This document is intended to provide you with experience with MOSFET circuit simulation via PSpice.

A complete circuit is provided here and you are asked only to design this exactly as below and then to familiarize yourself with specific characteristics.

First, here is the circuit, a simple common source amplifier.

- Note that the device is an NMOS transistor. For this device, the NMOS Body terminal and source terminal are connected as shown. The arrow point from right to left indicates that this is NMOS and that further this arrow indicates the p-type Body terminal (in which our n-channel will form). This Body terminal is grounded.

- Now, the MOSFET devices we introduce into our circuits are the MbreakN4 and MbreakP4 elements in the BREAKOUT library (four terminal devices including drain, gate, source and body.)

- We must access and edit the PSpice Device Mode.

- The MOSFET transistors will contain this typical Model Statement in the Model Editor for transistor parameters of
  - \( (k_n/2) = 100 \),
  - \( V_A = \lambda^{-1} = 100V \),
  - \( W = 200 \) microns
  - \( L = 0.2 \) microns
- Threshold Voltage = 0.4V for NMOS
- Threshold Voltage = -0.4V for PMOS

For the PMOS devices:
```
.model Mbreakp PMOS VTO=-0.4 KP=100u LAMBDA=-0.01 W=200u L=0.2u
```

For the NMOS device:
```
.model Mbreakn NMOS VTO=0.4 KP=100u LAMBDA=0.01 KP=100u W=200u L=0.2u
```

- Please be very careful with units and signs. If the unit “u” is not included, then PSpice assumes a gate width or length in meters, not microns!
- To access the Model Editor highlight the device in question by left-clicking on the device. As shown below, this will highlight the device.

Then, navigate to Edit > Model (or right click on the device and select Edit PSpice Model from the drop-down menu).
- This will bring up the window below.
• By default, the parameters are not set to our desired values. For example, the editor will show the model name and device type, and no other parameters.

• Enter our preferred values:

• Now, execute File > Save and your device has been updated.
Now, this system offers an opportunity to test bias analysis (by observing terminal voltages and current in M1).

This also offers an opportunity to examine gain by using the Trace tool.

The PSpice transient analysis Simulation Profiled configuration is shown here:

You should observe an output at the Drain (amplifier output) that appears similar to this:
Please make the following adjustments and observe the results that will be discussed below.

- Adjust the value of Rs to cause the transistor bias point to fall into the triode region. What value of Rs causes this to occur?

- Also, starting with the circuit values as given above, reduce the value of W from 100u to 10u (100 to 10 microns).

- Then, readjust the value of Rs to return the system to a Drain voltage value of 7 +/- 0.25V.

- What gain now appears?

- You are encouraged to make many adjustments and to experiment with this amplifier.

After this step of familiarization, we can then turn to an MOSFET circuit design and analysis problem.
WORKAROUND FOR SEASNET PSpICE FILE ACCESS ERROR

• The Problem:

Now, on a single user machine, for example, your home machine, running either PSpice 9.1 or 9.2, this error will not be seen. However, for the PSpice 9.2 version running in the shared SEASnet environment, you may see an error when attempting to save the Model Editor Library file.

• It is most convenient and recommended to use your home version of 9.1 or 9.2.

• However, for SEASnet users, an error related to file access permissions appears. The following is a resolution to the problem.

• First, as described in the Model Editor Tutorial, access the device in question by left-clicking on the device. As shown below, this will highlight the device.

For PSpice 9.2, we edit the model by left clicking on the transistor, then right clicking to bring up the menu shown. Select Edit PSpice Model.
• This will bring up the window below.

![PSpice Model Editor](image1)

• This is the PSpice Model Editor

• Now, on your single user machine, at this point you would simply navigate to File>Save. Then, you can return to your circuit to simulate.

• Now, on SEASnet, follow this procedure

• Navigate to File > Save As

• Select a file name other than the one suggested by PSpice. For example, see below where the user is updating the file name – by adding “_3” to the filename. Note that the extension remains as “.lib”

![Save As window](image2)

• Now, there is one more step to make before you can continue

• This is to open up the Simulation Profile editor.

• So, navigate to PSpice then to Edit Simulation Profile.
• This will bring up this window

![Image of Simulation Settings - CMOS Amplifier window]

• Now, click on the Libraries tab, this will display this interface:

![Image of Simulation Settings - CMOS Amplifier Libraries window]

• Now, you must delete the files that include your project name and are not the file that you just edited. For example here, we remove “cmos_amplifier.lib”. This is removed by clicking on the “X” button,

• Do not delete the “nom.lib” file.

• Now, close the Simulation Profile Editor and proceed with your simulation.

• Unfortunately, you will have to follow these steps each time you edit your transistor when using SEASnet.