Cadence Tools: *Design Example #1b:*
Estimating Lambda using Variable Sweep & Waveform Calculator

This tutorial continues from Example #1a with estimation of the channel-length modulation parameter LAMBDA from the BSIM3v3 model simulations. From the simplified drain current relationship:

$$I_D = K(V_{GS} - V_t)^2 (1 + \lambda V_{DS}) \quad \text{Eq \#1}$$

We can solve for LAMBDA as a function of drain-source current and voltage as:

$$\lambda = \frac{\dfrac{\partial I_D}{\partial V_{DS}}}{I_D - \left(\dfrac{\partial I_D}{\partial V_{DS}}\right) V_{DS}} \quad \text{Eq \#2, or more approximately as: } \lambda \approx \frac{1}{I_D} \frac{\Delta I_D}{\Delta V_{DS}} \quad \text{Eq \#3}$$

In this example, we will sweep Vds on n & p-type devices, then estimate LAMBDA from plots of the drain currents.

Tutorial Steps:

1. 1. Following similar procedures to Example #1a, create the schematic **schex1b** in your **ecen5007** library using **nmos4**, **pmos4**, **vsource** and **gnd** cells from the **analogLib** library:
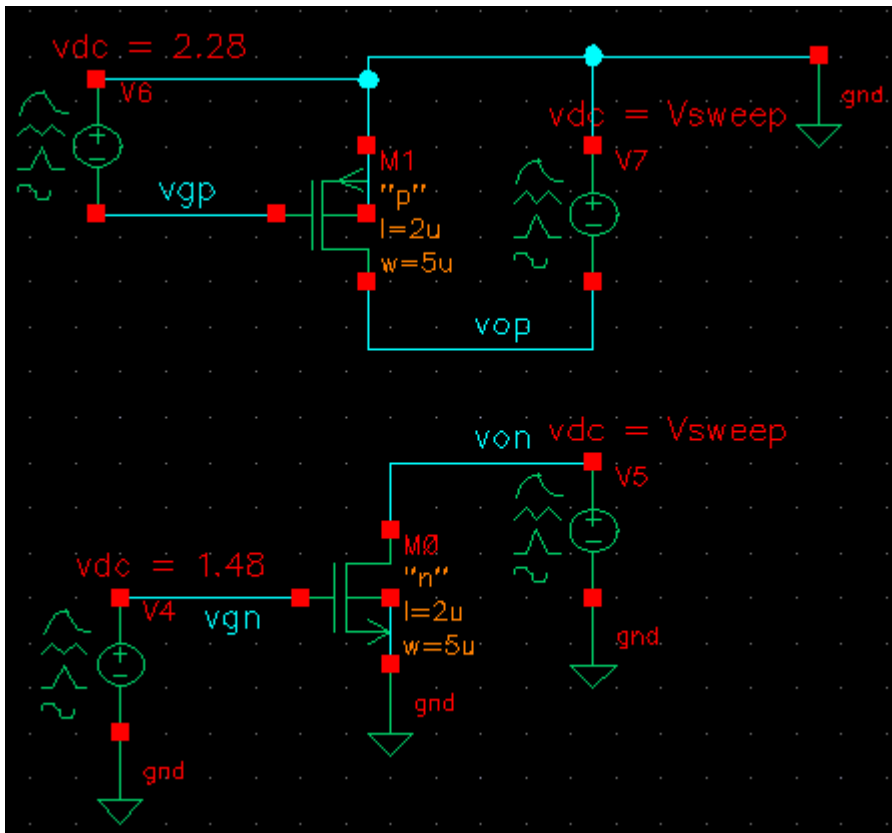
a. a. Set all sources to **vdc** with the voltages as shown above, where V4 & V6 voltages were selected to give Id = 50uA, and the drain-source voltages V5 & V7 have a DC voltage of **Vsweep** , which will be used to sweep Vds in the simulation. Assign the device sizes and types as shown as well.

2. 2. Open & setup the Analog Environment for simulation:

   a. a. Setup the Model Library file to point to the same **…/amic5/mos.scs …** typ file used in the previous example.

   b. b. Select **Variables → Edit …** from the menu (or right side icons)

   c. c. Type in Name: **Vsweep** , Value: **2** , then click **Add** (this specifies variable Vsweep, and gives an initial value of 2V):

   d. d. Click **OK** then from menu **Analysis → Choose … → dc → Design Variable**

e. e.   Then either type **Vsweep** or click **Select Design Variable** ➔ **Vsweep** ➔ **OK**

f. f.   Enter sweep range, type, and step size as shown, then click **OK**:

| OK | Cancel | Defaults | Apply | | Help |

Analysis ◇ tran ◆ dc ◇ ac ◇ noise
◇ xf ◇ sens ◇ sp ◇ envlp
◇ pss ◇ pac ◇ pnoise ◇ pxf
◇ psp ◇ pdisto ◇ qpnoise

DC Analysis

Save DC Operating Point ☐

Sweep Variable

☐ Temperature
■ Design Variable **Variable Name** Vsweep
☐ Component Parameter **Select Design Variable**
☐ Model Parameter

Sweep Range

◆ Start–Stop **Start** 0 **Stop** 5
◇ Center–Span

Sweep Type

Linear ▭ ◆ **Step Size** 0.01
◇ **Number of Steps**

**Add Specific Points** ☐

Enabled ■ **Options…**

g. g.   To setup signals to plot after simulation, you can either use Outputs ➔ To Be Plotted as in the previous example, or you can select the signals after the simulation as done here. By default, node voltages and supply currents are saved, but device currents are not. In our case, the supply currents would be sufficient, but for instructional purposes, follow the step below to force saving of all device currents as well:

i. i. **Outputs** ➔ **Save All …** then for device currents, select **all** then **OK** :

3. 3.   Run the simulation: click the "green light" icon on the right for Netlist and Run (you will get messages & the test file, but no waveforms)
4. 4.   Plot the device currents for M0 & M1
   a. **a.   Select Results → Direct Plot → DC**
   b. b.   To plot device current, go to the schematic window and click on the device terminals you want to plot the current flowing into. Click on the DRAIN of M0, and SOURCE of M1 so both currents are positive. You will get circles on the nodes you have selected:
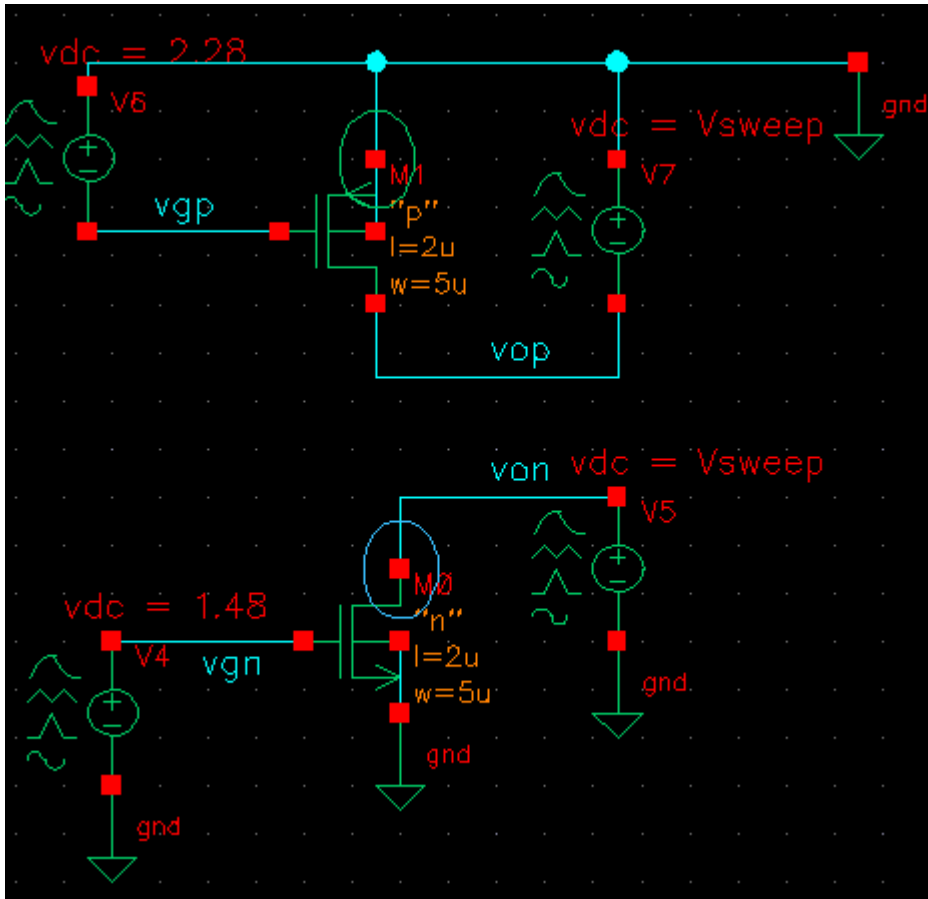
c. c. Hit ESC, then the following plots should pop up:
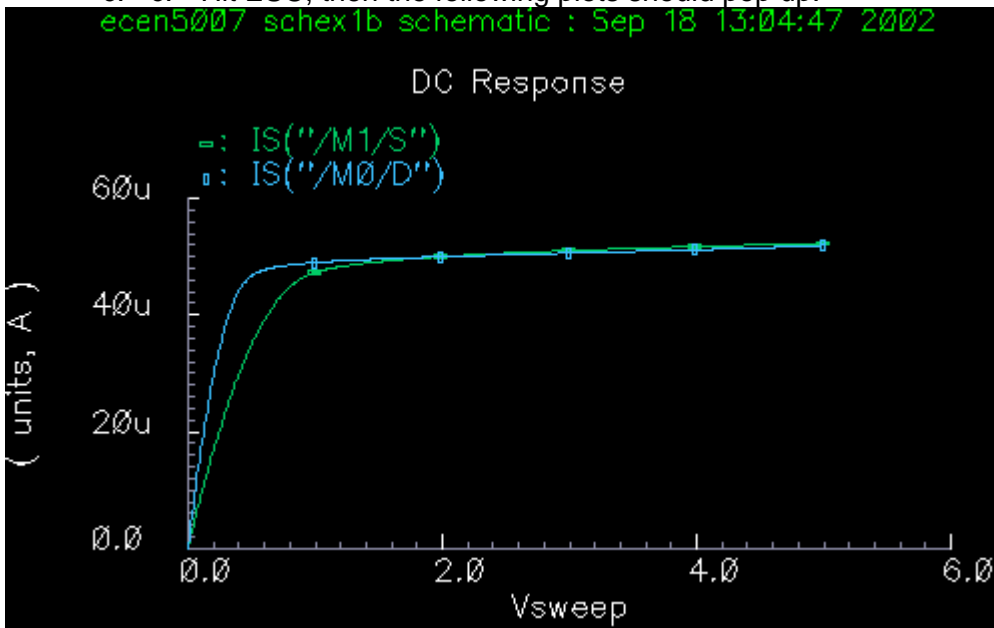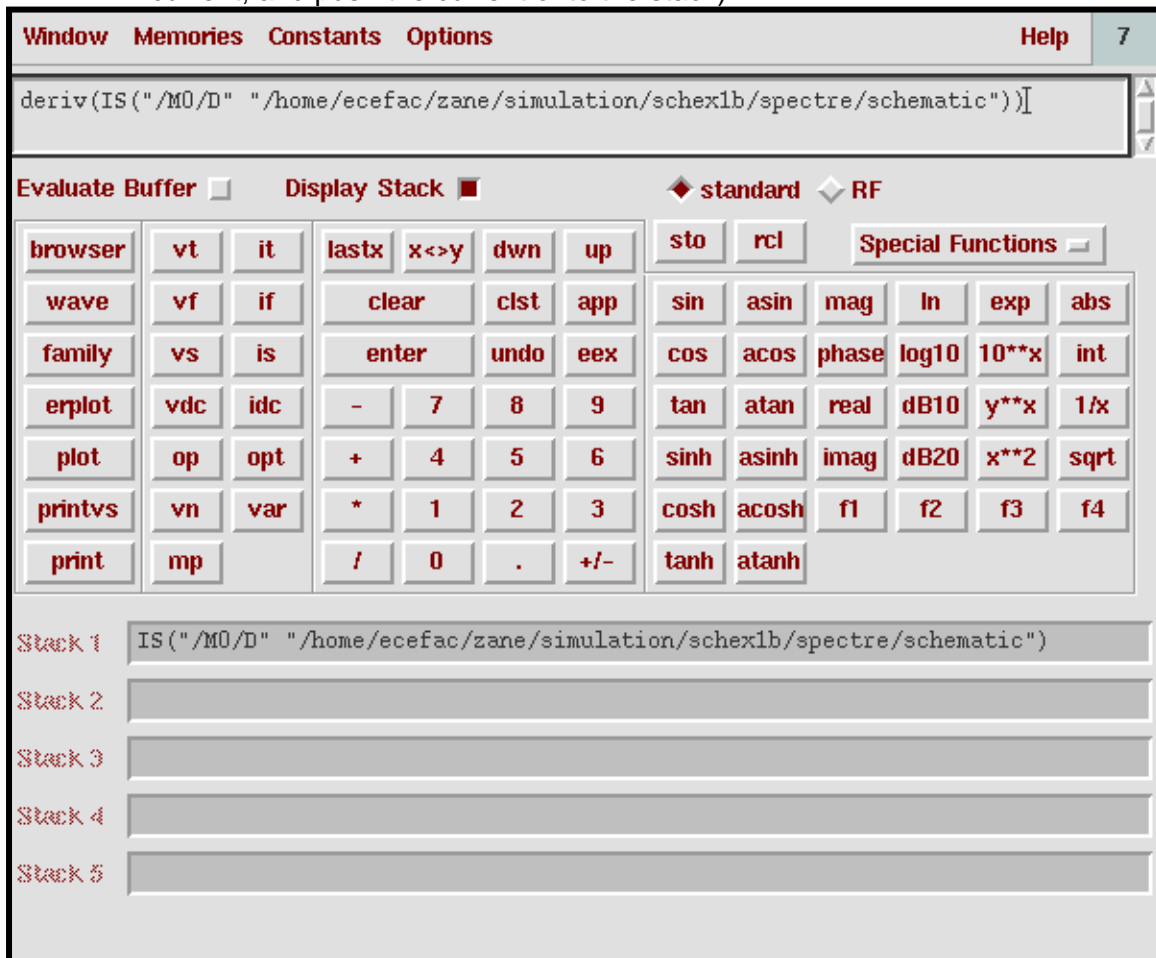

5. 5. Calculate the parameter LAMBDA:
   a. a. You could use the cursor bars to find delta I vs delta V, then estimate LAMBDA from Eq #3. In this case, we use the calculator function to perform the derivative and plot LAMBDA from Eq #2. You can use the calculator function to perform mathematical manipulation and/or export of the data from

your plots. The calculator is very useful and just takes a little getting used to.
Perform the following steps:

b. b. Click on the **Switch Axis Mode** icon on the left to separate the plots.
c. c. Click the **Calculator** icon on the left to bring up the calculator.
d. d. Depress the **Display Stack** button.
e. e. Click **wave** → then in the waveform window, click on the M0 (n-type) drain current → hit ESC key and return to the calculator:
f. f. Select**: Special Functions → deriv** (should place deriv(*) around your current, and push the current onto the stack):

| Window | Memories | Constants | Options | | | | | | | Help | 7 |

```
deriv(IS("/M0/D" "/home/ecefac/zane/simulation/schex1b/spectre/schematic"))
```

Evaluate Buffer ☐   Display Stack ■   ◆ standard ◇ RF

browser	vt	it	lastx	x<>y	dwn	up	sto	rcl		Special Functions ▭		
wave	vf	if	clear		clst	app	sin	asin	mag	ln	exp	abs
family	vs	is	enter		undo	eex	cos	acos	phase	log10	10**x	int
erplot	vdc	idc	–	7	8	9	tan	atan	real	dB10	y**x	1/x
plot	op	opt	+	4	5	6	sinh	asinh	imag	dB20	x**2	sqrt
printvs	vn	var	*	1	2	3	cosh	acosh	f1	f2	f3	f4
print	mp		/	0	.	+/–	tanh	atanh				

Stack 1   IS("/M0/D" "/home/ecefac/zane/simulation/schex1b/spectre/schematic")

Stack 2

Stack 3

Stack 4

Stack 5

g. g. Click: **var** → (in schematic window, select V5 or V7, go back to "VAR pop-up window") → select **Vsweep → OK**
h. h. Click: ***** (to multiply Vsweep with item in Stack 1, di/dv)
i. i. Click: **-** (to subtract above from Stack 1 item, Id)
j. j. Click: **1/x** (to take inverse)
k. k. Click: **wave** → (select M0 drain current from waveform again) → ESC
l. l. Select: **Special Functions → deriv** (di/dt for numerator)
m. m. Click: **dwn → clear → up** (to remove Id from stack 1)
n. n. Click: ***** (again to multiply)
o. o. Click: **plot** (to plot lambda as function of Vds)
p. p. To plot lambda for M1, cursor back in the calculator window and change all "M0/D" references to "M1/S", then click **plot**.
q. q. You should get the plots below:

ecen5007 schex1b schematic : Sep 18 13:04:47 2002

DC Response
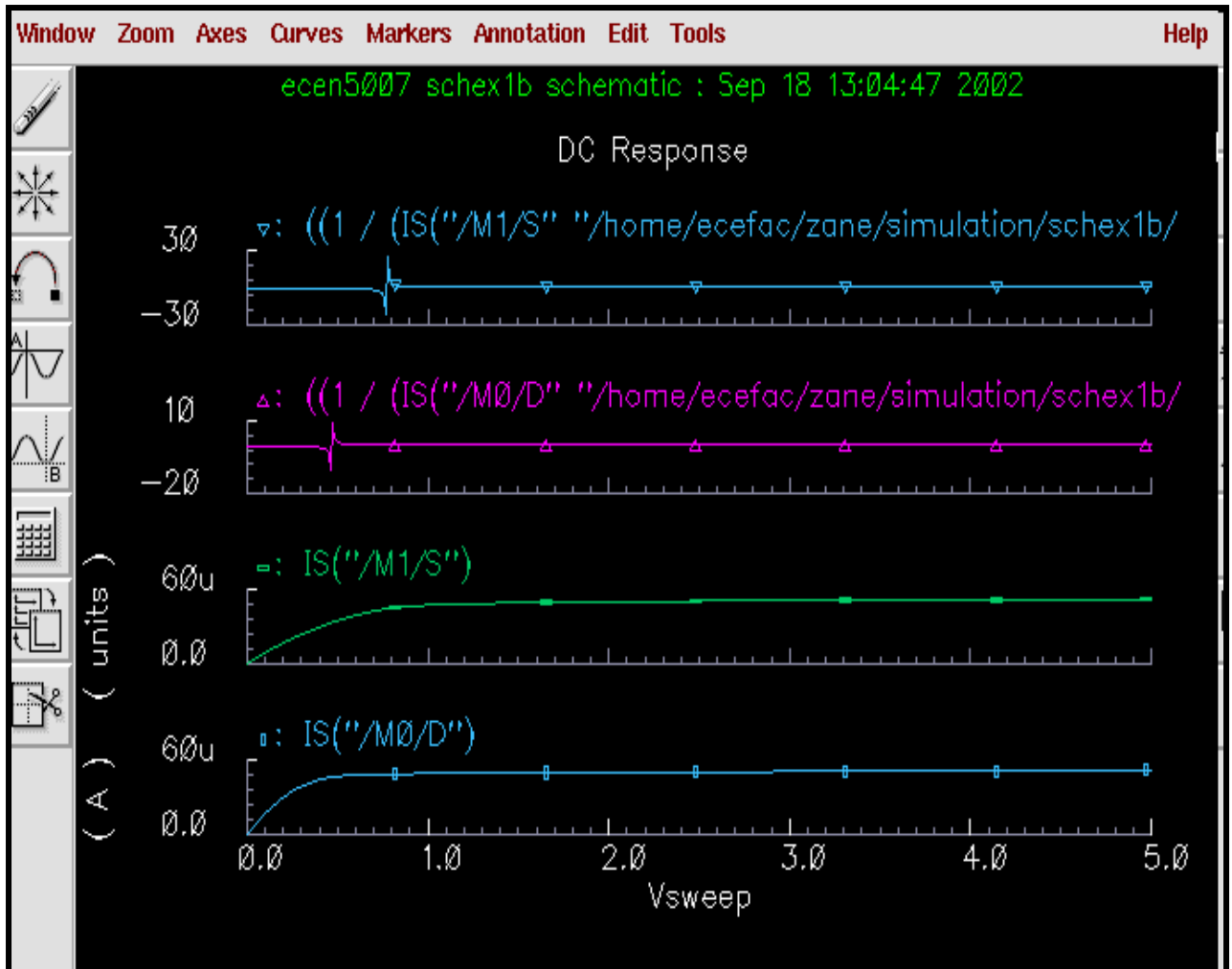
r. r.   Note that fitting the simplified Eq #1 to the BSIM model requires a Lambda that varies with Vds. From the plots, we get Lambda_n between 0.03 and 0.01 in the active region, with 0.01 a rough hand estimate over a wide range. Lambda_p varies from 0.1 to 0.01, with 0.03 to 0.02 a good hand estimate. Note also that these estimates are only valid for the device lengths used, L=2u. Additional simulations could be performed for the few device lengths typically used.