

ELECTRONICS SIMULATION

Hardware camp 2025

Thanh Sang

Lab supporter, Neutrino Lab, IFIRSE, ICISE

LTSPICE INTRODUCTION

1) Powerful, Fast, Free Simulator

2) Using PICE Model

3) Graphical Schematic Capture Interface

4) Supported By Analog Devices

Keyboard shortcuts

SCHEME AND WAVEFORM EDITING SHORTCUTS

Windows	Place Components*	Apple
W	wire	F3
G	ground	G
CG	com	
V	voltage	V
R	resistor	R
C	capacitor	C
L	inductor	L
D	diode	D
P	component	F2
N	label net	F4
T	text/comment	T
	spice directive right-click text field to open "Help me Edit" dialog	
B	bus tap	B
left-click	toggle directive/comment	

*Press **ESC** or right-click to exit mode.

Windows	Schemes, Waveforms, Symbols	Apple
X or Backspace	delete	F5
C	copy/duplicate*	F6
M	move* select components to move	F7
S	stretch* select anchor points to move	F8
R	rotate	R
E	mirror	E
	Schematic zoom area (drag over area) zoom in (click on scheme)	Zoom in and out with scroll wheel or use pinch on track pad
	Waveform zoom area is default mode	
	zoom out	
Space	zoom to fit, zoom extents	Space
G	toggle grid	
Z	undo	F3 or Z
Y	redo	F4 or Y

Choose mode first, then select component or waveform title.
*Press **ESC** or right-click to exit mode.

Windows	Edit Directives & Component Parameters	Apple
right-click >		
	edit directive with help	edit limited parameters
	edit directive directly	edit all parameters

Text preceded by an underscore, e.g. "_FAULT" is displayed with an overbar "FAULT".

Windows	Simulator	Apple
A	configure analysis	
R	run/pause	
S	stop	
L	view SPICE log	L
R	reset sim waveform T = 0	



Schematics can be edited even as a simulation runs.
Edits affect subsequent simulations.

Windows	Waveform Viewing	Apple
click or C	add cursor and see measure	click
L	label current cursor position	
C or M	clear all cursors	close measure dialog
click	highlight corresponding net in schematic	click
click	integrate	click
drag	move trace (to another pane)	drag
drag, hold	copy trace (to another pane)	
A	add trace	
P	add pane above	
B	add pane below	
U	move active pane up	
D	move active pane down	
D	select steps	
D	recenter	

Mouse actions are on waveform trace label.

Windows	Waveform Pan & Cursor	Apple
	No Cursors pan -25%	
	Cursor Present snap cursor to next time data point	
	Cursor Present cycle cursors through traces at current time data point	
	Cursor Present snap cursor to next data point	
	No Cursors pan -30%	
	Cursor Present bump cursor 10 data points	
	Cursor Present bump cursor 100 data points	
	pan with mouse	
	pan left and right with mouse	
	pan up and down with mouse	

Click in waveform pane to apply keyboard functions to active frame.

Fast • Free • Unlimited

©2024 Analog Devices, Inc. All rights reserved. Trademarks and registered trademarks are the property of their respective owners. Ahead of What's Possible is a trademark of Analog Devices. LTspice-9/24/24 | analog.com

Common Keyboard

W: wire

R: Resistor

C: Capacitor

V: Source

P: Pick up components from table

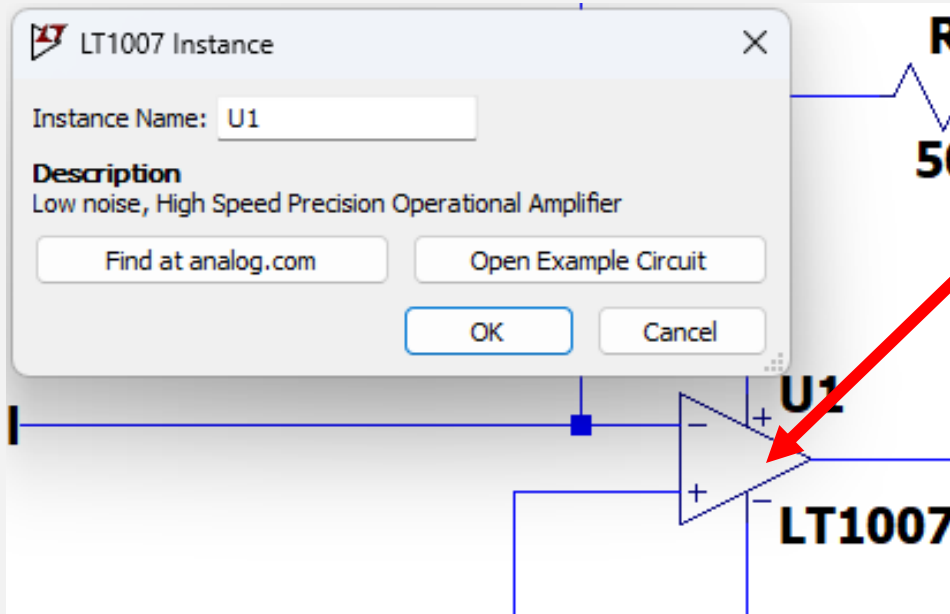
T: Spice Directive

Crt+R: Rotate component

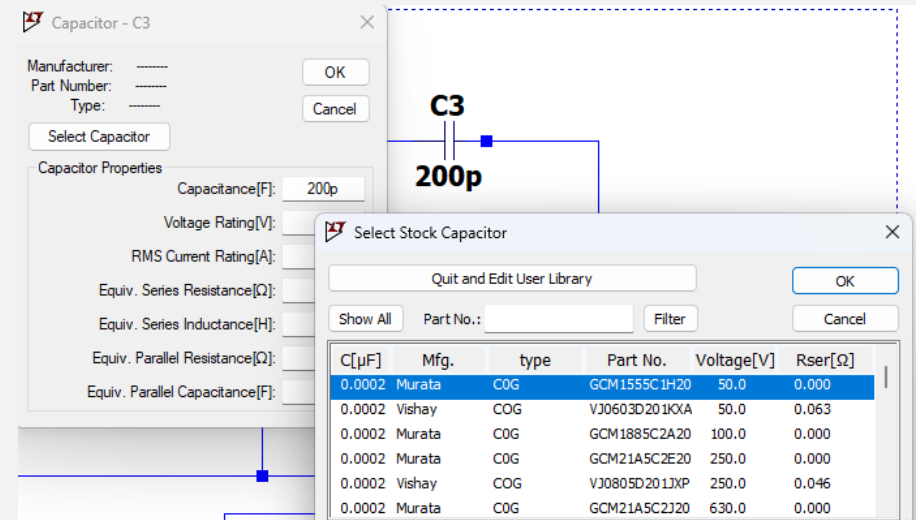
....

Almost of cases we use common keyboard

READING COMPONENT DOCUMENTATION AND CHOOSE TYPES OF COMPONENT



Right click
on component

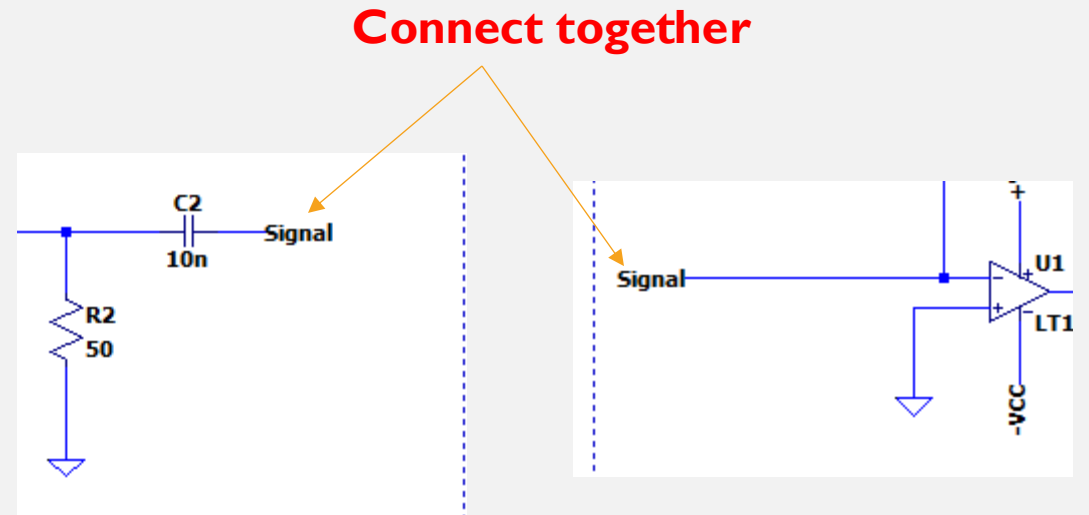


NETWORK LABEL

In **LTspice**, a **network label** (or **Net Label**) is a way to assign a custom name to a node in your circuit. This makes the schematic easier to read and allows for easy referencing of signals, especially when working with complex circuits.

How to Use Network Labels in LTspice:

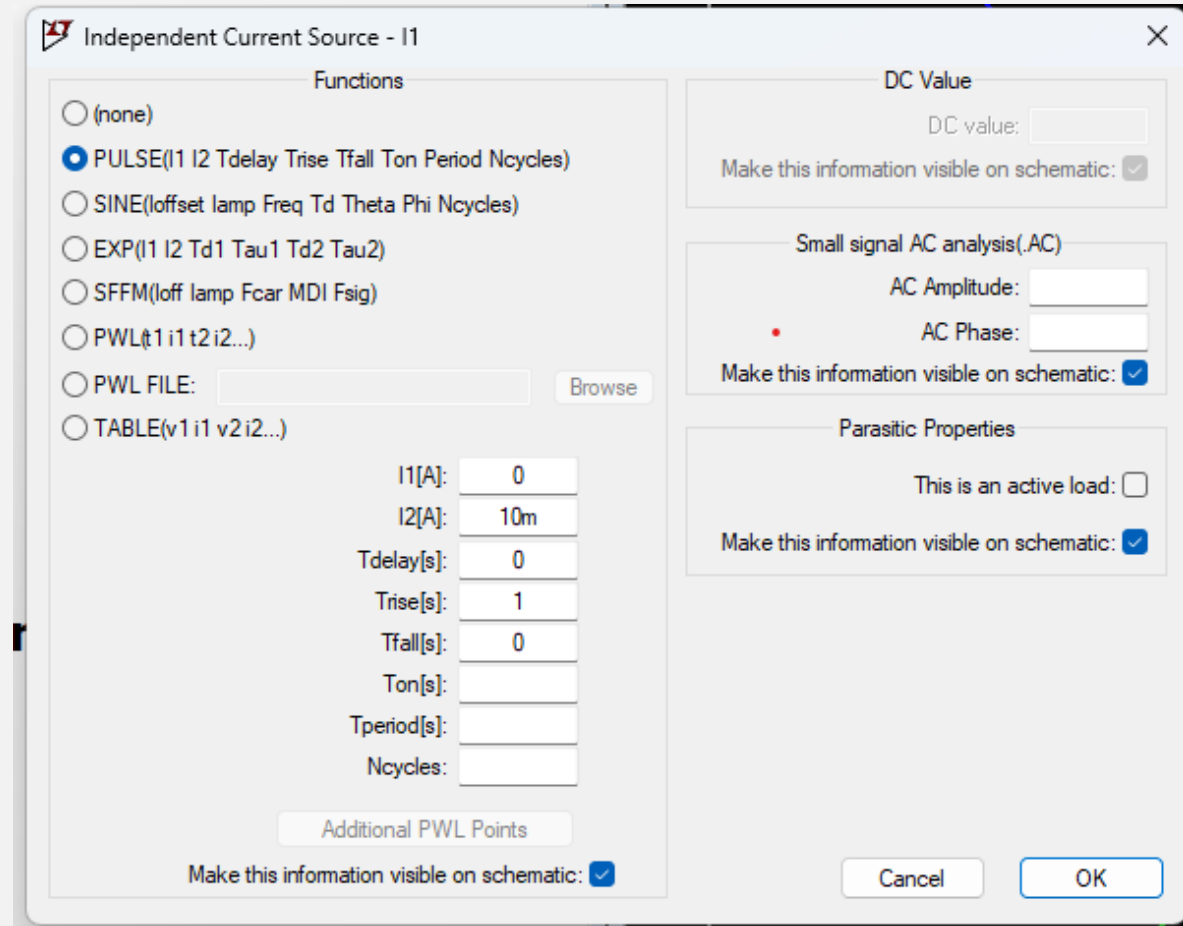
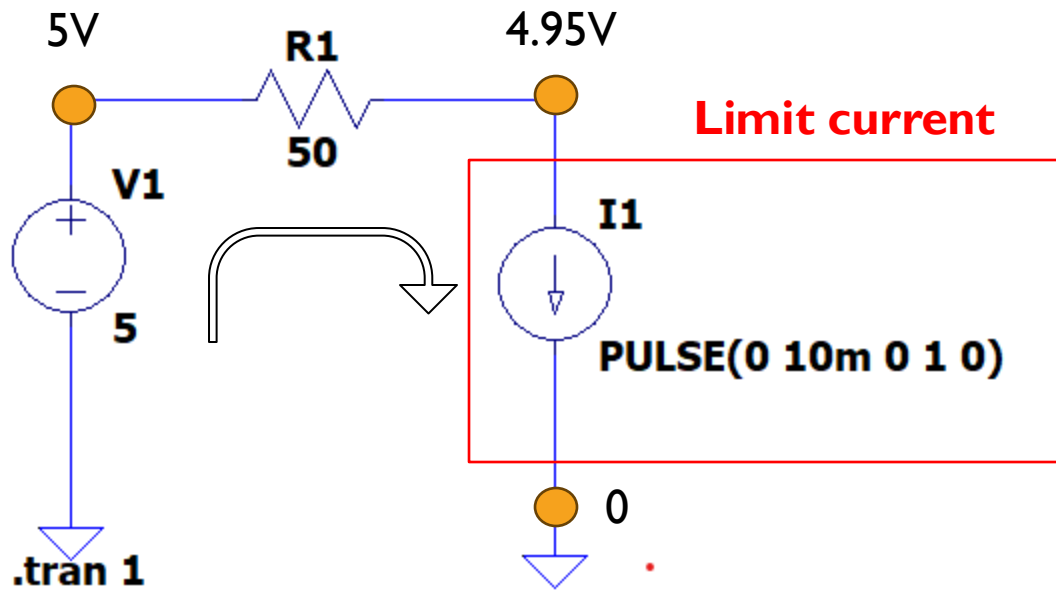
1. Open LTspice and your schematic.
2. Select the "Label Net" Tool:
Press F4, or Click on "Edit" → "Label Net" in the menu bar.
3. Enter a Name for the node.
4. Click on the Wire where you want to place the label.
5. Repeat for other nodes if needed.



SIMULATOR DIRECTIVES — DOT COMMANDS

- Whereas the circuit topology is typically schematically drafted, the commands are usually placed on the schematic as text. All such commands start with a period and are called "dot commands".
- .tran : , and examples...
- .step :
- Syntax: .step param List <value1> <value2> <value3>
- Syntax: .step param <minvalue> <maxvalue> <value_step>
- .text :
- .wave :
- .text :
- .save :

SOURCE AND CURRENT

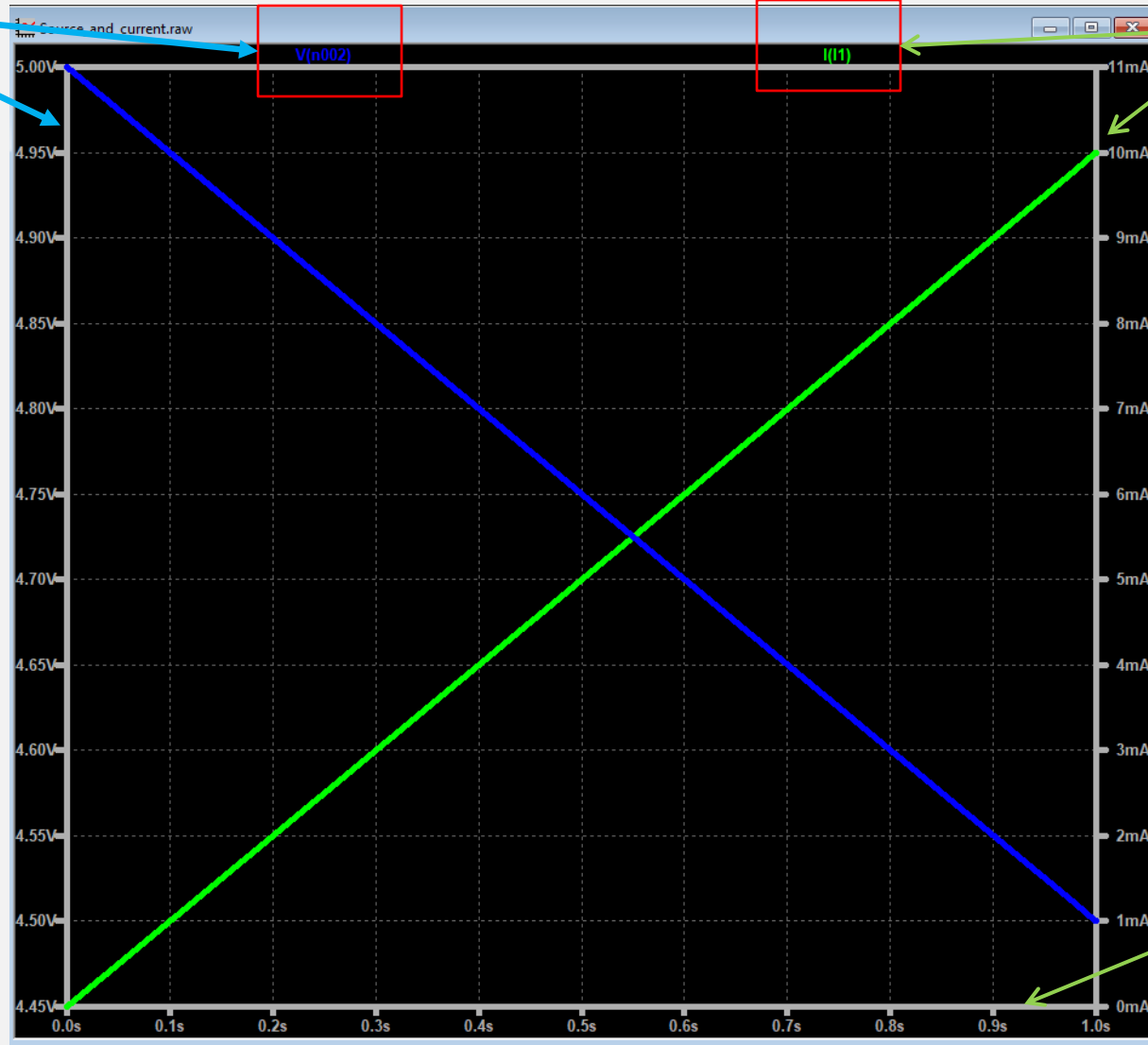


Limit current = 10m

⇒ Through resistor, voltage drops to 4.95V.

SIMULATION TRACE

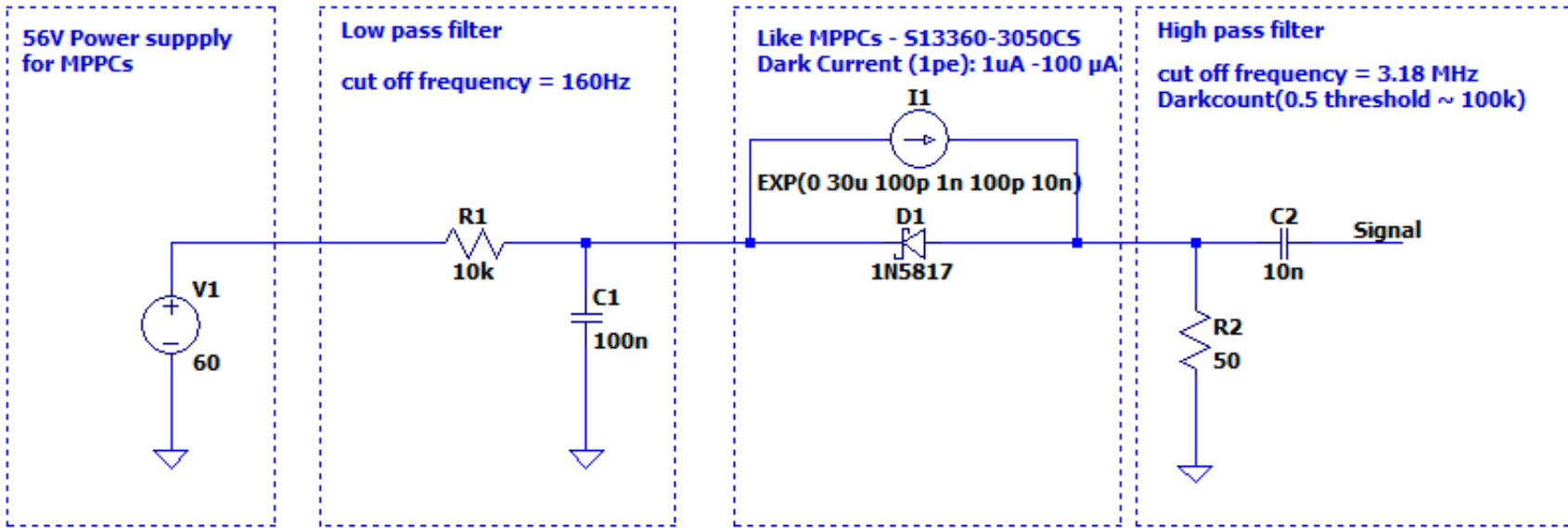
Volts axis
blue line



Current axis
green line

Time

SIMULATION MPPCS SIGNAL



For times less than rise delay time, the output current is I_1 .
For times between T_{d1} and T_{d2} the current is given by:

$$I_1 + (I_2 - I_1) \exp((t - T_{d1}) / \tau_1)$$

For times after T_{d2} the current is given by:

$$I_1 + (I_2 - I_1) (\exp((T_{d2} - t) / \tau_2) - \exp(T_{d1} / \tau_1 - t))$$

Name	Description	Units
I1	Initial value	A
I2	Pulsed value	A
Td1	Rise delay time	sec
Tau1	Rise-time constant	sec
Td2	Fall delay time	sec
Tau2	Fall-time constant	sec

SIMULATOR DIRECTIVES — DOT COMMANDS

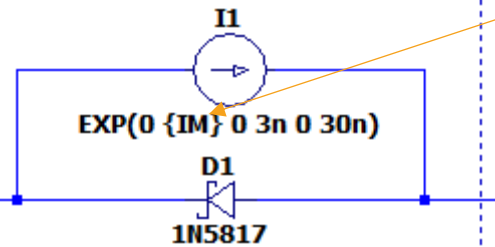
Code:

```
.tran 1m  
  
.step param IM LIST 30u 60u 90u
```

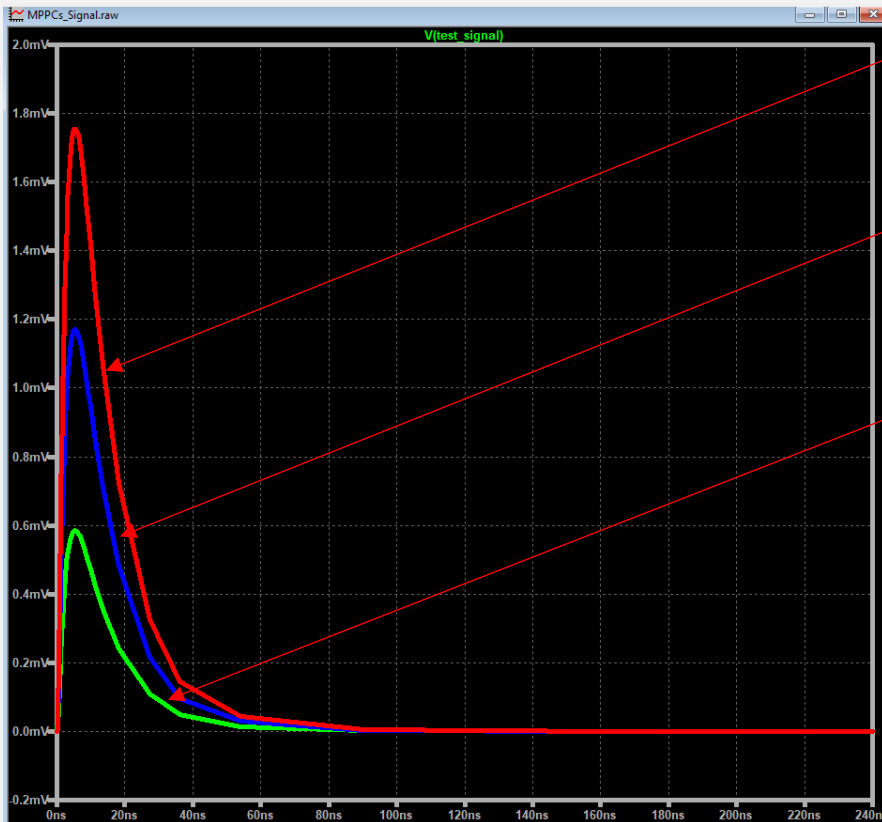
Syntax:

```
.STEP PARAM <ParameterName> LIST <value1> <value2> <value3> ...
```

Like MPPCs - C13551
Dark Current (1pe): 1uA - 100 μ A



Set
Parameter
Inside braces.
 $\{IM\}$



(3pe) $IM=60\mu\text{A}$

(2pe) $IM=40\mu\text{A}$

(1pe) $IM=20\mu\text{A}$

PREAMPLIFIER – CHARGE AMPLIFIER

When a photon hits the SiPM, it triggers an avalanche multiplication process in the microcells, leading to a **flow of charge**. This charge is then detected as a current or voltage signal.

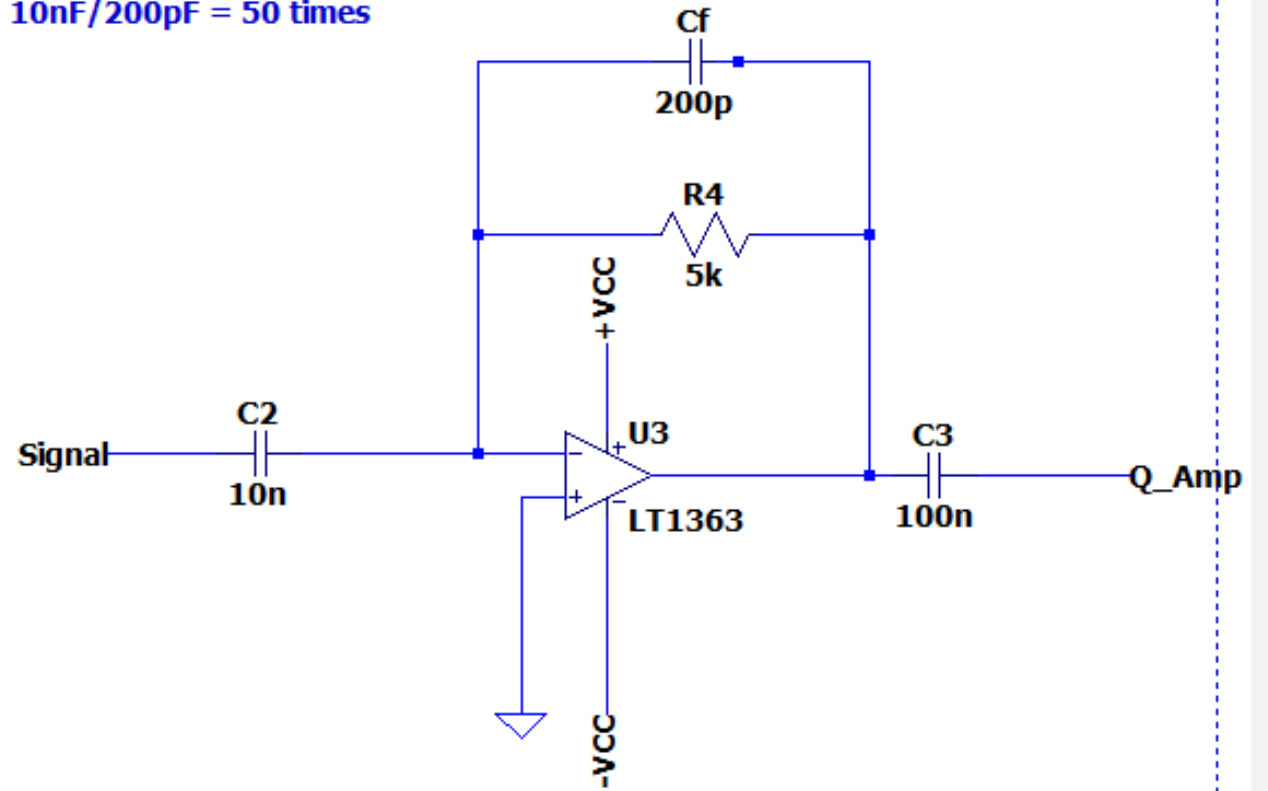
Charge Amplifier:

- + Energy measurement (pulse integration, spectroscopy, low noise).
- + Converts charge (Q) to voltage (V)
- + Stable gain

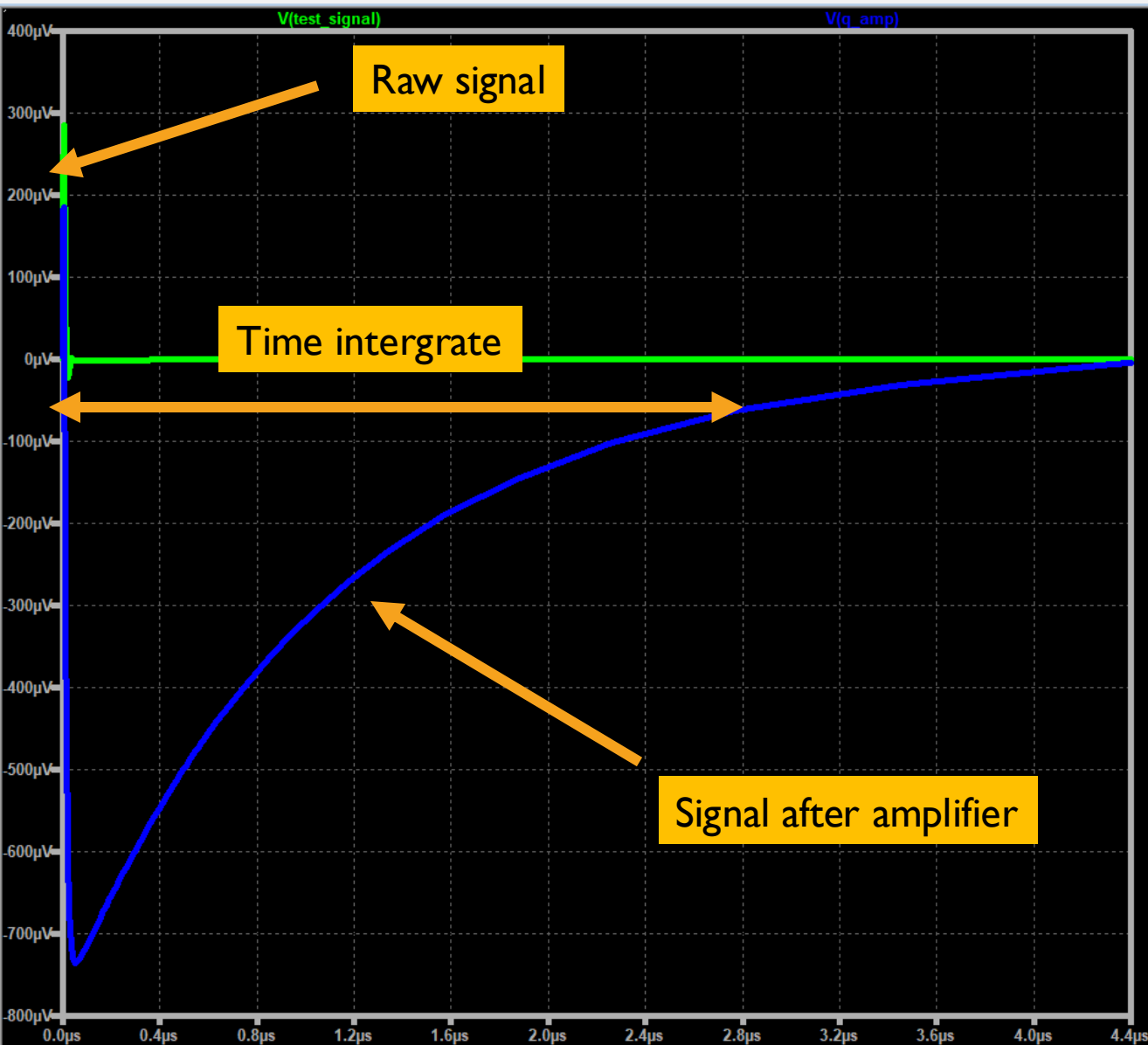
PREAMPLIFIER - CHARGE AMPLIFIER

Signal characteristic	LT1363 OP-AMP
1ns-rise time 10ns pulse handling (~40MHz bandwidth)	70MHz - Bandwidth.
0.5mV small signal amplification	1000V/ μ s High slew Rate.
Several ten μ A	2 μ A maximum Input Bias Current
Low output impedance	0.2mV Maximum offset voltage

Charge inverting amplifier:
 $10\text{nF}/200\text{pF} = 50$ times



CHARGE AMPLIFIER SIMULATION



Charge Amplifier

$$A = C_s / C_f$$

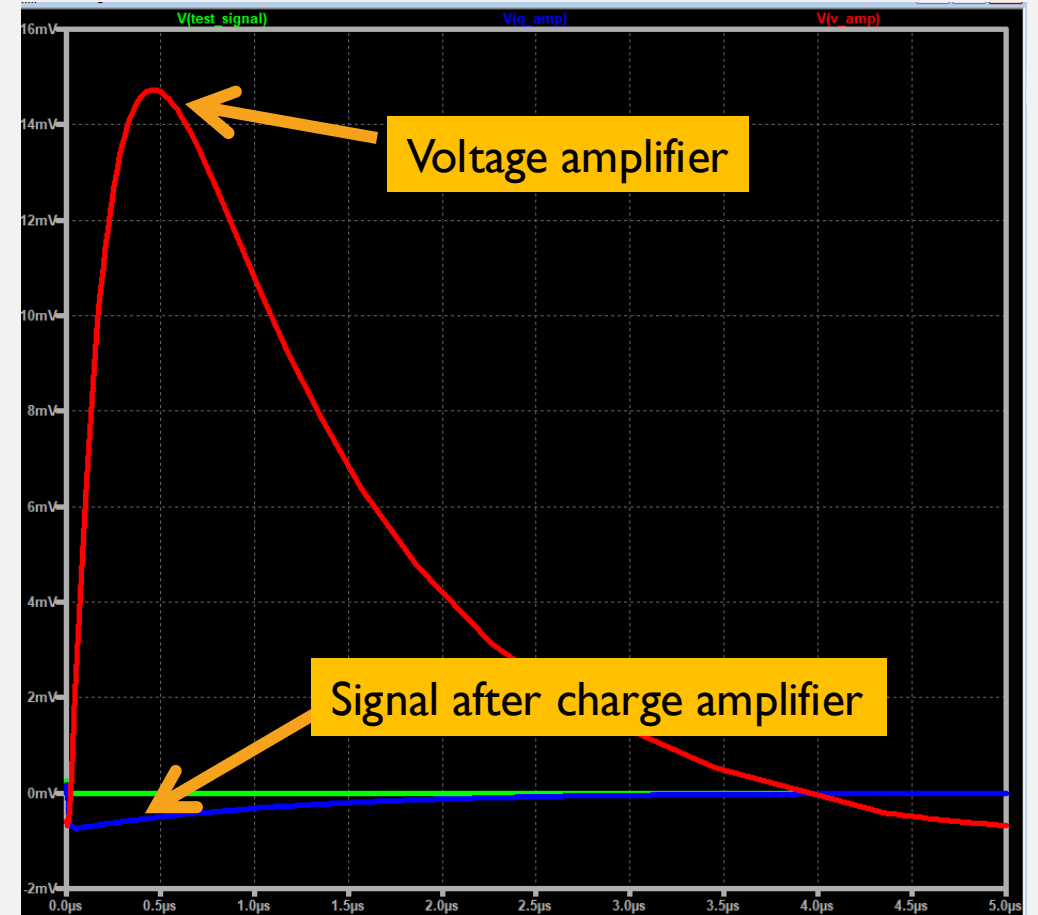
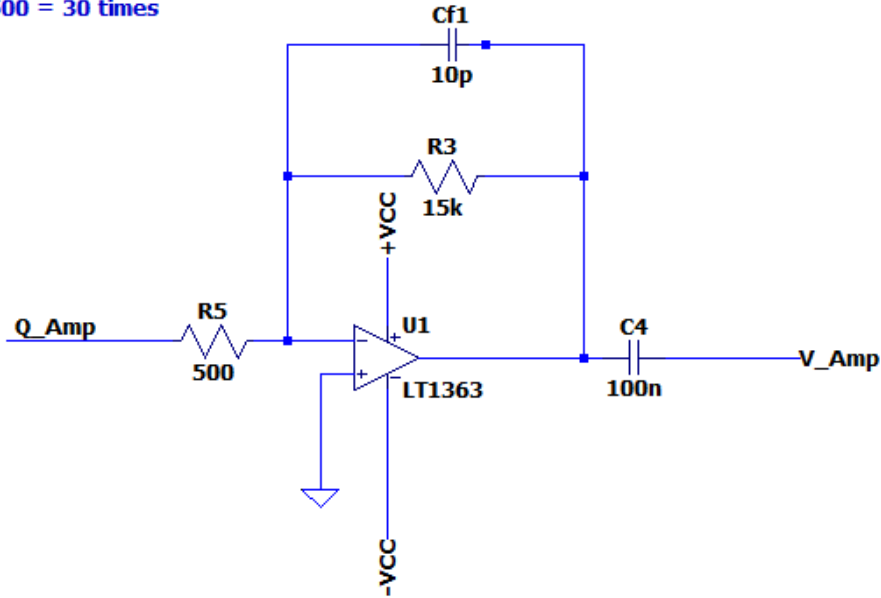
C_s : capacitance of signal

C_f : capacitance of feedback capacitor

VOLTAGE AMPLIFIER SIMULATION

Simulation result

Voltage inverting amplifier
 $15k/500 = 30$ times



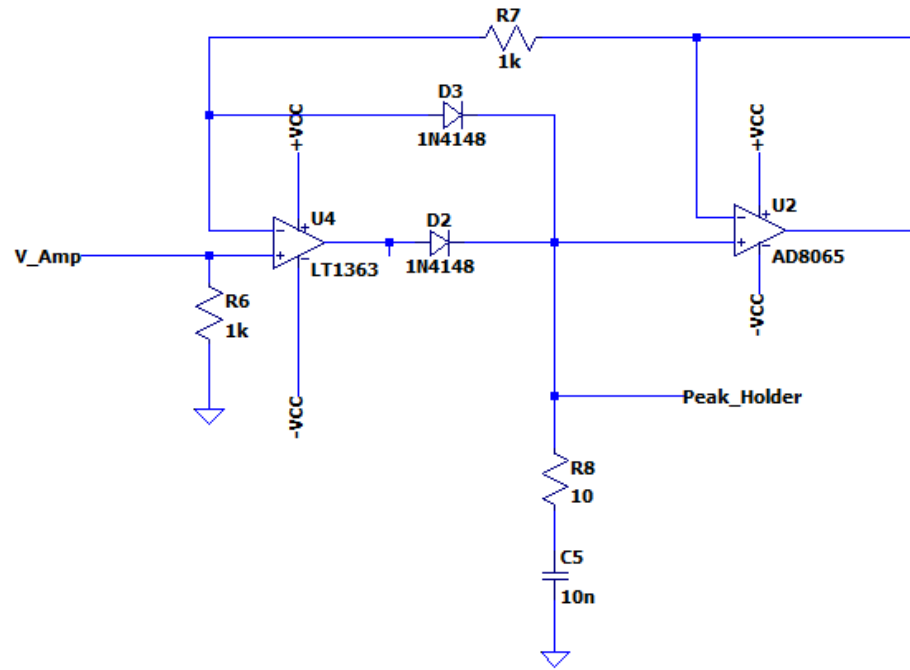
PEAK DETECTOR

Peak detector

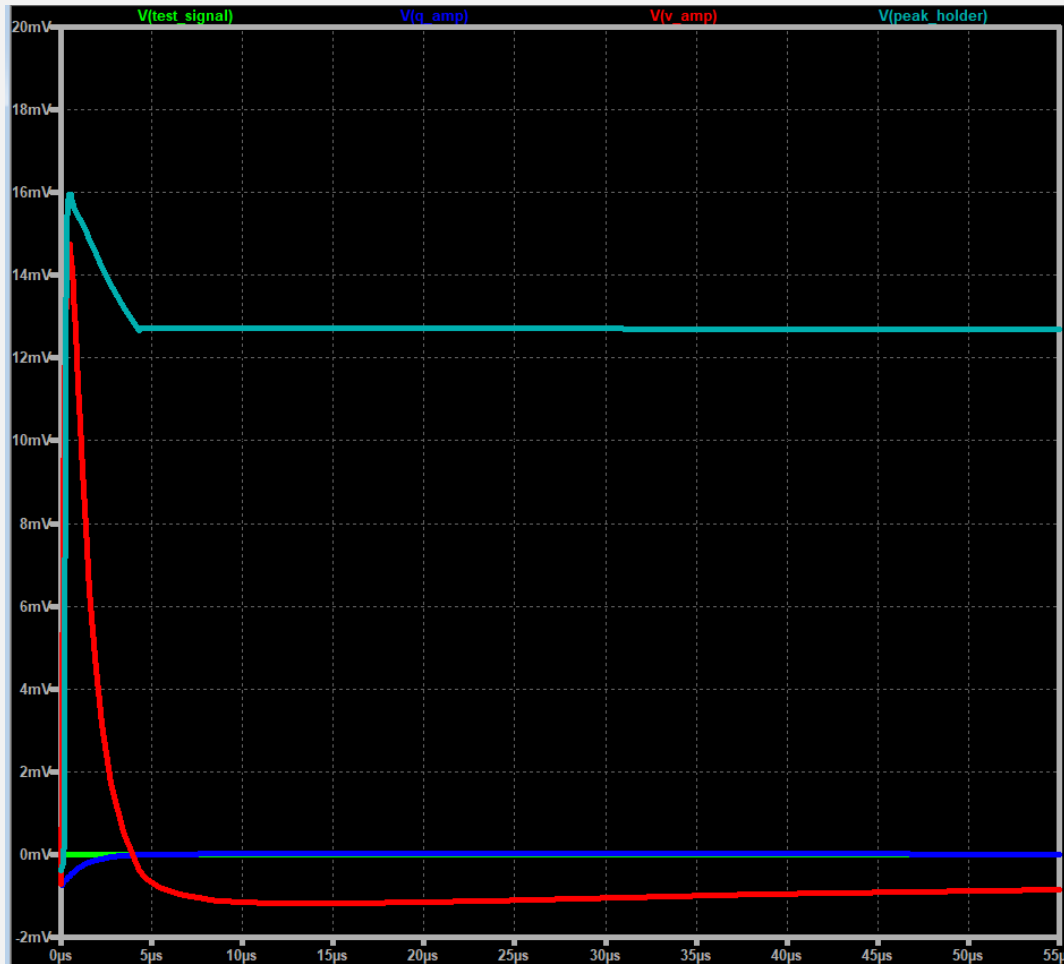
.step param R_Hold list 10 50 100

.step param C_Hold list 10n 100n 1u

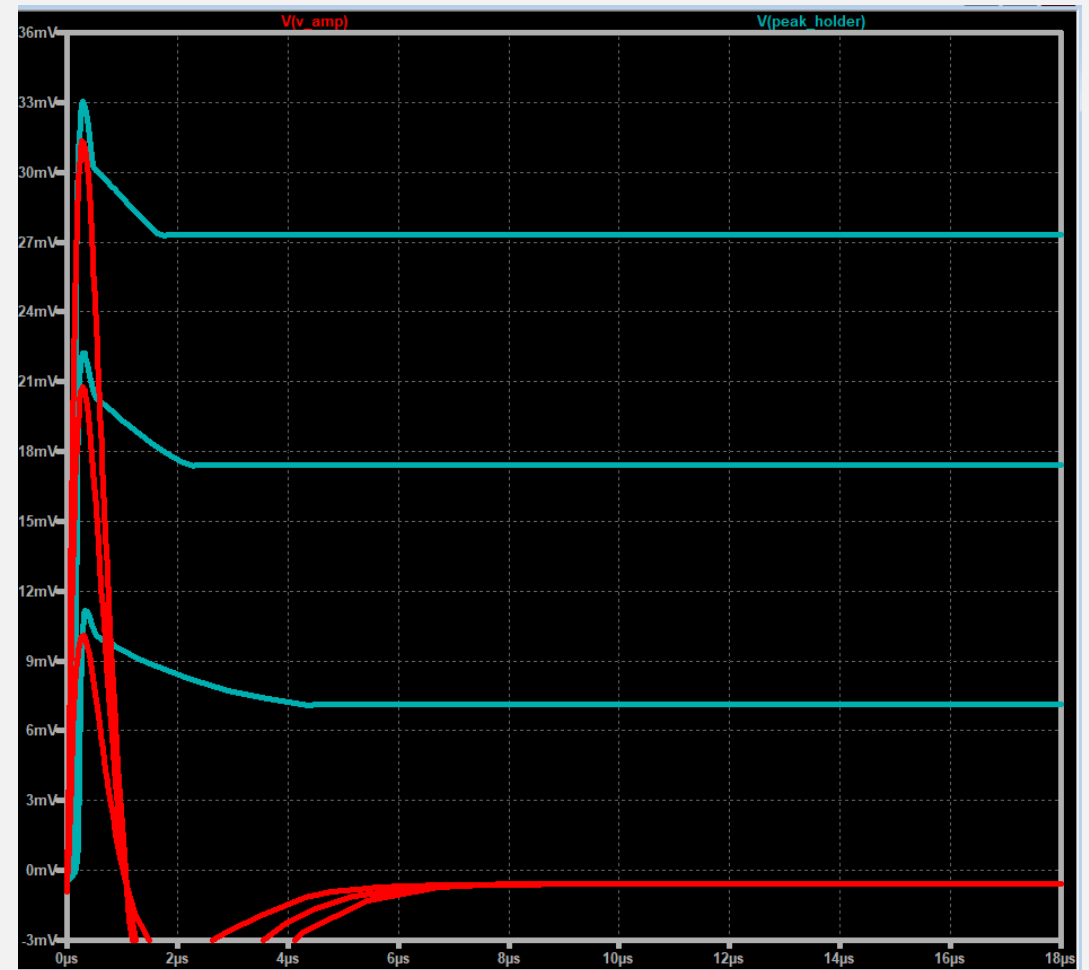
R7- hight peak
R and C hold - decay time



PEAK HOLD SIMULATION



Holding peak



Linearity of holding peak

BACK UP