IN 5230 Electronic Noise – calculation and countermeasures

Mandatory task number 1.

Deadline for delivery: Monday 16th of September at 08:00.
Assessment: Approved/Not approved.
Email to: jonheri@ifi.uio.no cc: joar@ifi.uio.no
Reports are submitted on an individual basis. The reports shall consist of all the schematics that are used to achieve the reported results, simulation results and text explaining what has been done, summary tables as well as an analysis of the results. The report should be submitted electronically as a pdf file. Schematics/symbols at LTspice format may be attached as well. Use white/gray background on schematics and simulation results. Avoid yellow color on curves. Use legends with multiple lines! The report should be in ENGLISH.

Tool

Download the free simulator LTspice/SwitchCad from Linear Technologies and install on the machine you want. The software is available at:
http://www.linear.com/designtools/softwareRegistration.jsp
(You may install program and libraries under C:\<Program Files>\LTC\LTspiceXVII\ .
The exact name of <Program Files> depends on your operation system. You may install it another place if you want but remember where you put it.)

1. Get familiar with the simulator

LM741

Recall circuit LM741 from the library which comes with the software.
(LTC\LTspiceIV\examples\Educational)

a) Transient Analysis.
Identify the components that belong to the feedback and which belongs to the LM741 amplifier. What gain do you expect to get theoretically? Let the DC offset on the positive input remain zero while you find the DC offset interval for the negative input where the output does not go into saturation. What is the relation between he input signals DC-offset, amplitude, gain, and output range? Explain!

b) Set up for frequency analysis (AC analysis). What is the DC gain and Gain Bandwidth (GBW)? What is the phase margin? What is the relation between DC gain and GBW in feedback systems?

c) Perform AC analysis with varying common DC-offset (common mode) and find the approximate area where the gain is greater than -6dB of maximum level.

d) Add a load capacitor and use the STEP function to create a plot in which the capacitive load is increased to 30pF in increments of 10pF. What is the new DC gain and GBW? What is the main effect of changing the capacitor’s value?

2. Frequency characteristics of some curves.

Draw a new schematic consisting of a voltage source and a first order low pass RC filter. The voltage source and the capacitor are grounded. Name the voltage source node “A” and the output of the filter “B”. Start by selecting RC filter values so that it has no attenuation (i.e. very high cutoff frequency).

a) Let the voltage source generate a sine wave with period 1 millisecond and amplitude 1V. Simulate over a period of 1 second and set maximum timestep to 1/10 period. Perform an FFT on node A. What did you expect? What is the strength of the strongest unwanted frequency component? Perform the same again with maximum timestep 1/100 of the period. What's going on? What is the strength of the strongest unwanted component? What does this say about the simulator setup?

b) Let the voltage source generate a symmetrical square pulse with the same period as the sine. Simulate with rise and fall time equal to i) half the period (that is a triangle pulse), ii) 1/10 of the period and iii) 1/100 of the period (the signals should look as clock signals, so set Ton = 0 for i), Ton=0.5m-Trise for ii) and iii)). Perform an FFT on node A and measure the strength and frequency of the strongest signal for the fundamental frequency. What's going on? What should the clock edges look like to reduce the high frequency components in the signal? (What's the downside of having gently sloping edges in eg. standard CMOS logic?)

c) Calculate and give RC filter values so that we get a cutoff frequency around 5 times over the fundamental frequency. Use the input described in point b-iii). Perform a transient simulation and measure at node A and B. How do the curves look like now? How is the FFT output?

3. Decoupling capacitors

In this subtask you shall find the size and number of capacitors required to reduce the supply voltage noise to an acceptable level. You will find all the equations in the first lecture note. (Literature background can be found in Ott 11.4). Each capacitor has an inductance of 10nH. The current spikes can be modelled as a triangular shape from 0A to 1A with a rise and fall time of 5ns. We want the voltage to be stable within 5% of 1.5V. The lower frequency corner is 1MHz.

a) What is the low frequency target impedance Zt based on the values above?
b) What is the number of capacitors required?
c) What is the total capacitance? And what is the capacitance per capacitor?
d) What is the high frequency corner decided by the current rise/fall time?
e) Draw a schematic and do transient and frequency simulation. The schematic consists of a current source, an inductor and a capacitor. The inductor and capacitor represent all the parallel coupled capacitors. (In addition, you may add a resistor in parallel with the source to empty the current between the spikes and remove the step behaviour. If it is not too small it will not influence on our noise analysis). Make sure to get the correct value on the inductor.
f) What will be the difference if the raise/fall time is reduced to 1ns?

4. Parasitic capacitive coupling

Two lines are routed in parallel over a length of 10cm. The capacitance between the lines is 0.1pF/cm. In one line (S=source) there is a signal with 5V swing that is received as noise in the second line (O = object). The receiving line has a capacitance and a resistor in parallel to ground of 10pF and 10MΩ.

a) How much noise is captured in the receiving line? Show this both by calculation and simulation.

b) We will put a shield around the receiver. The shield is grounded and it has a capacitance with the inner conductor of 1pF/cm. For each cm with shield, we can disregard the capacitance between the object O and the source S for the same distance. Show by simulation and calculation what noise is received with a 2cm, 5cm, 9cm, and 9.9cm shield.

5. Artificial sources of transient analysis

Sometimes we need to create artificial noise sources to see how the circuit responds to noise. LTspice has a general voltage source "BV" (Arbitrary Behavioral Voltage Source) where the voltage may be described by functions. Some of the functions are RAND(i), RANDOM(i) and WHITE(i), which generate a random number within a range. For RAND(i) and RANDOM(i) the interval is [0: 1], while for WHITE(i) it is (-0.5, 0.5). These functions draw quasi random numbers within their intervals depending on the <u>integer</u> of the seed *i*. According to typical use in simulators and programming languages RAND(i) and RANDOM(i) is uniform within the range [0:1), i.e. all values within the ranges are equally likely and no values outside will ever happen (opposed to the normal/Gauss distribution). The WHITE(i) function is not so often available and should be, as expected from the name, a semi gaussian distribution but limited within (-0.5,+0.5). Real gaussian distributions with no upper and lower limits has typically to be developed in other software and to be included as data files. (As informed in the lecture there has been some inconsistency in LTspice regarding this. In last years version all of RAND, RANDOM and WHITE had a uniform distribution while now it seems as all three are semi gaussian. We need to take some creative solutions to handle this.)

To generate a sequence of random numbers we may use "time" as the index. "Time" has the units in seconds and thus will generate a new number each second. If we want new numbers, for example, every millisecond, we have to multiply with the inverse of a millisecond (=NoiseRate) and *time*1E3* will be the index. Often, we need more sources and it is important to make them independent.

For the next two tasks it does no matter whether you use a uniform, semi gaussian or a real gaussian source. WHITE is proposed but the others are equally good.
a) Make two BV sources *WHI1* and *WHI2* both having WHITE(time*1E3) as function. Subtract the resulting curves in the simulation window and comment on the result.

b) Change the index in one of the sources to say WHITE(time*1E3 + 1), subtract the resulting curves and comment on the new result.

This noise model has a limited validity due to the limited value range. One possibility is to generate a pseudo normal distribution based on the uniform sources. This can be done by adding a number of uniform sources. The format of a BV function may look like: function(time*NoiseRate*Nsource+Isource). If we generate the pseudo normal noise source from 12 of these sources, Nsource (number of sources) is 12 while Isource (index of the individual 12 sources) are from 0 to 11. The standard deviation is 1.

c) In this year's version of LTspice it does not seem as we have a uniform source and we have to use uniform random distributions available on file on our website instead of WHITE. On our web-site you will find twelve files rnd1.txt to rnd12.txt. Read each of them into twelve voltage sources and add them together with a BV source. Show the simulation result. What is the maximum and minimum values that these 12 sources can generate? Try to find the standard deviation. What is the probability that a real gaussian source will generate a value outside of the maximum and minimum values for this range?

A possibility is to generate a real normal distributed noise (or any other noise distribution) with another application (e.g. Excel) and store it as a text-file.

d) Create a voltage source FILE1 browsing file PWLsrc_10k_1ms.txt (from course webpage) (10k: 10k points, 1ms: 1ms between points). Compare the curve with the curves you got with the –other functions you have looked at so far.

e) Create a new BV source FILE2 (general voltage source) and give it the function DELAY(V(FILE1),1m). Subtract the two curves and comment on the result.

Enjoy! :)