Transient and Frequency Analysis of various Analog, Op-Amp and Mixed Signal Circuits using LTSpice

Introduction: -

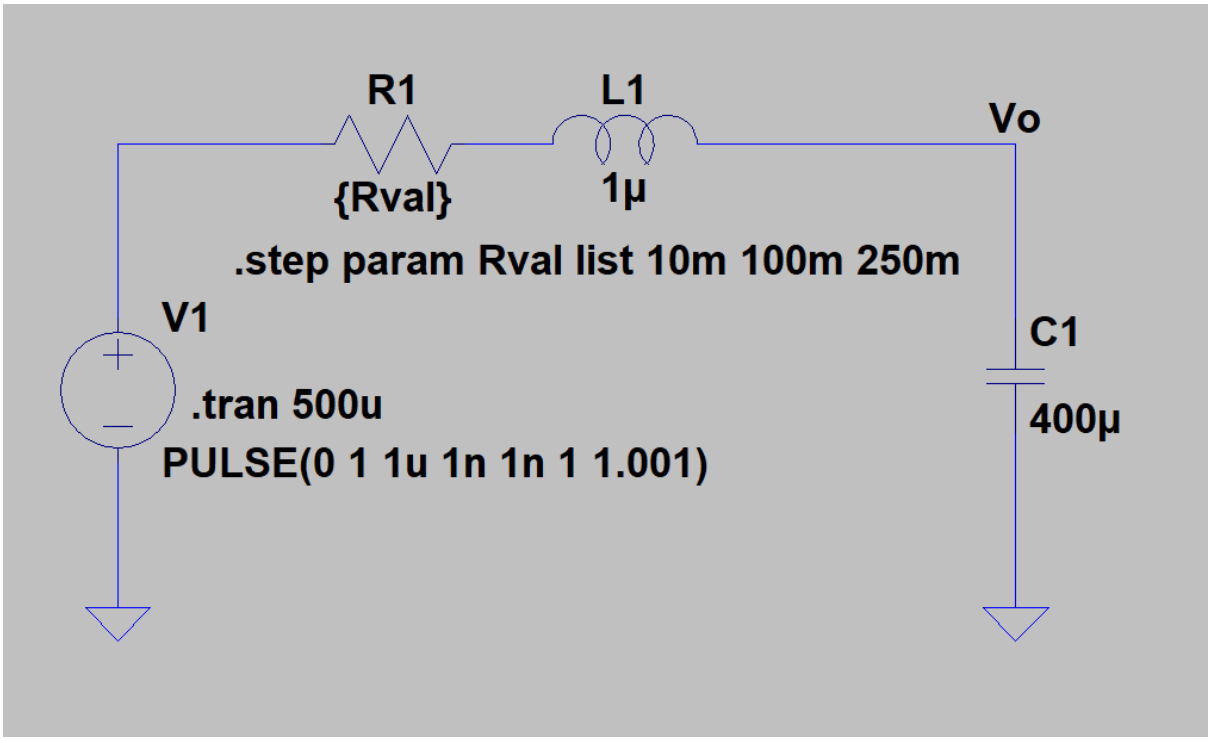
The project aims to simulate, analyse and understand the behaviour of various analog circuits by utilising transient and frequency analysis on LTSpice.

Following are a few circuit simulations.

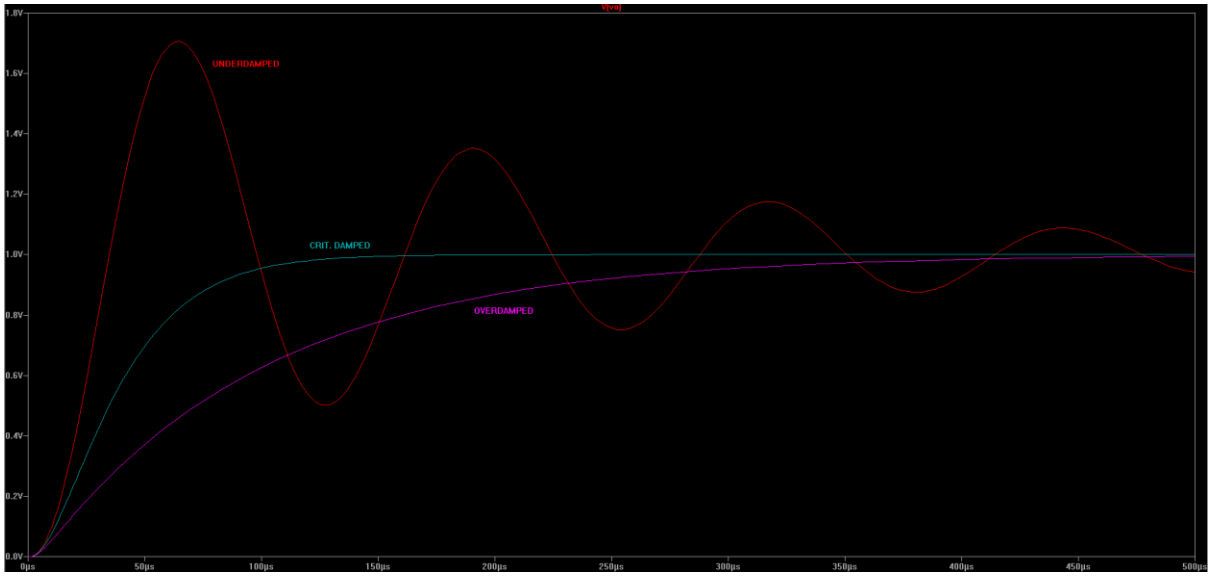
Transient analysis of series RLC circuit: -

For **C = 400uF** and **L = 1uH**, we get the value of **R = 100 m-Ohm** to be the condition for critically damped (from $\mathbf{R^2 = 4(L/C)}$). Any value of R less than this would render the circuit to operate in an under-damped condition while the value of R greater than this would make it operate in an over-damped condition.

Attached are the circuit along with simulation parameters and the waveform.

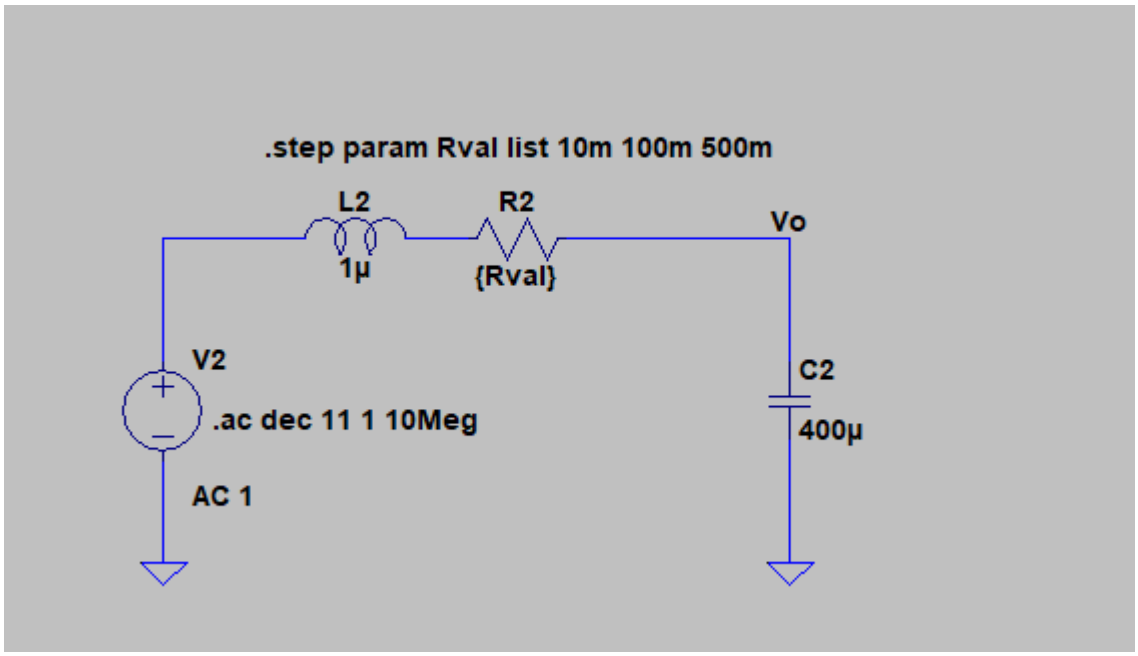


Schematic for transient analysis



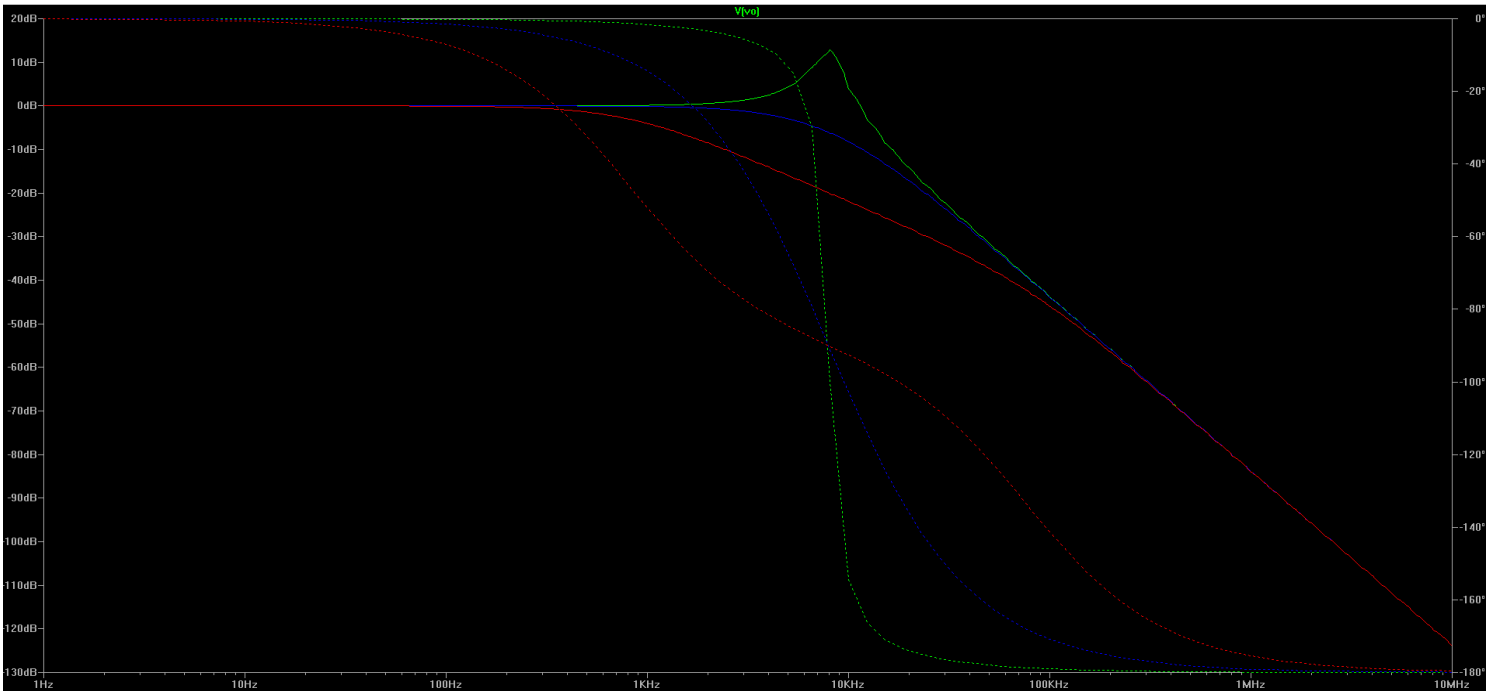
Transient analysis plot

Frequency Analysis of Series LRC Circuit: -



Schematic for AC analysis

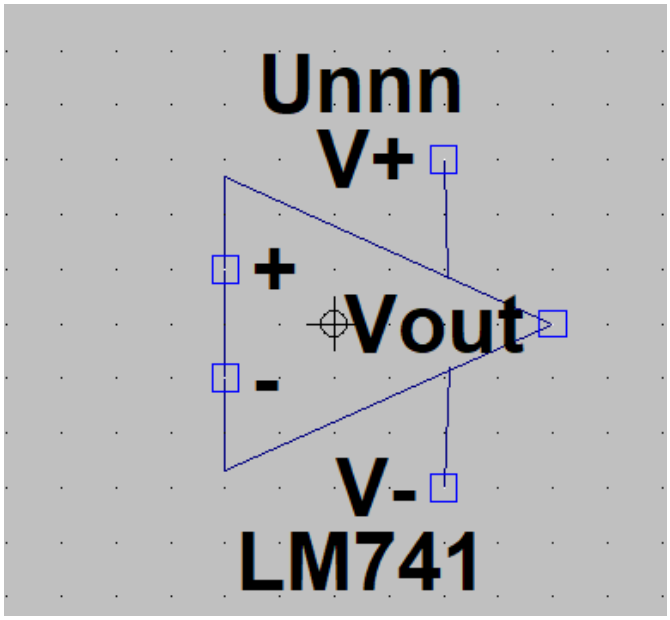
I have used a similar circuit (LRC instead of RLC), performed AC analysis using decade sweep from 1Hz to 10 MegHz, and parameter sweeps to represent 3 cases for showing bode plots of over, under and critically damped cases.



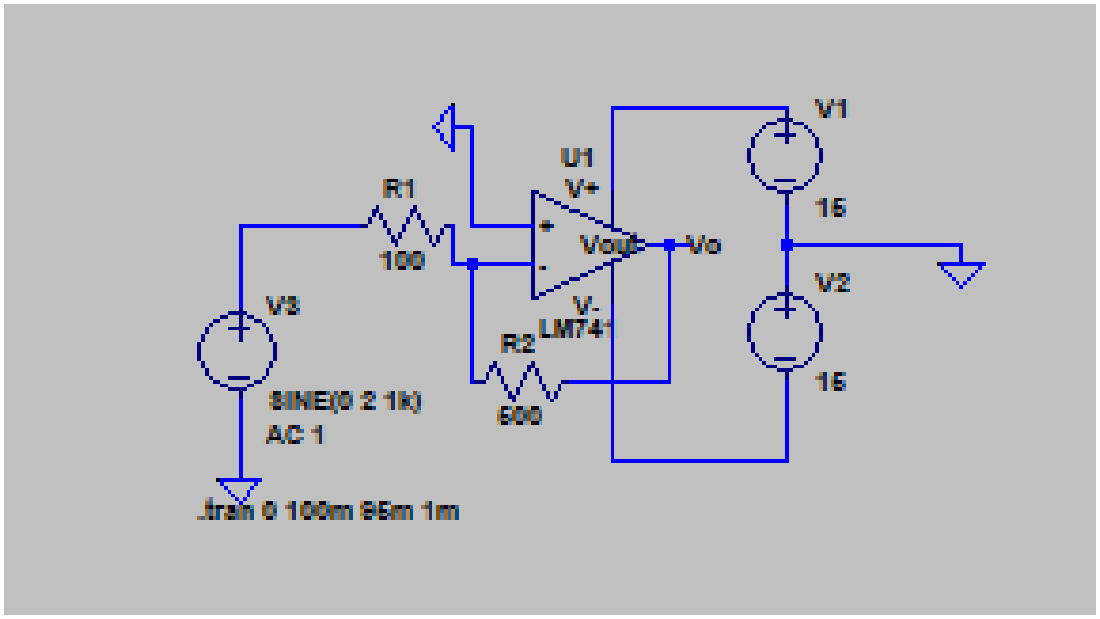
Bode plots

Inverting Op-Amp: -

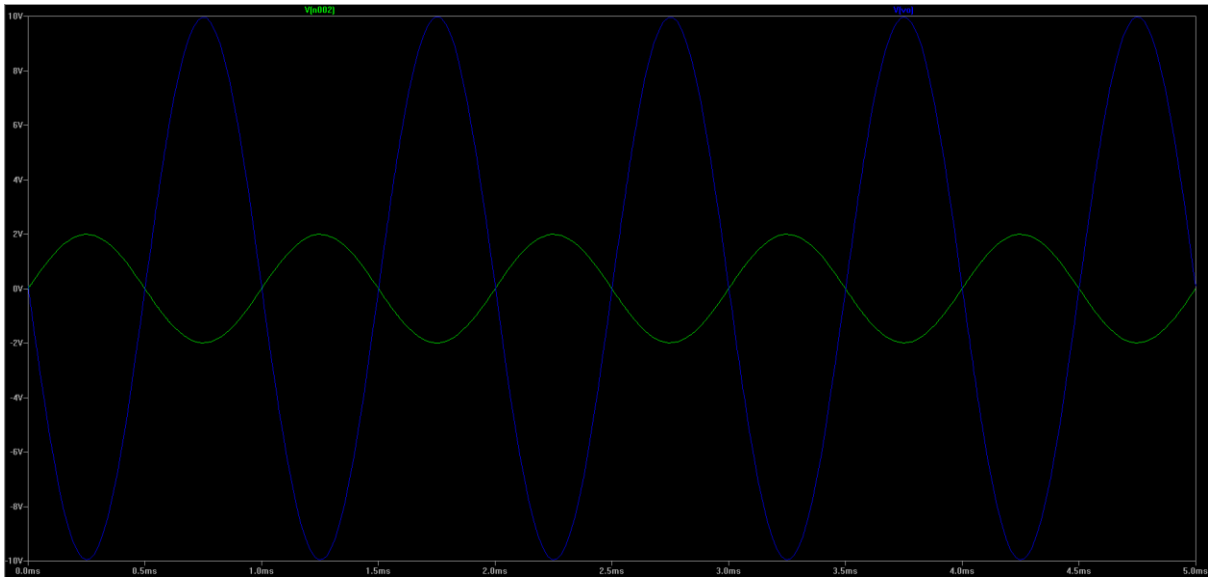
I have utilised the LM741 Op-Amp here. Since it is not available in the standard LTSpice library I had to download its file from the TI website and restructure the schematic making it visually easier to understand.



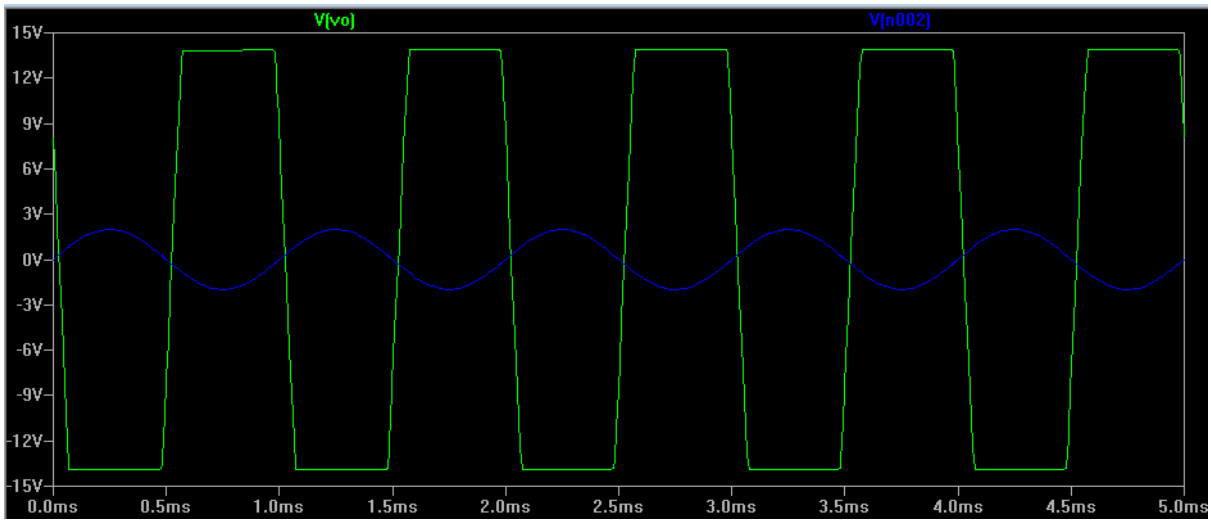
Created schematic for LM741



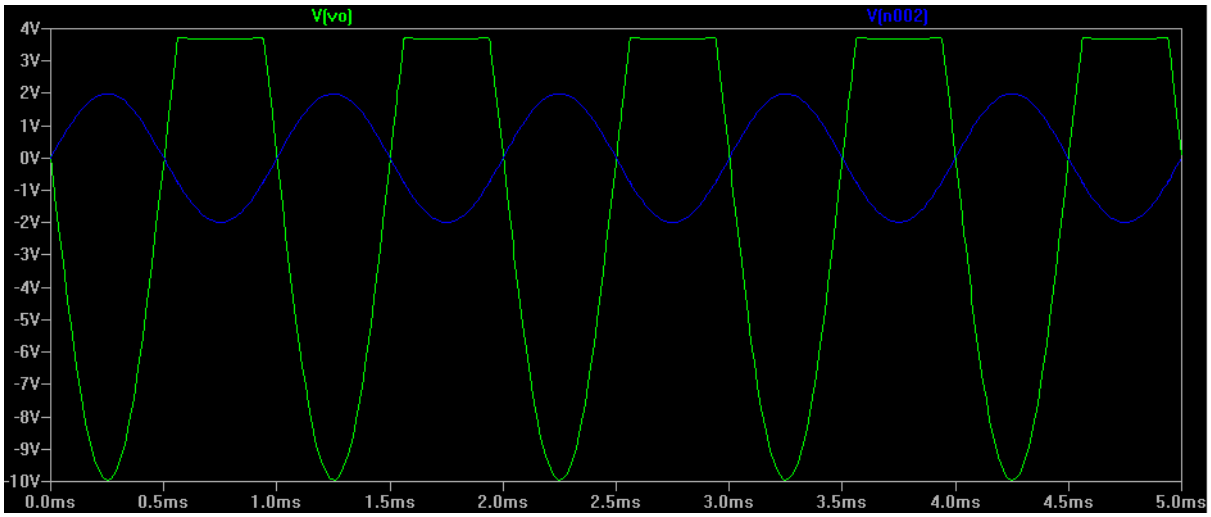
Schematic for inverting Op-Amp



Waveform for inverting Op-Amp

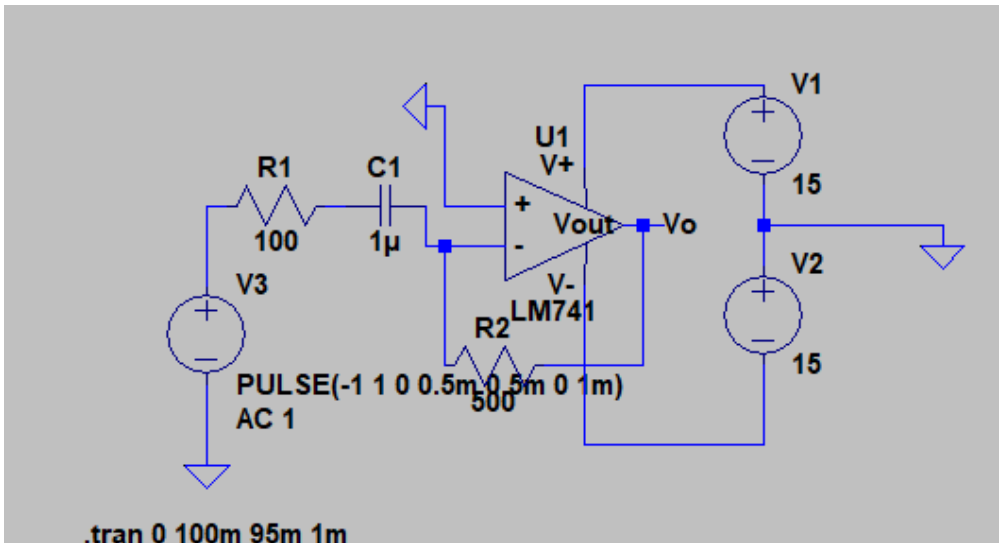


Clipping when gain is excess

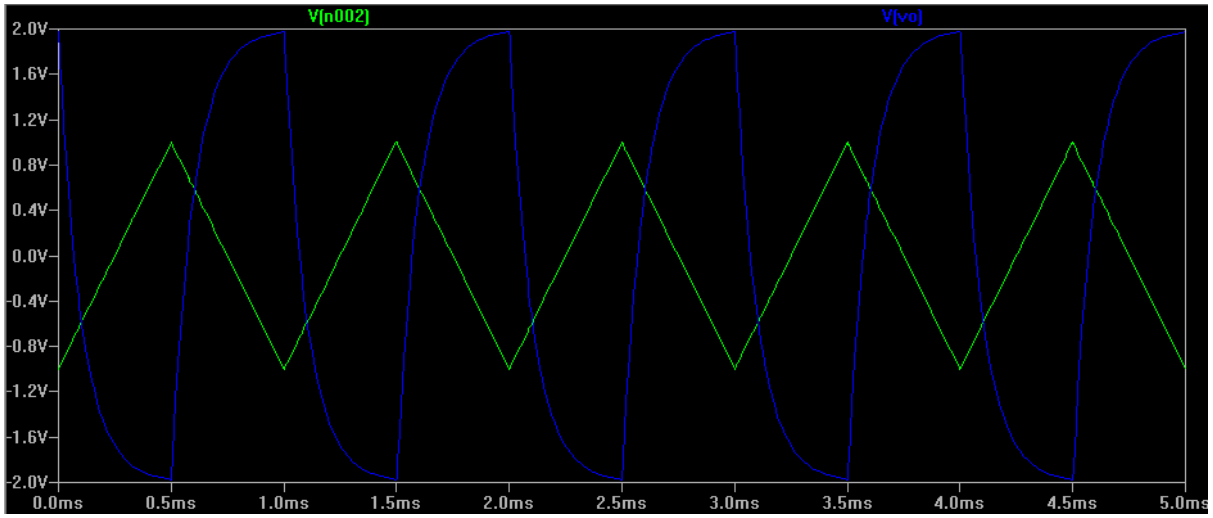


Positive clipping when Vcc+ is reduced

Differentiator Op-Amp: -



Differentiator Op-Amp Schematic


V(n002)
V(vo)
2.0V
1.6V
1.2V
0.8V
0.4V
0.0V
-0.4V
-0.8V
-1.2V
-1.6V
-2.0V
0.0ms
0.5ms
1.0ms
1.5ms
2.0ms
2.5ms
3.0ms
3.5ms
4.0ms
4.5ms
5.0ms

Waveform