

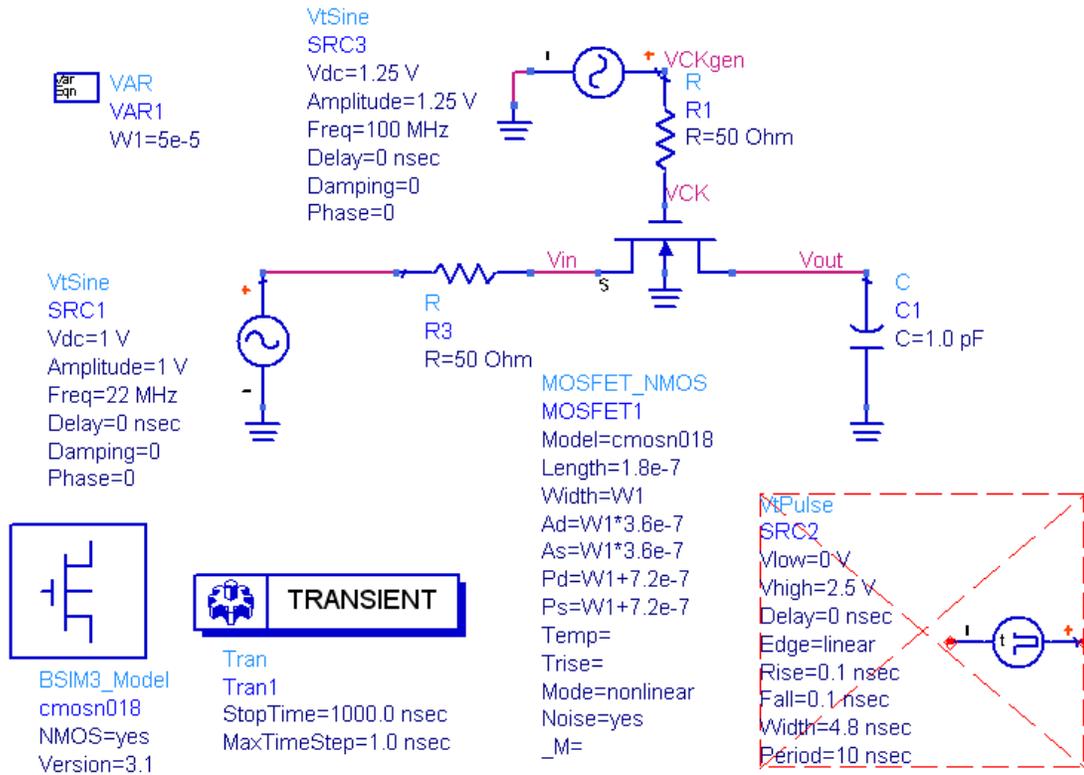
Due Monday April 18. Turn in at class or by 5pm.

Track and hold circuit. Review the Track and Hold part of the paper:

M. Choi and A. Abidi, "A 6-b, 1.3 GSample/s A/D Converter in 0.35 μm CMOS,"
 J. Solid State Cir., Vol 36, #12, pp.1847-1858, Dec. 2001.

Refer to the ADS tutorial documents on the course web site.

1. Start with a very simple MOS track and hold circuit. See the appendix to this HW set for a description of how to configure the MOS device.



a. Perform a transient simulation of the circuit and plot V_{in} , V_{out} , V_{CK} between 0 and 100 ns. Use the following conditions:

Input amplitude/offset	Input frequency	Clock amplitude/offset	Clock frequency	Hold capacitor	NMOS width
1V/1V	22 MHz	1.25V/1.25V	100 MHz	1 pF	50 μm

Carefully compare V_{in} and V_{out} . Identify and measure if possible any of the sampling errors. Then, modify the input amplitude and offset to reduce at least one of these errors. Compare results between a sine wave clock generator and the pulse generator clock source.

b. Go back to the sine wave clock source. Do a Fourier transform of the output voltage/input voltage and identify the primary spectral lines up to and including 100 MHz. Plot on a dB scale. The function “fs” in ADS performs a time domain – frequency domain transformation. It has the following syntax:

```
y = fs(x{, fstart, fstop, numfreqs, dim, windowType, windowConst,
tstart, tstop, interpOrder, transformMethod})
```

Use all default values except set $f_{start} = 0$ and $f_{stop} = 100$ MHz.
 $x = V_{out}/\text{amplitude of } V_{in}$.

The transient simulation must be run long enough to obtain adequate frequency resolution on the frequency spectrum.

The number of frequencies plotted = $(f_{stop} - f_{start}) * (t_{stop} - t_{start}) + 1$.

Since we would like 1 MHz resolution, and our span of interest is 100 MHz, we must run the simulation for 1 usec to get 101 points (0 – 100 MHz at 1 MHz increments).

c. Measure second and third order harmonic distortion, HD2 and HD3, (fundamental – to – harmonic ratio in dB) using markers and equations.

d. Add a charge compensating MOSFET clocked with CKbar. Adjust the width to obtain the least pedestal error. Compare pedestal error with the uncompensated version. Has the HD2 and HD3 been reduced?

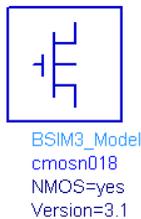
e. Now, simulate the differential sampler as shown in the paper by Choi and compare HD2 and HD3 on the differential output with a single-ended version.

Setting up MOSFET Parameters for ADS simulation.

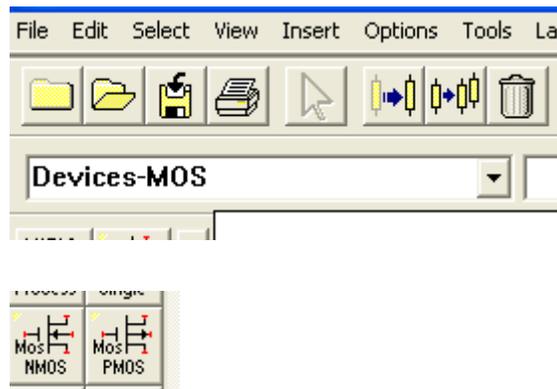
1. Using a MOS device in ADS requires that both the model and devices are included on the circuit schematic. Models for 0.18 um NMOS and PMOS devices were obtained from the Mosis website (www.mosis.org) and imported into ADS. This can easily be done from the ADS Main Menu if the HSPICE formatted parameters on the Mosis measurement files are saved as a text file. Open up a project then

- File > Import
- Browse to find the file.
- More Options: identify the type of format (HSPICE) and indicate whether the first line is a comment.

Anyway, this has already been done, and you can obtain models for the NMOS and PMOS devices (cmosn018 and cmosp018) from an archived ADS file CT.zap found on the course web page. Copy these model icons and place them on your schematic.



2. Next, scroll down on the palette selector and use the Devices – MOS menu. Select the MOSFET_NMOS or MOSFET_PMOS device to place on your schematic.



- MOSFET1 This is the instance name. Can be whatever you want to call it. Each device requires a unique name.
- Model=cmosn018 (must agree with device model)
- Length= 1.8e-7 (gate length - nonadjustable; model does not scale)
- Width (channel width. W1 is defined in a VAR statement. Dimensions are in meters)
- Ad, As: area of source and drain. I assumed the contact length was 2 lambda.
- Pd, Ps: perimeter of source and drain. Same assumption.
- Temp: this defaults to 27C
- Trise: increase in temperature above Temp
- Mode=nonlinear. Default. Why else would you use this model?
- Noise: includes model for channel noise – useful in noise figure simulations.
- _M= Multiplication factor.



- The area and perimeter must be specified to produce the correct area and sidewall capacitances.
- The multiplication factor should be used whenever a large device is required. The actual performance at high frequencies will be quite different for one mosfet with a very long gate and one with the same total length but consisting of multiple instances of the device connected in parallel. The reason is that a long, very narrow gate behaves as a distributed RC line. Thus, there will be a phase shift or delay from one end of the gate to the other. This will degrade the performance of the device significantly.

April 2005