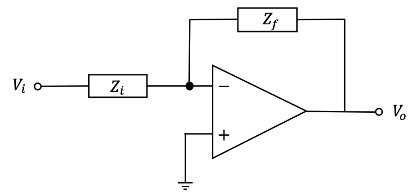
UC San Diego, ECE 100

Lab Project 4: Design of a Differentiator

This project aims to design a signal processing circuit that will output the time derivative of the input scaled by a desired factor. This *differentiator* should convert a triangle wave of 1 V_{pp} (peak-to-peak amplitude) at 1 kHz into a square wave of ± 1 V (at 1 kHz, of course). The polarity of the square wave is unimportant, but the 10% to 90% rise time must be less than 20 μs, and the output voltage must not overshoot more than 15%, i.e., 0.3 V.

The circuit can be designed using the operational amplifier inverter configuration. If the op-amp is ideal, this simply requires an input capacitor and a feedback resistor. Then the ideal transfer function is $H_{ideal}(s) = -sRC$, which (apart from a phase shift of 180°) is what we need. Unfortunately, the simple design will be either marginally stable or completely unstable, and some *compensation* will be required. In this case, "unstable" means that the circuit will oscillate with no input present and thus is useless as a differentiator. The nature of the compensation is very much like how the follower was modified to drive a large capacitive load. In this project, we will study how to do this compensation theoretically.

The closed-loop gain or the transfer function for the op-amp inverter, including the frequency-dependant open-loop op-amp gain, $a(s)$, is:



$H(s) = A(s) = \frac{V_o(s)}{V_{in}(s)} = -\frac{Z_f(s)}{Z_i(s)} \times \frac{a(s)B(s)}{1+a(s)B(s)}$,

where $T(s) = a(s)B(s) = a(s)\frac{Z_i(s)}{Z_i(s)+Z_f(s)}$ is the loop gain.

When the loop gain is large, $A(s) \to -\frac{Z_F(s)}{Z_I(s)}$, which can be called the "ideal" transfer function because it would apply if the op-amp were ideal. The factor $\frac{aB}{1+aB}$ gives the deviation from ideal behavior. All feedback circuits have a factor like this in their transfer function. The possibility of having a zero in the term $(1 + aB)$ causes instability.

1. **System Level Design**

Using $H_{ideal}(s) = -\frac{Z_f(s)}{Z_i(s)} = -sRC$ in an op-amp differentiator, we know that $v_o(t) = -RC\frac{dv_i(t)}{dt}$. We need $v_o(t) = \pm 1\ V$ or $2\ V_{pp}$ for a $1\ V_{pp}$ triangle wave input signal. We must choose the RC product necessary to meet the specs. A reasonable choice for the resistor is $R = 100$ kΩ. You can then find the required capacitor value.

(a) Check that R and C have correctly been calculated by simulating the differentiator circuit with an ideal op-amp. Read the **Notes** included in this document to learn more about finding this component. Use VPulse as the input and set the parameters to provide a few cycles for a $1\ V_{pp}$ triangle wave. Make sure that the output voltage is the desired square wave. Include these plots in your report and show your work for the calculations of the capacitor value.

(b) We can think of the differentiator as the product of H_{ideal}, which converts the triangle wave into the desired square wave, and a second filter $\frac{aB}{1+aB}$ which turns out to be a second-order low-pass filter. This low pass filter will have finite bandwidth and, thus, finite rise time. We will have to ensure the bandwidth is broad enough that the rise time will meet the spec. Write the expression for $\frac{aB}{1+aB}$ in normalized form assuming $a(s) = \frac{G'}{s}$. You will see that it is a quadratic low-pass filter. Find expressions for ω_0 and ζ in terms of G' and $\tau = RC$. For a second order circuit, t_r (rise time) depends on both ω_0 and ζ, however for a first order circuit with transfer function $H(s) = \frac{1}{1+\frac{s}{\omega_0}}$, the rise time can be calculated as $t_r = \frac{2.2}{\omega_0}$. In a second-order system, with small ζ values, the rise time will be shorter than this limit. Using this limit, what is the minimum value of G (the unity gain bandwidth in Hz) needed for the circuit? Will the minimum G of the LF411 be satisfactory? Provide calculations. (Look up the datasheet for the min G value for the LF411 op-amp). The spec provided in the introduction part of the lab states the requirement on the rise time.

(c) Estimate ζ using the typical value of G for the LF411. What overshoot would you expect? Use $\%os = 100 * exp(\frac{-\pi\zeta}{\sqrt{1-\zeta^2}})$. Clearly, this will not meet the spec. The problem is that the phase margin of the loop gain is rather small. We must increase the phase margin to at least 45°. Make a Bode plot of $a(s)B(s)$ using MATLAB, find the unity gain frequency, and then read off the phase margin. Use the provided MATLAB script to find the phase margin.

```
s = tf('s');    %define s as a transfer function variable
G1 = 2*pi*3e6; r = 10e3; c = 0.5e-7; %NOTE your constants may ...
    be different
tau = r*c;
T = G1/(s*(1+s*tau));
bode(T,{100,1e6});  %does the entire Bode plot for 100<w<1e6
margin(T);  %calculates phase margin for you
```

Using MATLAB, you can also easily confirm that the system has too much overshoot.

```
H = T/(1+T);  %calculate the closed loop transfer function
step(H, 1e-3);  %plot the step response
stepinfo(H)     %Calculates overshoot for you
```

(d) We can improve the phase margin by modifying the loop gain. Adding a zero near the unity gain frequency can be very helpful in such cases. You can do this by putting a compensation resistor R_C in series with the input C. Write the modified expression for $a(s)B(s)$. You will see that adding R_C places a zero at $\frac{1}{R_C C}$. Find the value of R_C that puts this zero at the unity gain frequency. Create the Bode plots of the compensated loop gain, $a(s)B_C(s)$, and find the new phase margin. Show that you can obtain exactly the same effect by putting C_C in parallel with R instead of R_C in series with C. Create the Bode plots of the revised loop gain.

(e) We need to check the overshoot with the compensation resistor. The overshoot should be much smaller but may still be over the spec. Increase R_C until the overshoot just meets the spec. Check the phase margin with this R_C.

Include the MATLAB plots for parts c, d, and e in your report.

2. **Circuit Level Spice Simulation**

The system-level design was done using a rather simple model for the op-amp, and it may be too simple to get the behavior exactly correct. Thus, we must simulate the circuit using a macro model for the desired op-amp. (A macro model is a simplified circuit model that captures most (though not all) the circuit's behavior using a simplified schematic.)

(a) Simulate the original differentiator (with $R_C = 0\ \Omega$) using the specified input triangle wave and the LF411 op-amp. Plot the input and output waveforms and measure the rise time and the overshoot. Include the simulation result in your report.

(b) Repeat the simulation with the calculated value of R_C. Measure the rise time and overshoot. Does it meet the spec? It should be close. If it does not meet the spec, adjust R_C until it does meet the spec. Include the simulation result in your report.

(c) Find the C_C that should put a zero in exactly the same place as R_C and simulate to confirm that the rise time and overshoot are the same. C_C is the capacitor in parallel with R, instead of having R_C in series with C in the differentiator circuit. Include the simulation result in your report.

3. **Measurement**

It is essential to test real systems where stability is an issue because small variations in the circuit may be quite important. In this case, the real op-amp significantly differs from its spice model, which can impact stability. Furthermore, the actual circuit, as constructed on a breadboard, has significant "stray" capacitance at the circuit nodes, which can be important too. Before you start, check the calibration on your scope probes and adjust if necessary. Would you expect slew rate limiting to be a factor with this circuit?

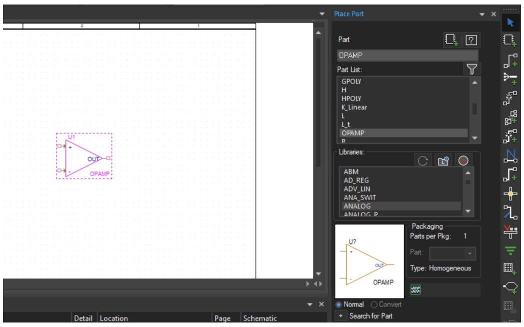
(a) Set up and measure the original differentiator (with R_C =0 Ω) using the specified input triangle wave. Measure the rise time and the overshoot. Make a hard copy of the waveform and your measurements. Include these results and a picture of your circuit setup in your report.

(b) Repeat the test with the calculated value of R_C. Measure the rise time and overshoot. Does it meet the spec? It should be close. If it does not meet the spec, adjust R_C until it does meet the spec. Make a hard copy of the waveform and your measurements.

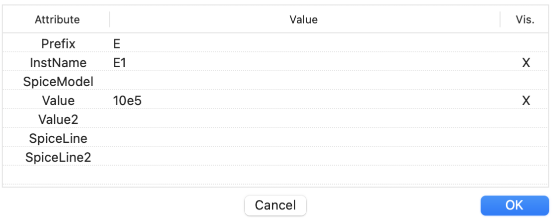
(c) Try the test with the equivalent value of C_C instead of R_C and see if the overshoot and rise time are the same in real life.

Notes:

Note1: In PSpice, you can use a part named ”opamp”, located in the PSpice/Analog library, as an ideal op-amp.



Note2: In LTSpice, you can use a part named ”e”, as an ideal op-amp. In the attributes of the part, add ”10e5” to the Value attribute. This component has an infinite input resistance and a zero output resistance similar to an ideal op-amp. The voltage gain of this component will be 10^5 V/V. The following picture shows how to modify the attributes.



Attribute	Value	Vis.
Prefix	E	
InstName	E1	X
SpiceModel		
Value	10e5	X
Value2		
SpiceLine		
SpiceLine2		

Lab Report:

Make sure you have the following in your lab report:

- Calculations for finding the capacitor value and a copy of the input triangle wave and output square wave (part 1a).
- Expressions for $\frac{aB}{1+aB}$, ζ, ω_0, and minimum G value for the circuit (part 1b)
- MATLAB simulations (part 1c)
- MATLAB simulations with R_C and C_C compensators (part 1d)
- MATLAB simulations with adjusted R_C (part 1e)
- Spice schematic of the original circuit and simulation with measured rise time/overshoot (part 2a)
- Spice schematic of the circuit with adjusted R_C and simulation with measured rise time and overshoot (part 2b)
- Spice schematic of the circuit with added C_C and simulation with measured rise time and overshoot (part 2c)
- MATLAB script

This list is not all-encompassing; check with your TAs, but it serves as a helpful checklist.