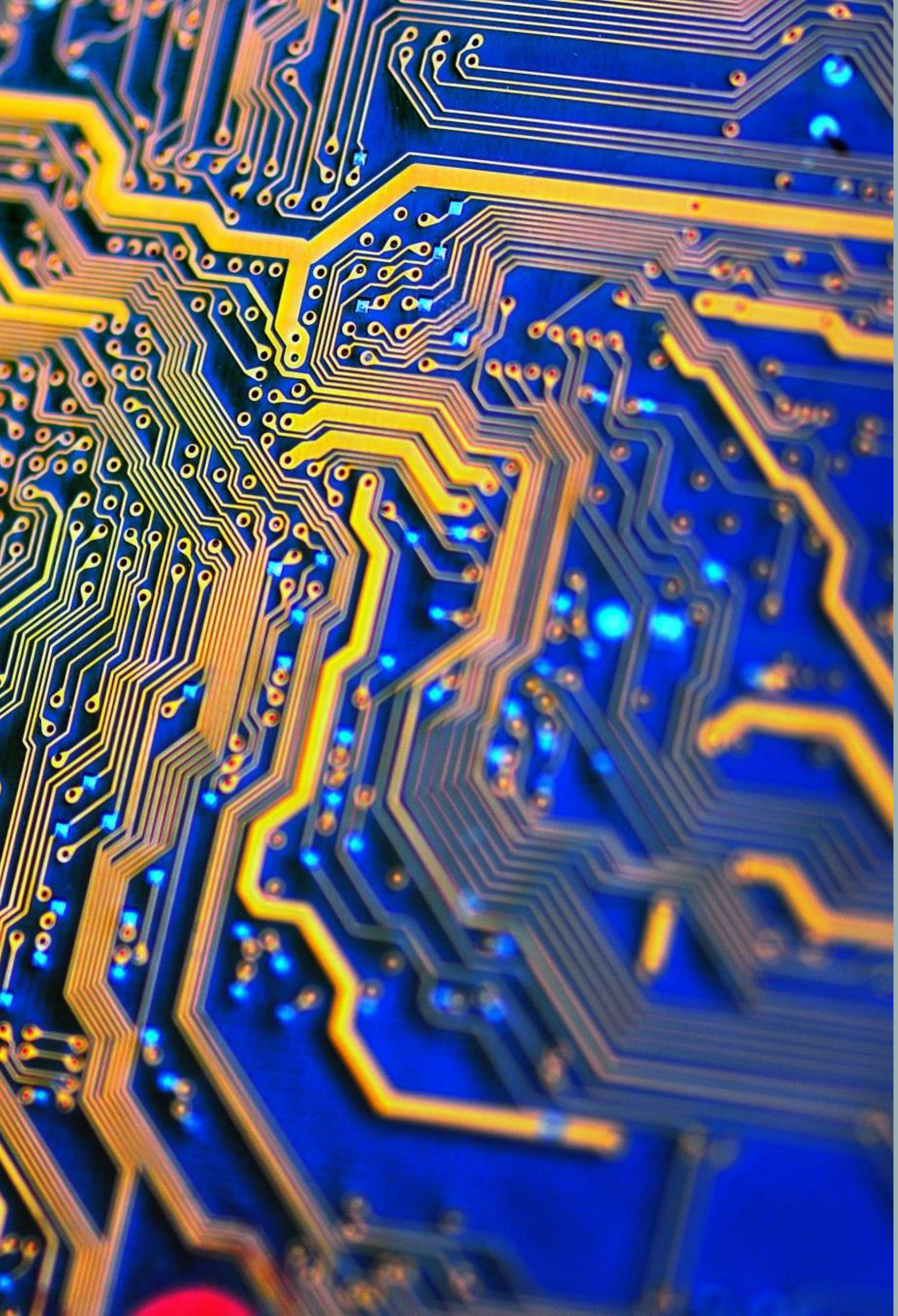


ELECTRONICS SIMULATION

Hardware camp 2025

Thanh Sang
Lab supporter, Neutrino Lab, IFIRSE, ICISE

LTSPICE INTRODUCTION

1) Powerful, Fast, Free Simulator

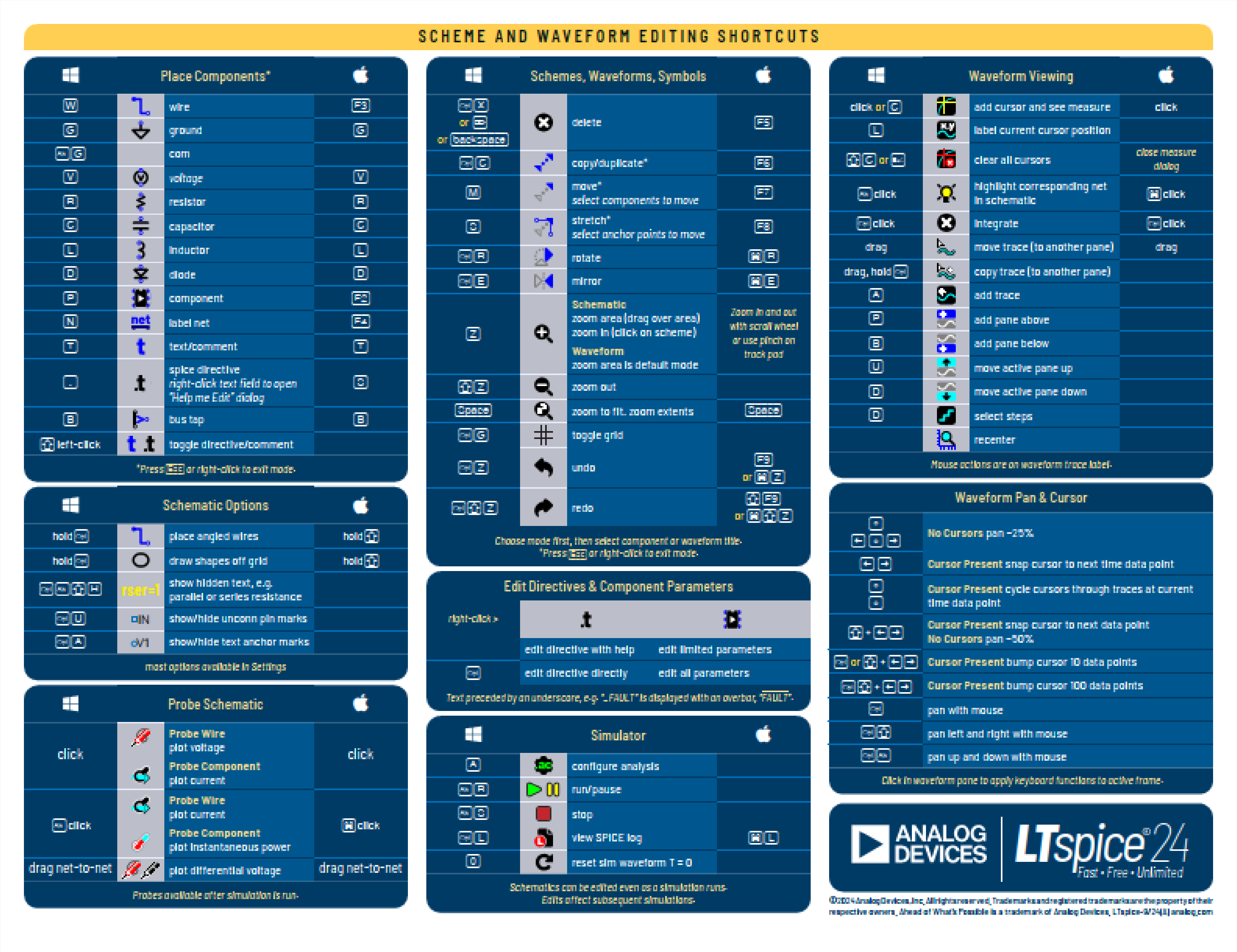
2) Using PICE Model

3) Graphical Schematic Capture Interface

4) Supported By Analog Devices

Link Guide: https://www.analog.com/en/resources/design-tools-and-calculators/ltspice-simulator/ltspice-recommended-reading-list.html

Keyboard shortcuts

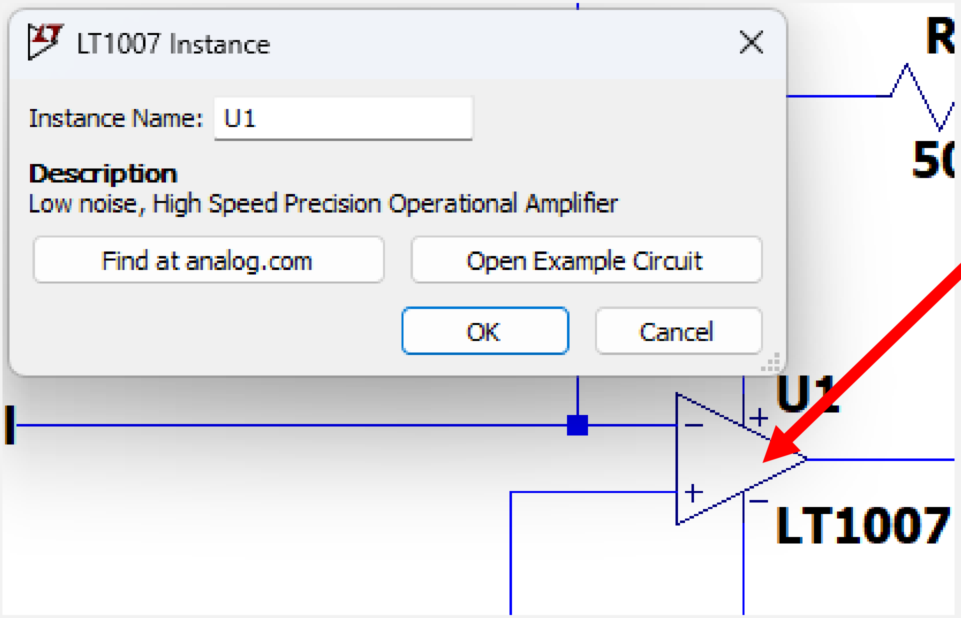


Common Keyboard

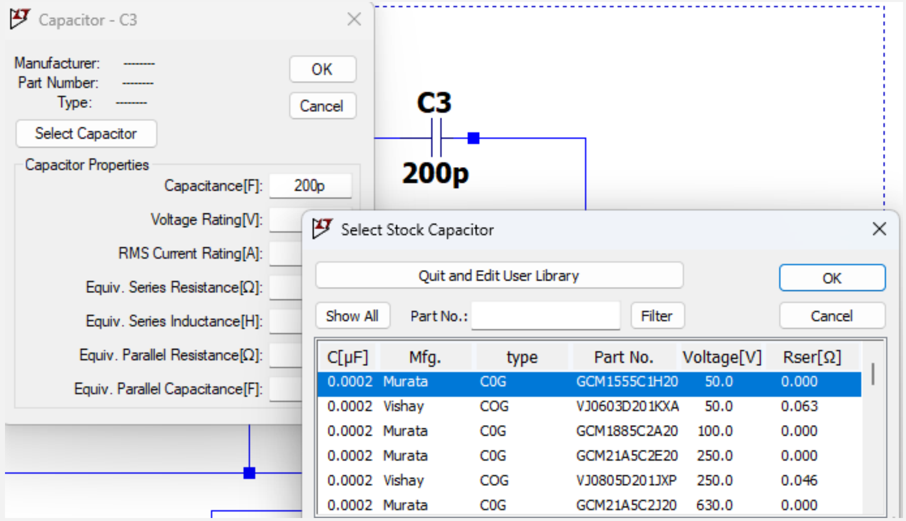
W: wire
R: Resistor
C: Capacitor
V: Source
P: Pick up components from table
T: Spice Directive
Crt+R: Rotate component
….

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:

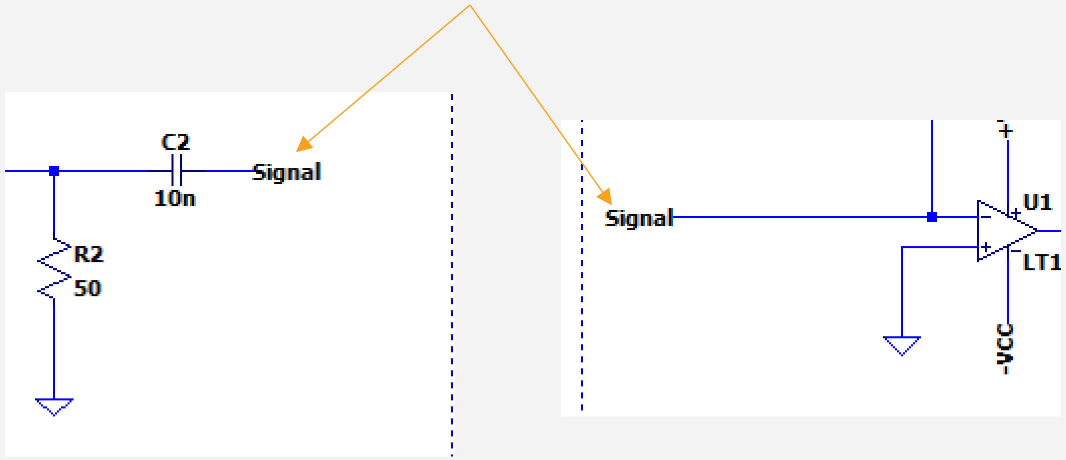
   Press F4, orClick on "Edit" → "Label Net" in the menu bar.

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.

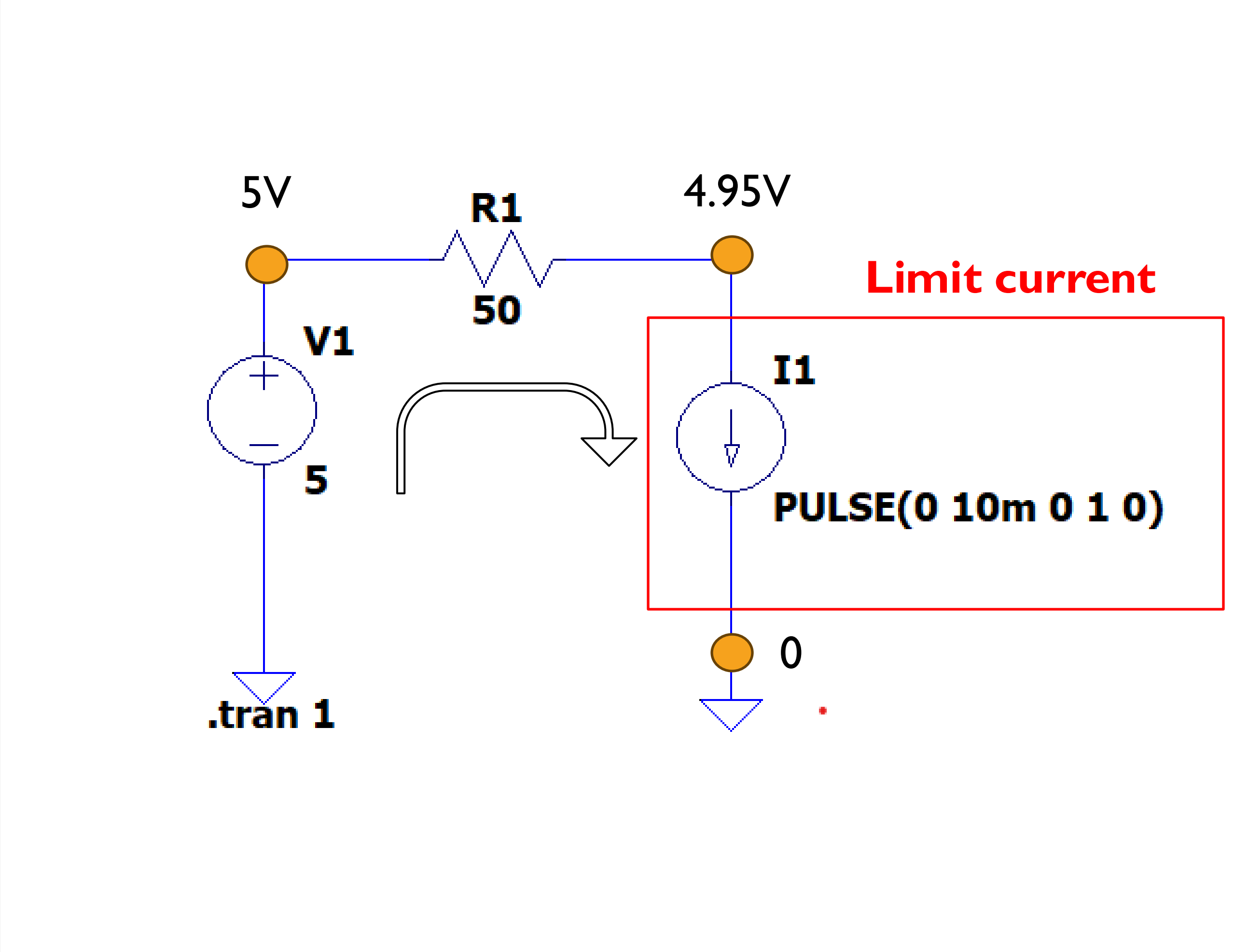
Connect together

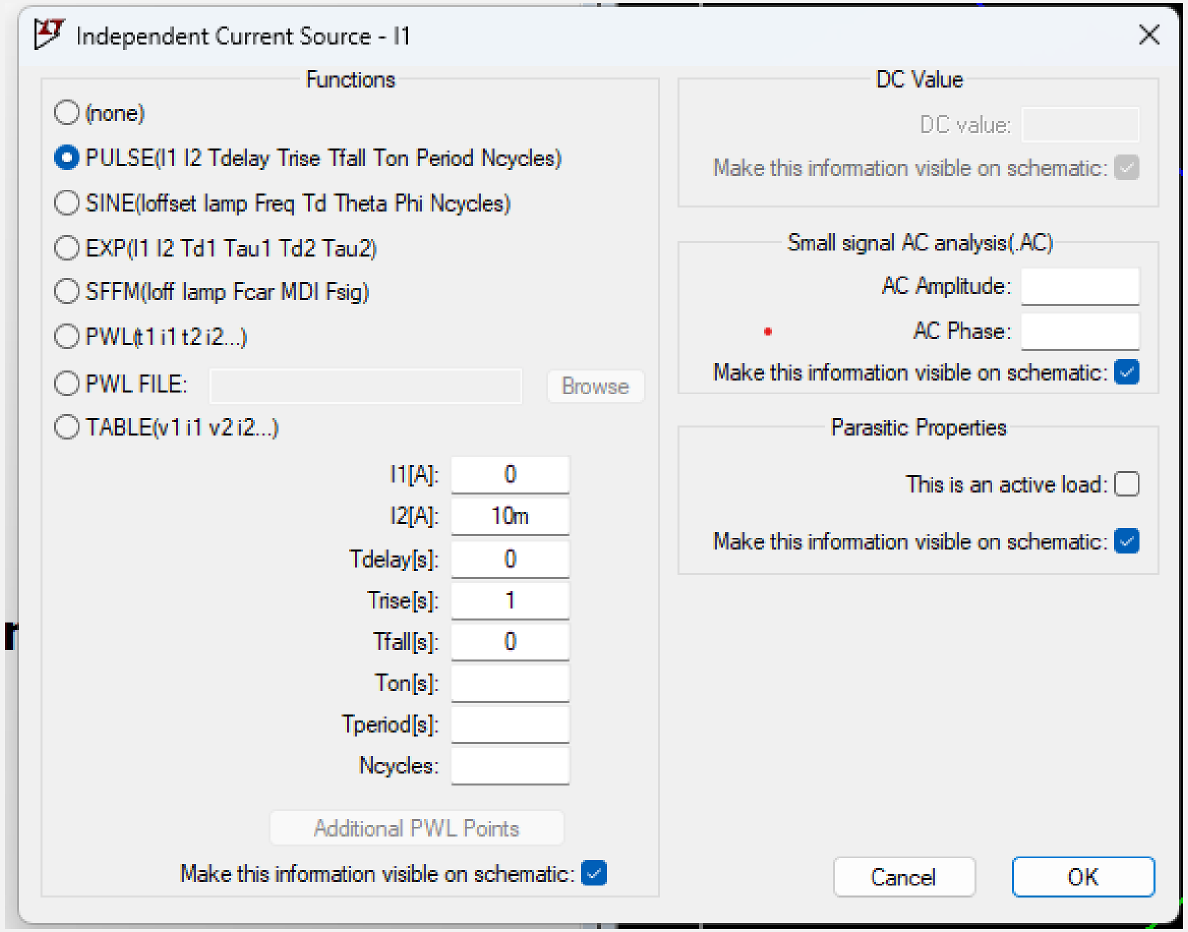


SIMULATOR DIRECTIVES — DOT COMMANDS

- 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



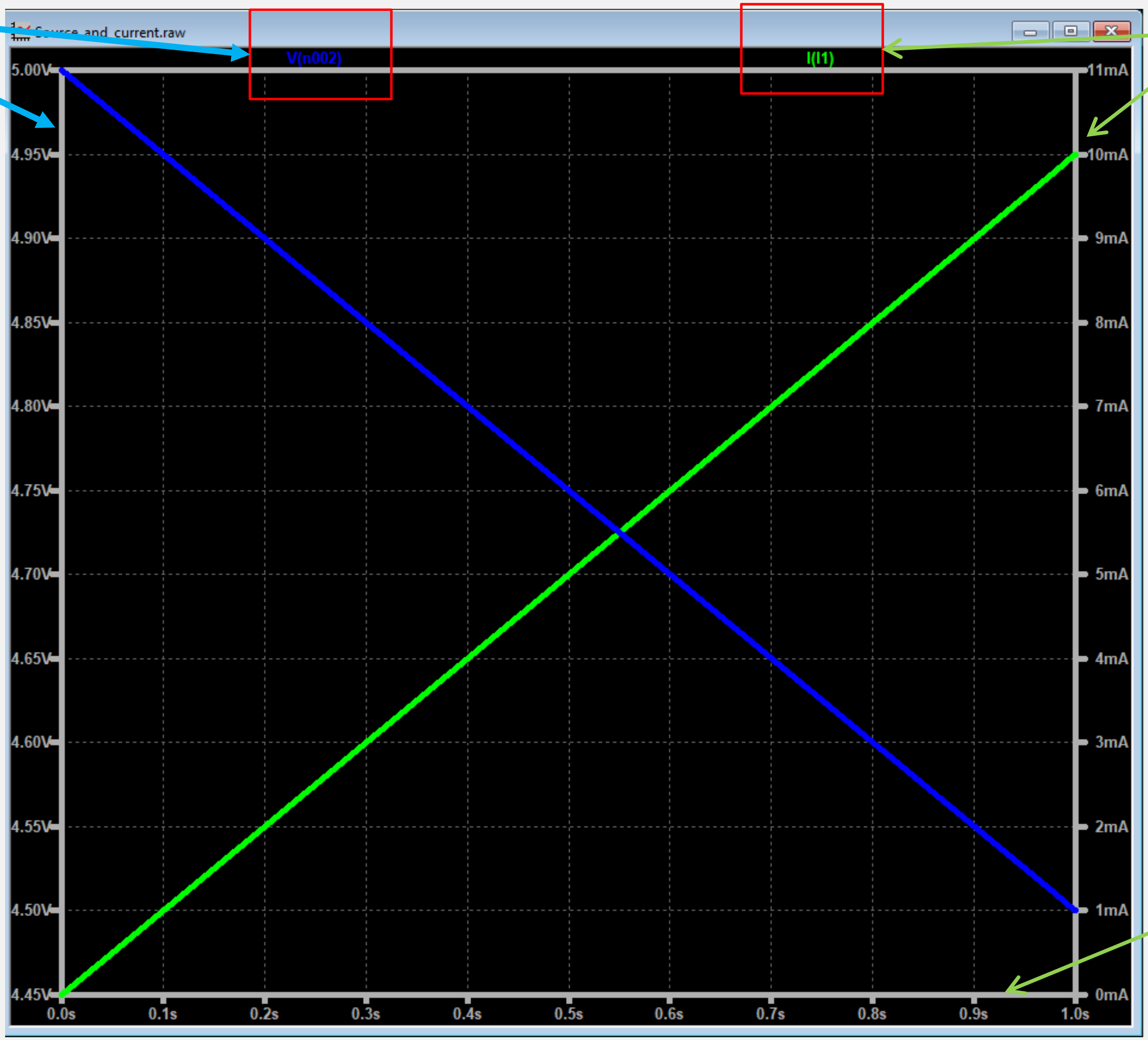


Limit current = 10m

⇒ Through resistor, voltage drops to 4.95V.

SIMULATION TRACE

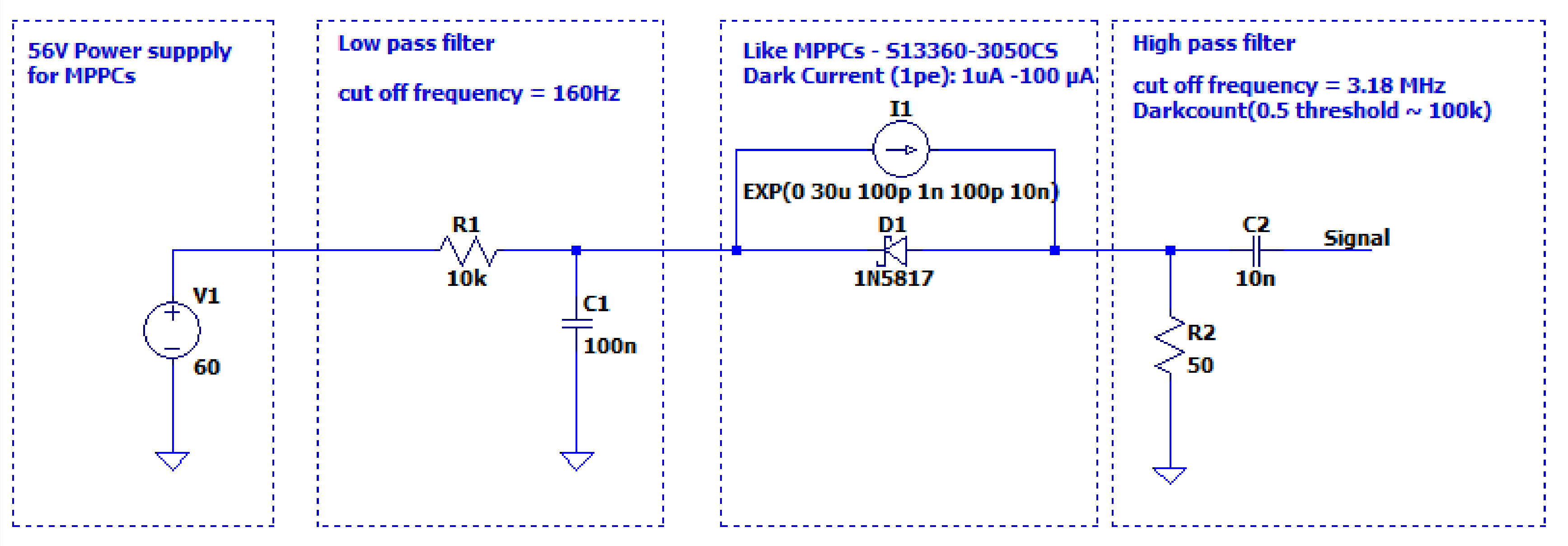
Volts axis
blue line



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

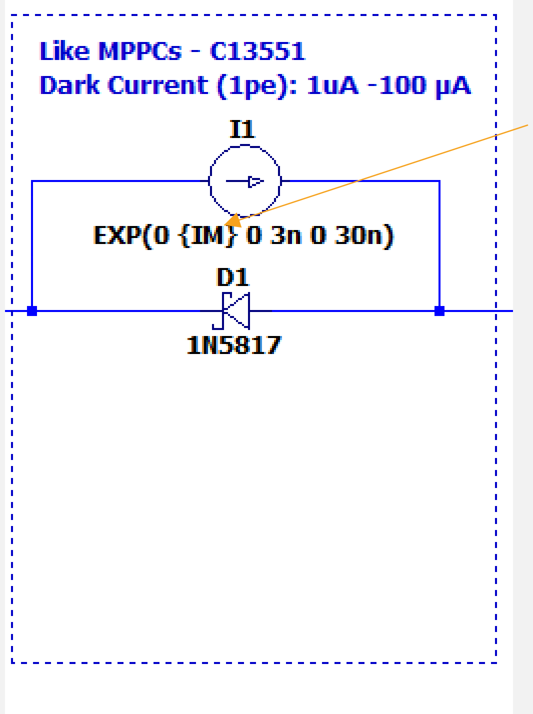
SIMULATOR DIRECTIVES — DOT COMMANDS

Code:

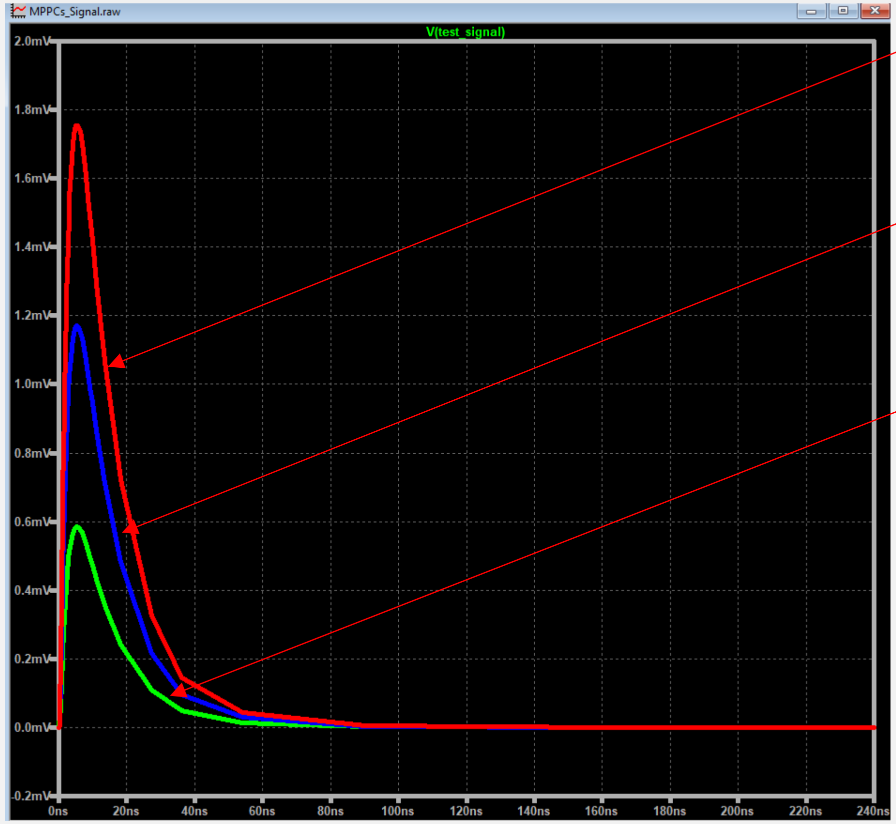
```
.tran 1m

.step param IM LIST 30u 60u 90u
```

Syntax:

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



Set
Parameter
Inside braces.
{IM}



(3pe) IM=60uA

(2pe) IM=40uA

(1pe) IM=20uA

PREAMPLIER – CHARGE AMPLIER

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:

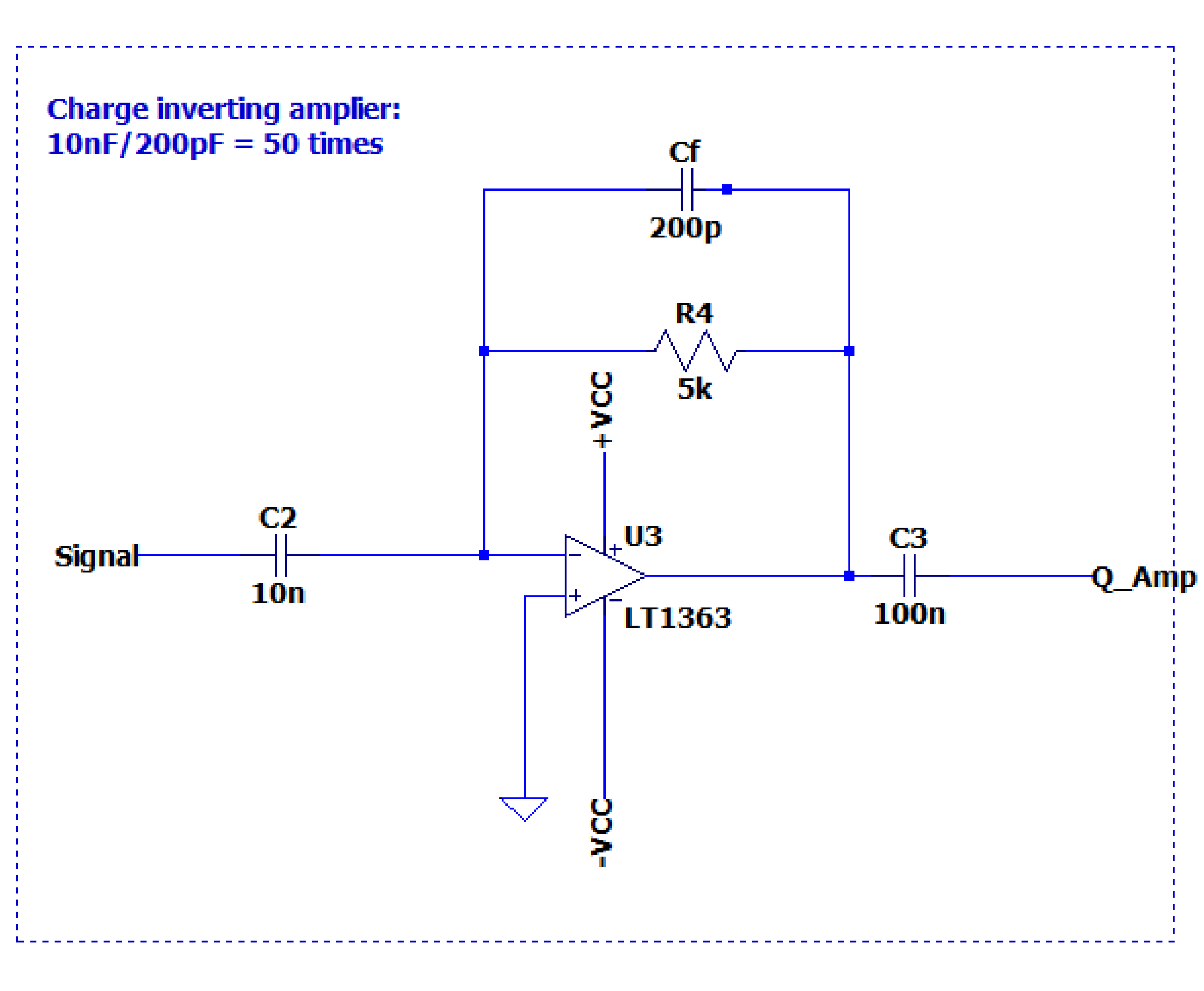
+ Energy measurement (pulse integration, spectroscopy, low noise).

+ Converts charge (Q) to voltage (V)

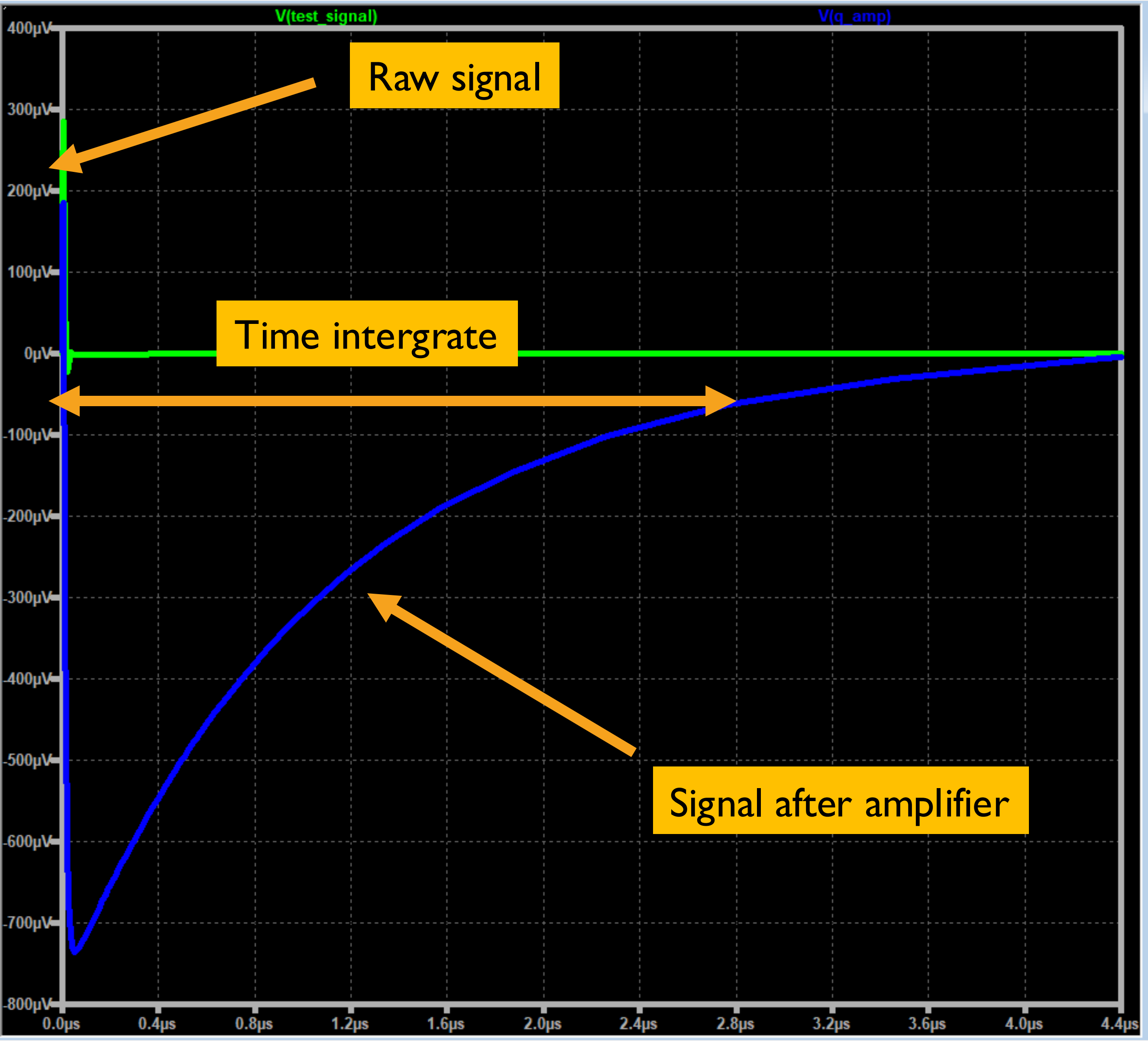
+ Stable gain

PREAMPLIER – CHARGE AMPLIER

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



CHARGE AMPLIFIER SIMULATION



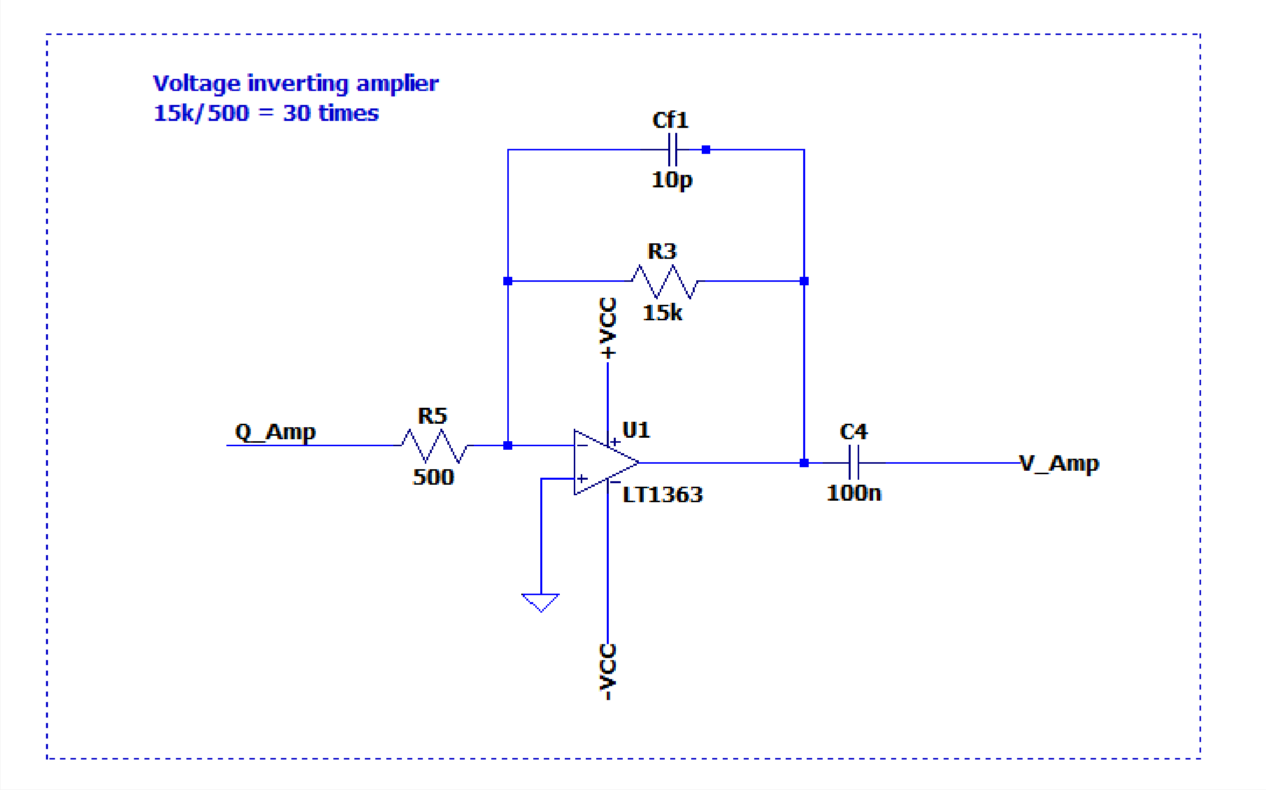
Charge Amplifier

$A=C_S/C_f$

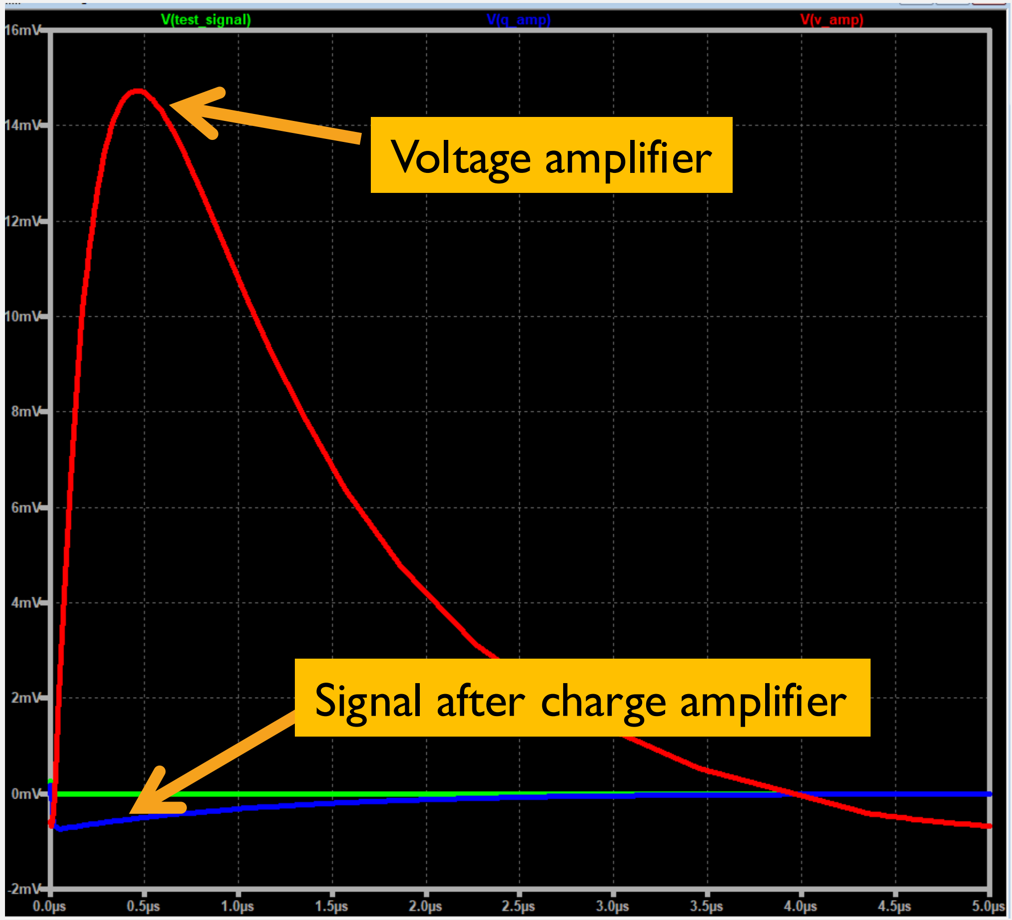
C_s: capacitance of signal

C_f: capacitance of feedback capacitor

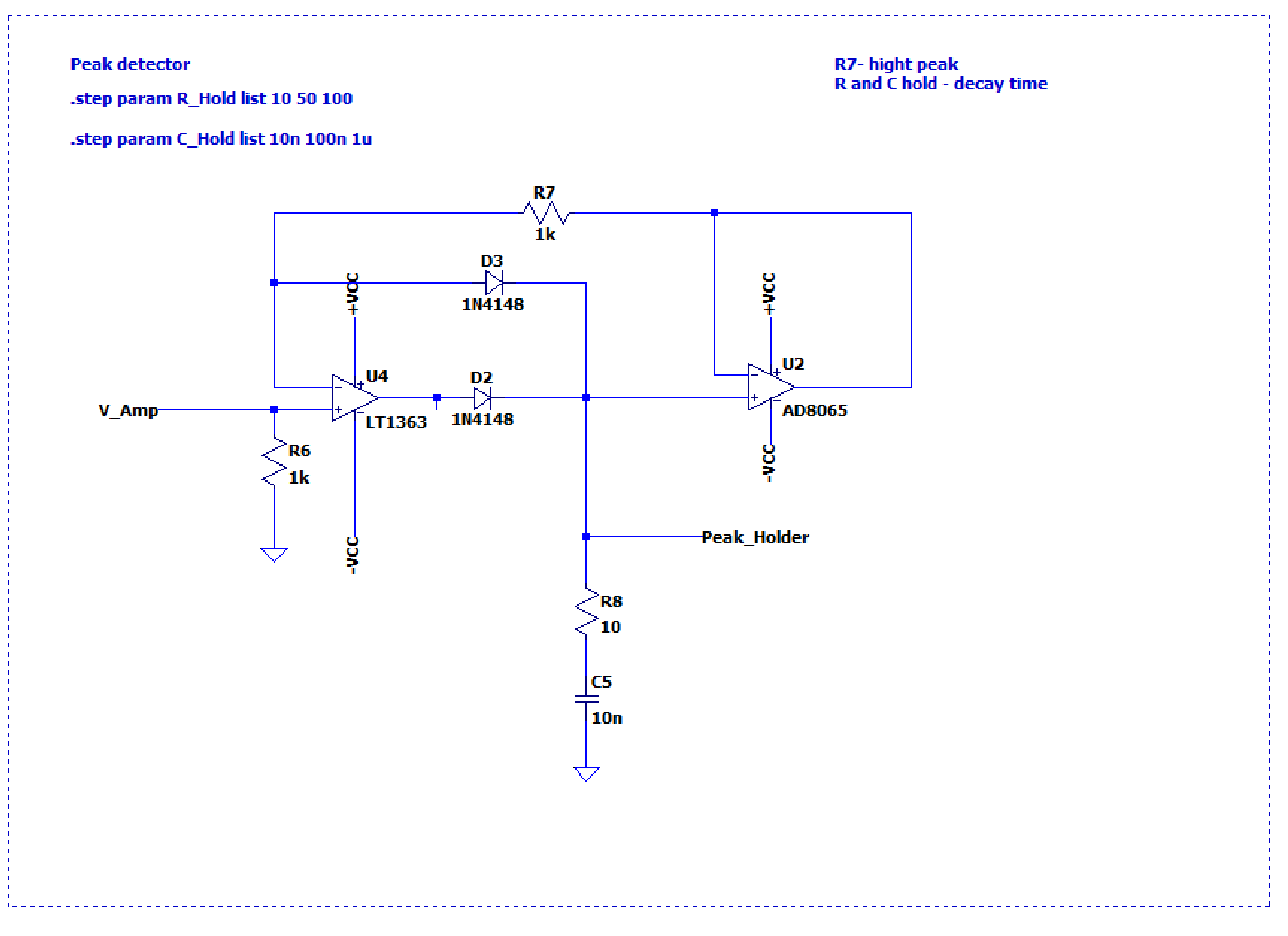
VOLTAGE AMPLIFIER SIMULATION



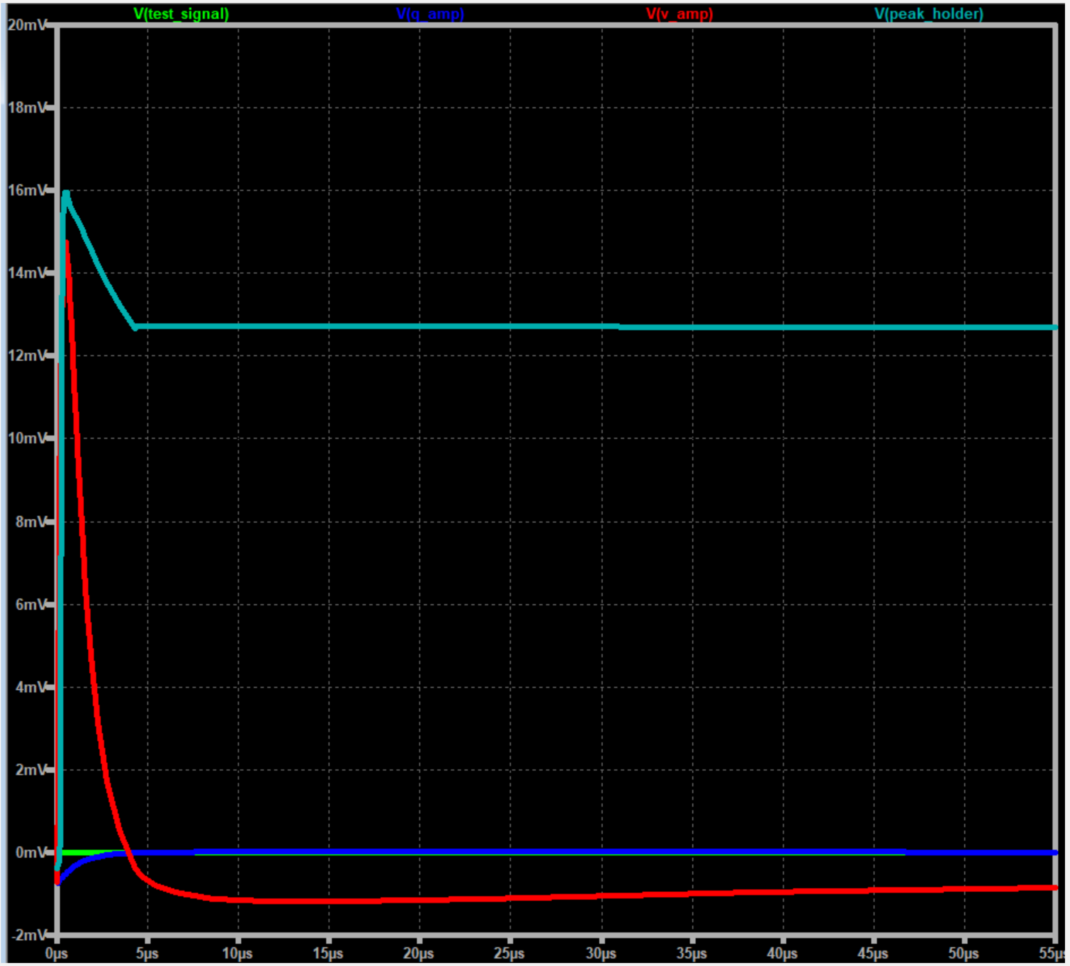
Simulation result



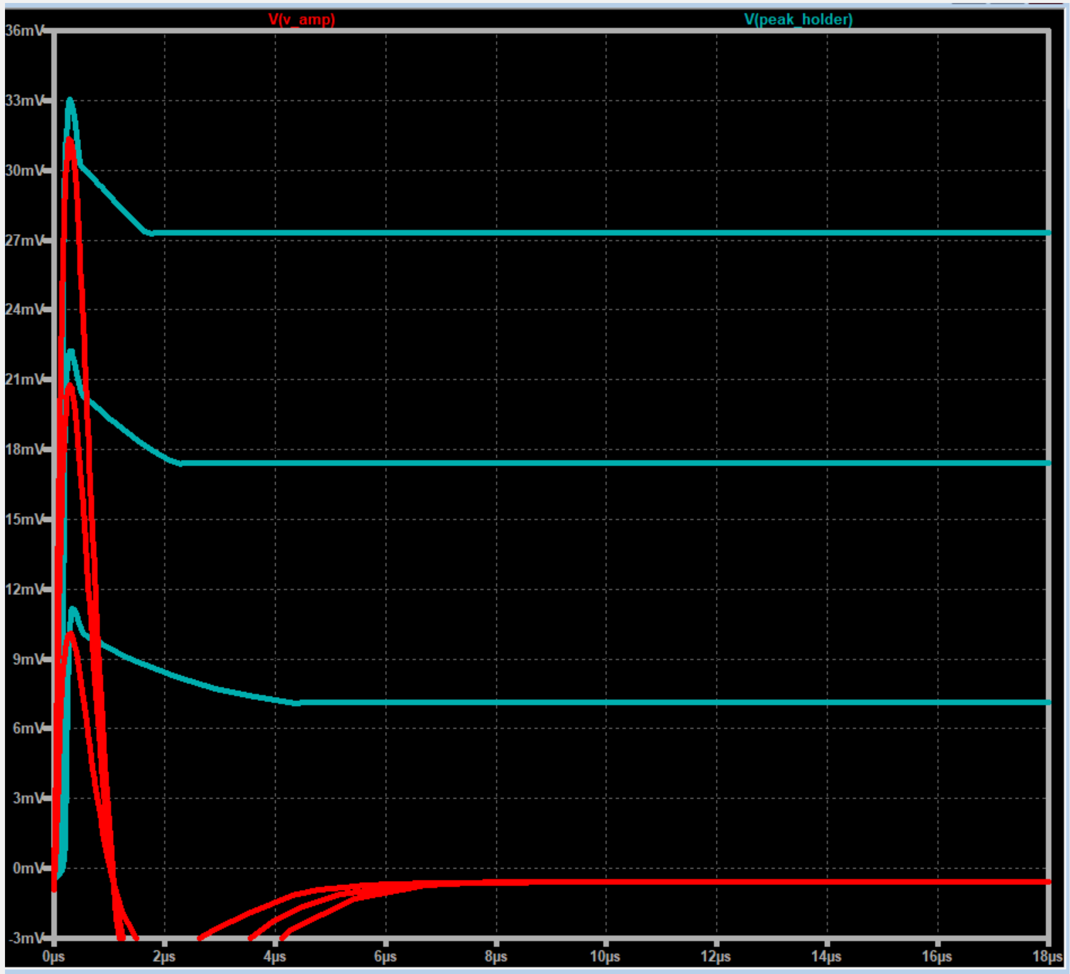
PEAK DETECTOR



PEAK HOLD SIMULATION



Holding peak



Linearity of holding peak

BACK UP