222 and PN2907) that we will use in the lab. But no 2N7000. What to do? That's why this document.

There is a "BREAKOUT" library that contains an assortment of "generic" components that can be adapted as needed. The library includes a generic NMOSFET, called "NBREAKN". That can be used instead of the 2N700, but it needs to be modified. The circuit shown in Figure 1 was built using the NBREAKN with Q point data. Figure 2 shows the AC transient response.

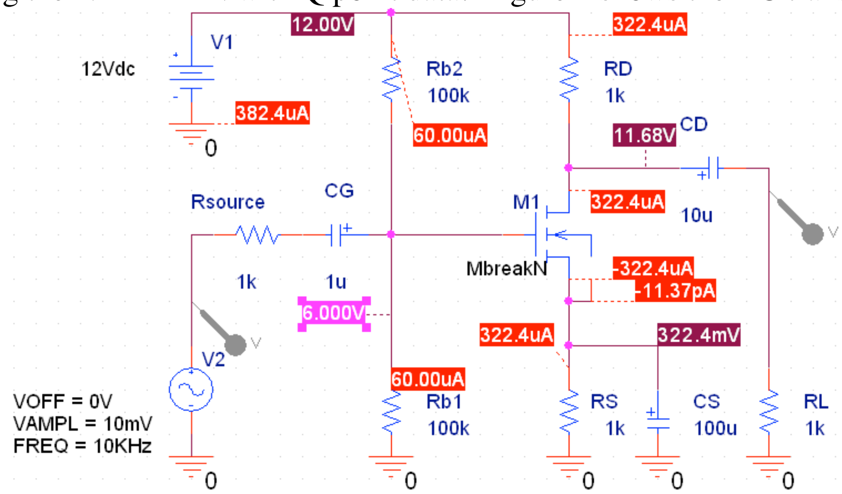

Figure 1 MOSFET circuit using unmodified NBREAKN part

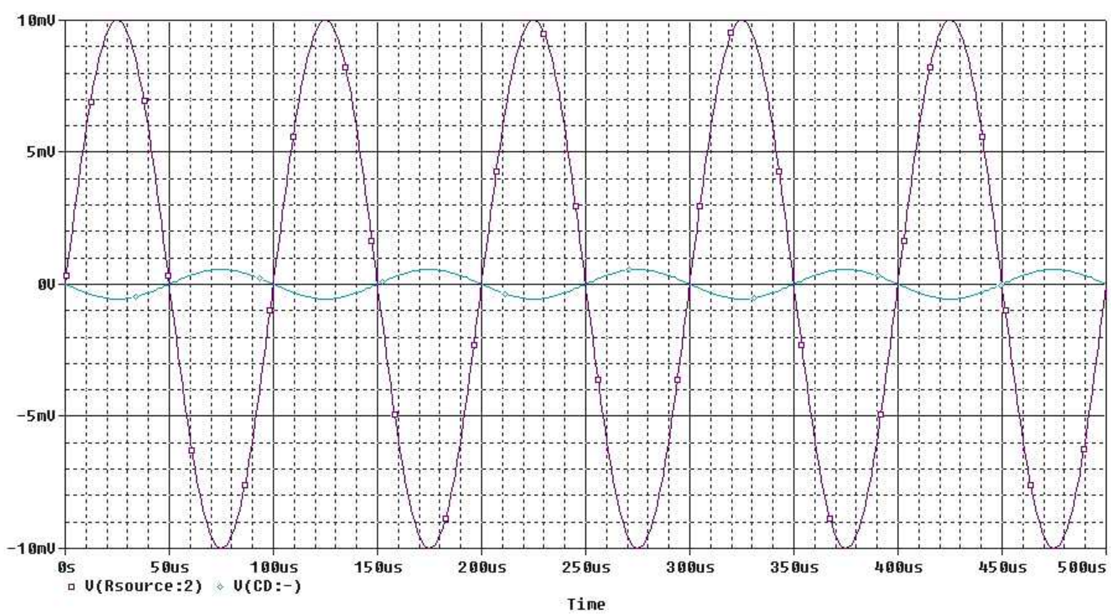

Figure 2 Transient response

The first thing to notice is that V_{GS} is 5.7 Volts! This thing obviously does not have a low V_{TN}. The 2N7000 has a nominal V_{TN} of 2.1 Volts ($V_{GS(th)}$ on the data sheet). So, we can't use this part as-is. K_n is probably also way off. Because of the low collector current, there's not much amplification, as is apparent in Figure 2: 10 nV peak ! (I don't understand the input trace. The input Voltage is specified for the VSIN source at 10mV. Much bigger than the output.) The problem is that our MOSFET does not match the 2N7000. (But, we don't have an error.)

To fix the problem, we need to adjust the PSpice model of the MOSFET. Select the MOSFET (M1) and then, on the edit menu, select "Edit PSpice Model". What we want to put into the model are values of K_n and V_{TN} appropriate to the 2N7000. The edit box is shown, as modified, as Figure 3. You need to know what to call the parameters you want to change. K_n is actually "Kp" and V_{TN} is actually "Vto". (To find the desired value for K_n, find in the data sheet a specification for gm (transconductance) at a particular I_D, then solve for K_n knowing that (equation 4.8(b) in the textbook).

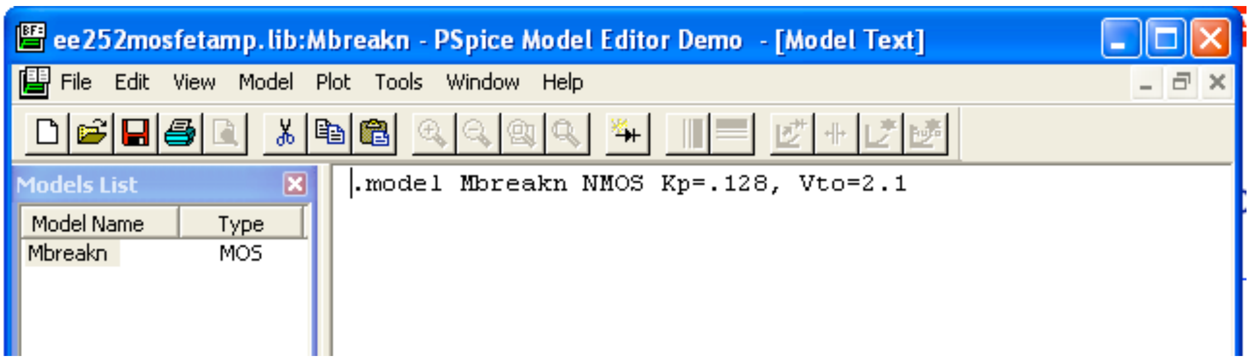

Figure 3 Editor for MOSFET PSPice Model

How do you know what to call the various parameters? Fortunately, we have in the EVAL library one lonely MOSFET, the IRF9140. You can plop one down then select it and bring up its PSPice model, and see what various parameters are labeled. If later you want to include Lambda and parasitic capacitances, you can look at that model and see what they are called. It is shown in Figure 4 below:

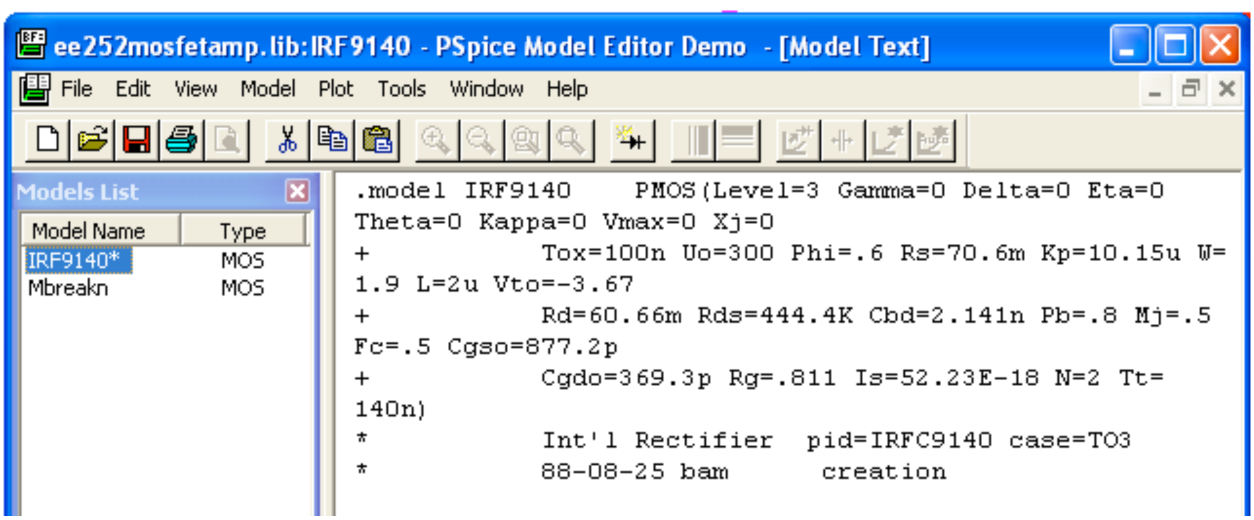

Figure 4 Example MOSFET model fully populated: the IRF9140

Now, having modified the model, the simulation is re-run giving the Q point shown in Figure 5 and the transient response of Figure 6. Looks a lot better. If we were expecting VD at 8 Volts, we are pretty close to that. Our gain is now a factor of 15; much better!

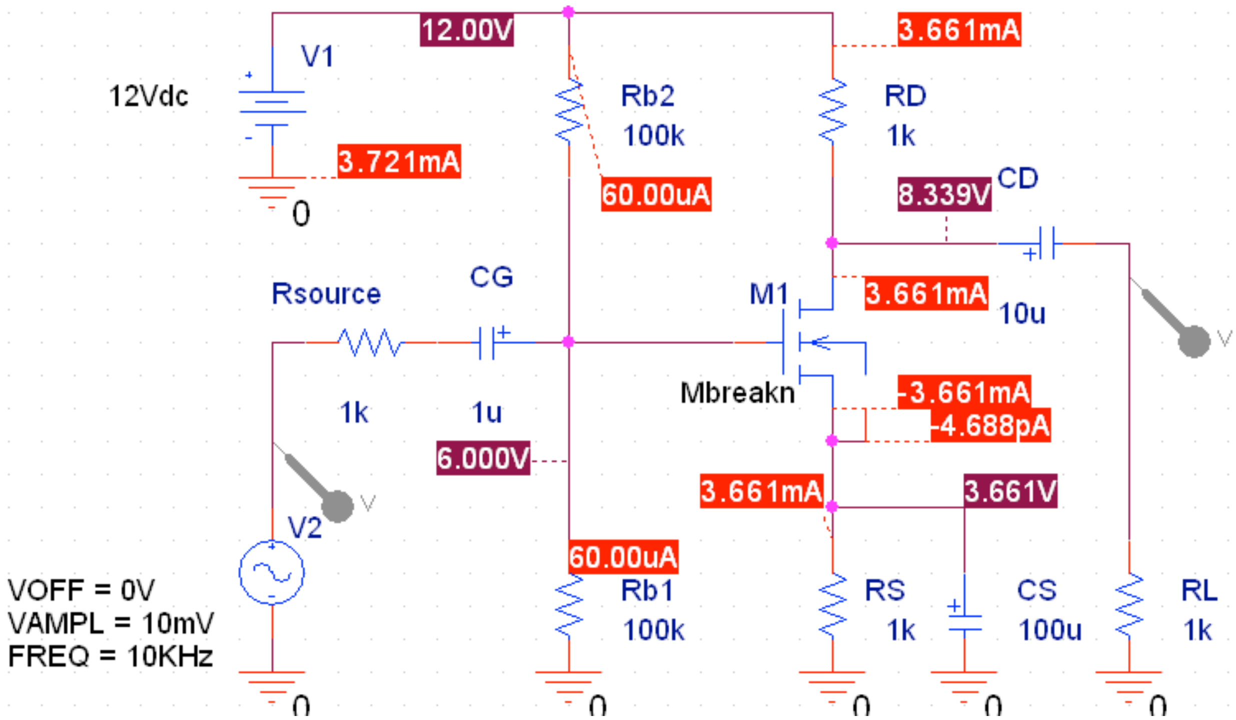

Figure 5 Q Point in MOSFET circuit with modified MBREAKN model

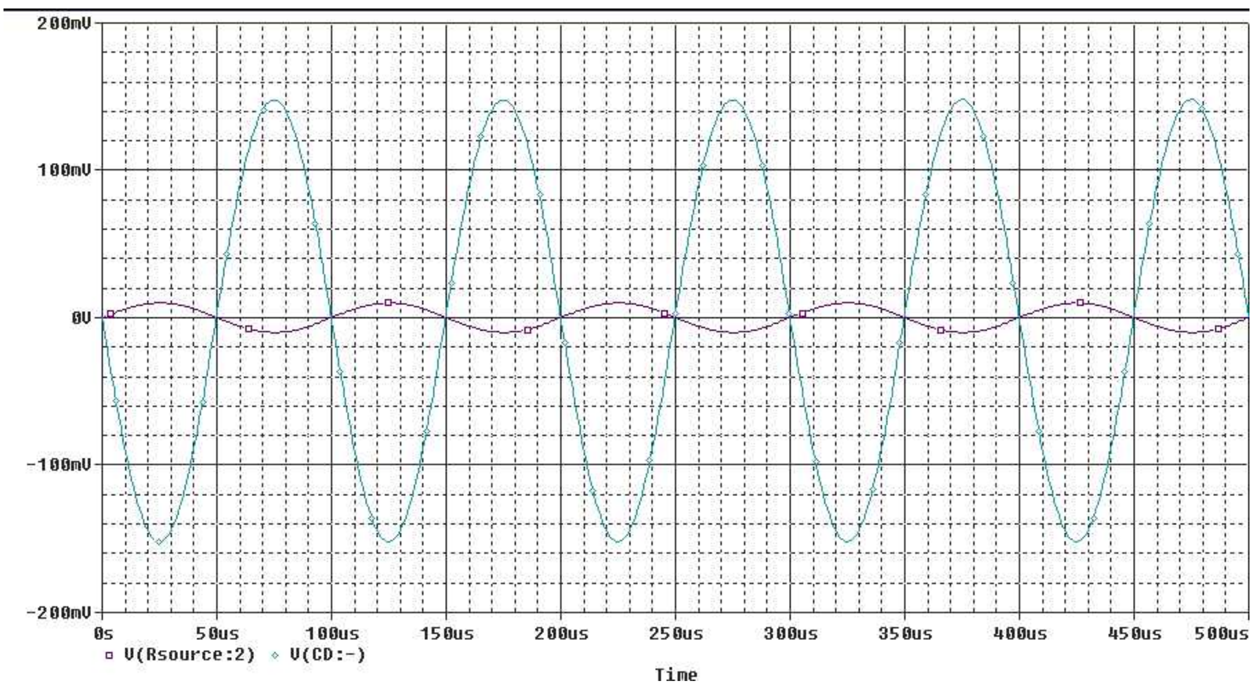

Figure 6 Transient Response with modified MBREAKN model