www.simiode.org
SIMIODE
A SYSTEMIC INITIATIVE FOR MODELING INVESTIGATIONS
& OPPORTUNITIES WITH DIFFERENTIAL EQUATIONS

# STUDENT VERSION
# Op Amp Differentiator

Virgil C. Ganescu
York College of Pennsylvania
York PA 17403 USA
vganescu@ycp.edu

**Abstract:** In this validation-oriented setup, the output waveform (function) of a operational amplifier type of differentiator circuit is determined analytically from the first order governing ordinary differential equation and the results are compared with the data acquired from analyzing the numerical model (using Multisim [5]) of the circuit.

## SCENARIO DESCRIPTION

An operational amplifier (op-amp) differentiator (or a differentiating amplifier) is a circuit configuration which produces an output voltage amplitude that is proportional to the rate of change of the applied input voltage (signal).

An op-amp differentiator is an inverting amplifier, which uses a capacitor in series with the input voltage. Differentiating circuits are typically used to find the high-frequency component of the input signal and in applications of edge detection. In essence, these operational amplifier circuits are meant to produce output voltages which are directly proportional to the input voltage's rate-of-change with respect to time.

For a sine wave input, the output of a differentiator is also a sine wave, which is out of phase by 180° with respect to the input (cosine wave); this stems from the fact that the input signal to our subject circuit is applied to the inverting input of the op-amp.

Differentiators have frequency limitations while operating on sine wave inputs; the circuit attenuates all low frequency signal components and allows only high frequency components at the output [1].

### Part I: Analytically Solving the Governing Expression of the Circuit

Consider the Op-Amp Differentiator circuit in Figure 1.

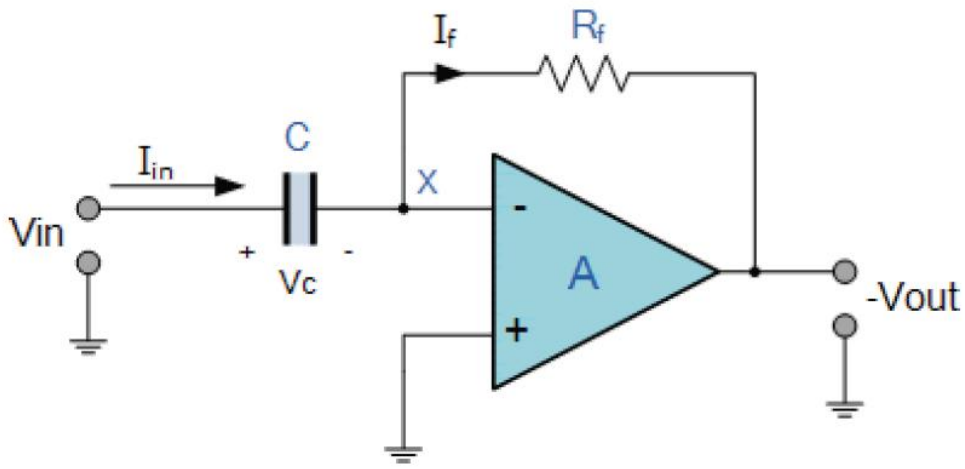


**Figure 1.** Schematic of Op-Amp Differentiator Circuit.

Since the input signal is applied to the capacitor, only alternating current (ac) type of input voltage changes will be "allowed" to pass through, and whose frequencies are dependent on the rate of change of the input. At low frequencies, the reactance of the capacitor is high, resulting in low gain, and hence low output voltage from the op-amp; at high frequencies however, the reactance of the capacitor is much lower resulting in a higher overall gain, and higher output voltage, respectively [2].

Due to the fact that the node voltage of the operational amplifier at its inverting input terminal is zero, the current ($I_{\text{in}}$), flowing through the capacitor will be given by:

$$I_{\text{in}} = I_{\text{f}} \tag{1}$$

and

$$I_{\text{f}} = -\frac{v_{\text{out}}}{R}\,. \tag{2}$$

The charge on the capacitor is given by:

$$Q = C \cdot v_{\text{in}} \tag{3}$$

and therefore, the rate of change of $Q$ (which is the capacitor current) can be calculated as:

$$I_{\text{in}}(t) = \frac{dQ(t)}{dt} = C\frac{dv_{\text{in}}(t)}{dt}\,. \tag{4}$$

Ultimately, from relationships (1), (2), and (4) we conclude that the output voltage (function) governing the design above can be stated as:

$$v_{\text{out}}(t) = -CR\frac{dv_{\text{in}}(t)}{dt}\,. \tag{5}$$

Equation (5) is a first order (ordinary) differential expression, typically having the output voltage $v_{\text{out}}$ as the unknown (function of time). $v_{\text{out}}(t)$, determined in Part I, will be compared with $v_{\text{out}}(t)$ computed using a numerical simulator, Multisim (to be demonstrated in Part II). Note that these results may be eventually compared with the measured $v_{\text{out}}(t)$ from an actual (physical) circuit/prototype.

For this modeling scenario, an $v_{\text{in}}(t) = \sin(100t)V$, $R = 1k\Omega$ and $C = 1\mu F$ have been selected.

$v_{\text{out}}(t)$ obtained using $v_{\text{in}}(t) = \sin(100t)V$, $R = 1k\Omega$ and $C = 1\mu F$ in (5) represent instantaneous values of the output voltage generated by the differentiator circuit. PEAK values of $v_{\text{out}}(t)$ will be observed for periodic instances in time when $\cos(100t)$ will be equal to $\pm 1$. Additionally, note the "minus" (to be interpreted as a phase shift) in front of the output signal coming out of the differentiator circuit.

## Part II: Determining $v_{\text{out}}(t)$ Using Modeling Software (Multisim)

Multisim software [5] integrates the industry-standard SPICE software [6] simulation with an interactive schematic environment to instantly visualize and analyze electronic circuit behavior. By adding powerful circuit simulation and analyses to the design flow, Multisim helps researchers and designers reduce printed circuit board prototype iterations and save development costs. One of the attractive features of using Multisim centers on the almost non-existent learning curve. Free, limited time trials (for educational purposes) can be downloaded from [4]. Workstations - with Multisim already installed - from a dedicated electrical engineering lab, say, can also be used. Using Multisim, as seen in [3], the circuit shown in Figure 2 is built, with the source voltage $v_{\text{in}}(t) = \sin(100t)$, that is, input capacitor $C(1\mu F)$ and feedback resistor $R(1k\Omega)$ just used in Part I; additionally, a generic 741 operational amplifier is recommended (with $\pm$15V DC biasing levels).

In order to visualize the signal coming out from the circuit, an oscilloscope (XSC1 - top right corner in Figure 2 is utilized. As connected, XSC1 is used to monitor the values of the output voltage of the differentiator circuit, as well as indicate how this output voltage behaves. For ease in contrasting the data gathered via the procedures suggested in Part I and Part II, PEAK (highest or lowest) values are used for $v_{\text{out}}(t)$, while identifying points in time where PEAKs can be viewed on XSC1 and use said time values to analytically estimate $v_{\text{out}}(t)$ - as derived from (5).

Even more specifically, while in Multisim, the simulation is turned on, and then paused, once stabilized. On the display of the oscilloscope XSC1, any of the two cursors is dragged to any of the PEAKs (positive or negative PEAK) of the voltage visualized. These identified PEAKs are observed at the very points in time ($t$, in seconds) we use to determine the PEAK values of $v_{\text{out}}(t)$, as calculated from (5).

Cursor 2 (blue) from Figure 3 indicates a PEAK output voltage of approximately -0.099 V (instance occurring at time $t = 293.171$ s). Note that this is just one of the many PEAKs! To

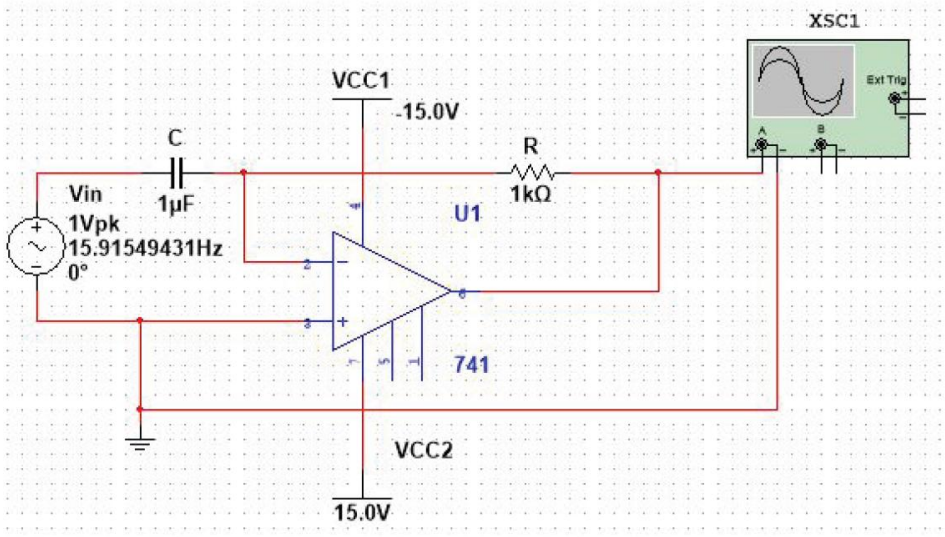


**Figure 2.** Multisim Schematic of the Op-Amp Differentiator Circuit.

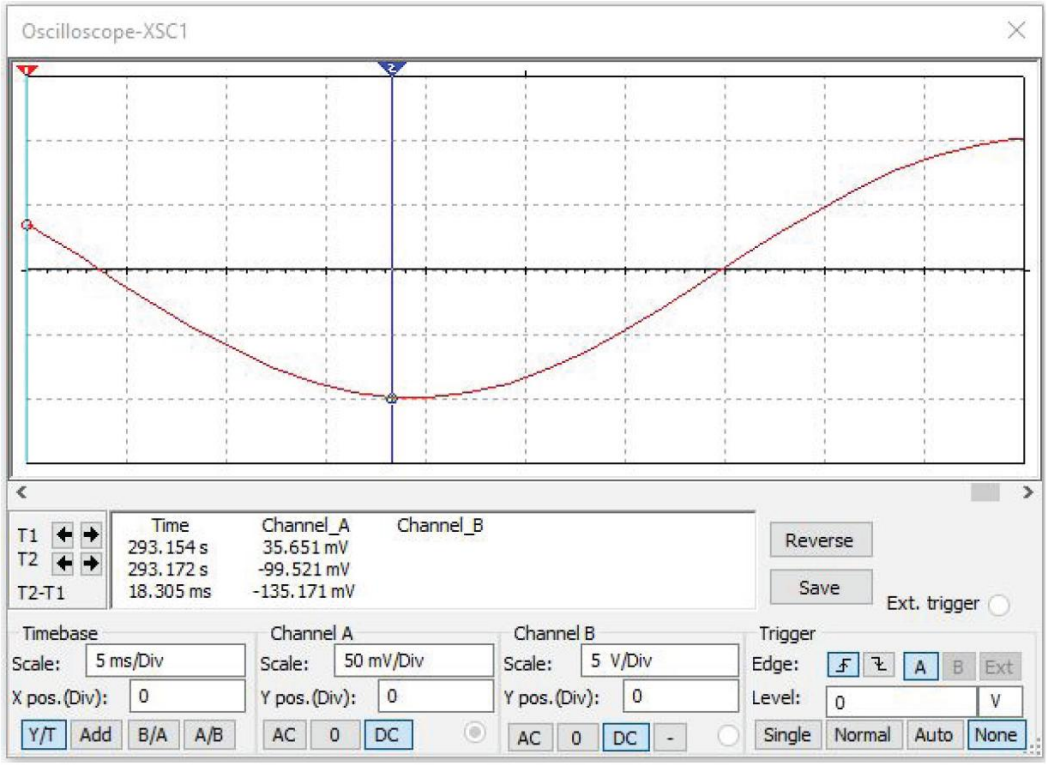


**Figure 3.** Screen of the oscilloscope (XCS1) from Figure 2.

concisely summarize the modeling efforts, a table similar to Table 1 can be used.

| Time (in s) | $V_{out}$ : Analytical (in V) | $V_{out}$: Multisim (in V) |
|---|---|---|
| 293.171 | -0.098 | -0.099 |
| another instance where another peak was observed ... | | |
| another instance where another peak was observed ... | | |
| another instance where another peak was observed ... | | |

**Table 1.** Analytical vs Numerical Output Voltage Levels.

## REFERENCES

[1] Electronicshub. 2020. *Operational Amplifier a Differentiator*. https://www.electronicshub.org/operational-amplifier-as-differentiator. Accessed 2 August 2020.

[2] Electronics-tutorials. 2020. *Differentiator Amplifier*. https://www.electronics-tutorials.ws/opamp/opamp_7.html. Accessed 2 August 2020.

[3] Ganescu, V. C. 2019. *Multisim Brief Tutorial*. https://use.vg/On4CH7. Accessed 2 August 2020.

[4] Multisim. 2020. *Free Academic Evaluation of Multisim*. https://lumen.ni.com/nicif/us/academicevalmultisim/content.xhtml. Accessed 2 August 2020.

[5] National Instruments. 2020. *What is Multisim?* https://www.ni.com/en-us/shop/electronic-test-instrumentation/application-software-for-electronic-test-and-instrumentation-category/what-is-multisim.html. Accessed 2 August 2020.

[6] Rhome Semiconductors. 2020. *Types of SPICE Models*. https://techweb.rohm.com/knowledge/simulation/s-simulation/01-s-simulation/8294. Accessed 2 August 2020.

[7] Testguy. 2020. *Peak vs. Average vs. RMS Voltage*. https://testguy.net/content/%20270-Peak-vs-Average-vs-RMS-Voltage. Accessed 2 August 2020.