WORKSHOP ON LTSPICE

Fabrice WATEAU

What and why LTSPICE

- What ?
  - Analog part of pspice simulation software
  - Drawing interface and component library provide by LTC
- Why ?
  - Free download on

    www.linear.com/designtools/software/
  - Easy to use
  - But only analog analysis

Objective

- See the different ways to use this software
- Use LTSpice as an help during futur studys and projects

How do we work together ?

- 1st day :Presentation of the software and its possibilitys
- From 2nd to 8th day :
  - First step with Ltspice
  - .OP, .DC, .TRAN and .AC simulation
  - ,STEP, .PARM and .MEAS add tools

  For all simulations

  - First we perform simulation together
  - Second you try yourself on exercises
- The 9th day you try to use the new knowledges to resolve some small problems with Ltspice and my help,

OVERVIEW

Main simulation tools

- .OP : calculate the opearting point of a circuit
- .DC : the same as .OP but for several value of one/several voltage(s) or current source. The result is given in a graph
- .TRAN : use simulator as a scope to see the timing diagram of differents signals
- .AC : calculate the frequency response of circuit

OVERVIEW

Additionnal tools

- .STEP use with .PARAM command allows to perform analysis in which value of component can be varied .
- .MEAS allows to perform measurement on the result of the analysis (timing periode, frequency, max and min value of a curve, …)

Analysis

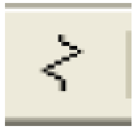
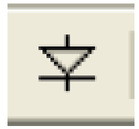
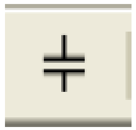
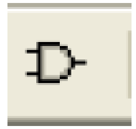
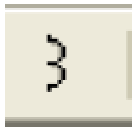
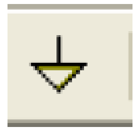
Different steps to perform a simulation

1. Create a new sheet and save it on a folder specially created for all the files of the simulation
2. Drawing schematic
   - Place and move component. Don't forget the ground
   - Place wire to connect components
   - Place label to mark the most important node
3. Set the analysis (.OP, .DC, .TRAN or .AC)
4. Run simulation
5. Interpret results

Drawing schematic

Place and move component

Select component by press a key or click on the correspondant pictures in the tool bar

Component	Press Key	Click	Component	Press Key	Click
Resistor	R		Diode	D	
Capacitor	C		Other component	F2 key	
Inductor	L		Ground	G	

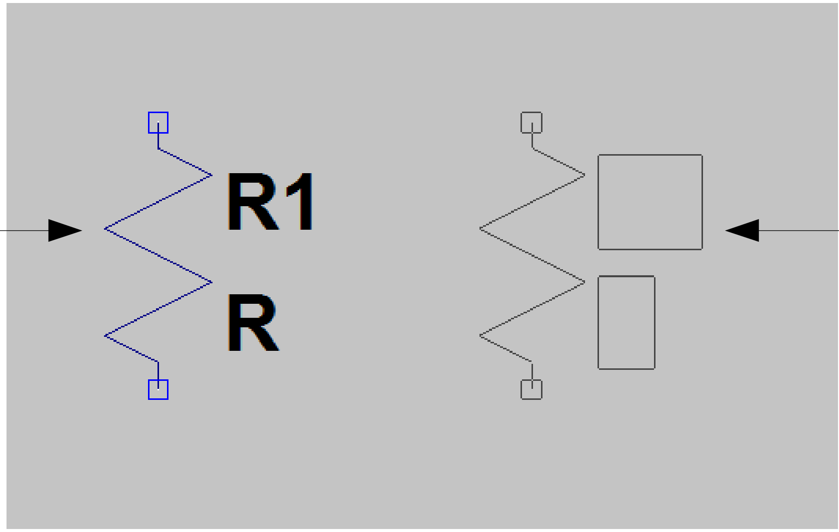
Move the mouse until the position you want, and left click to place the component.

Drawing schematic

Place and move component

To move and rotate a componant, it must be select (grey). It's not, select it whith  and use « CTRL+R » to turn it, and mouse to move it.

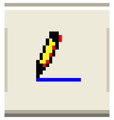
Fixed component, cannot move or rotate



Not fixed component, can move and rotate

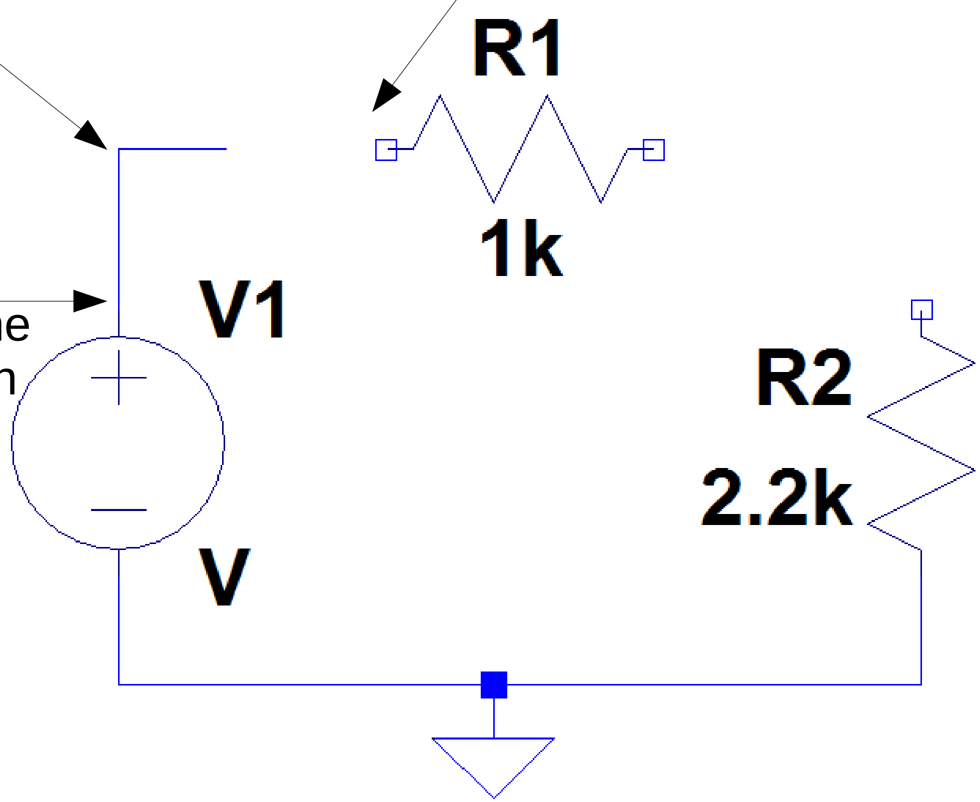
Drawing schematic

Connect component : trace wire

Press « F3 » key or click on  to select wire tools.

Left click to change direction

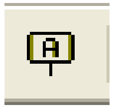
Left click on the terminal point of one component to begin

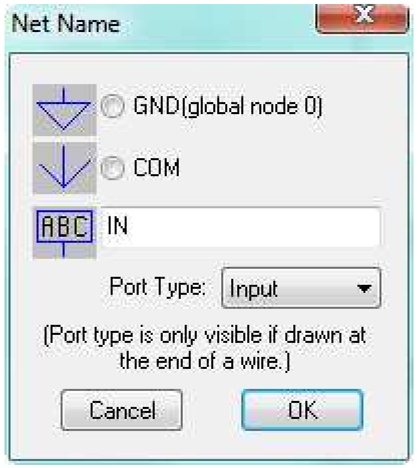


Left click on the terminal point of the other component what you want connect, or riht click to finish the wire

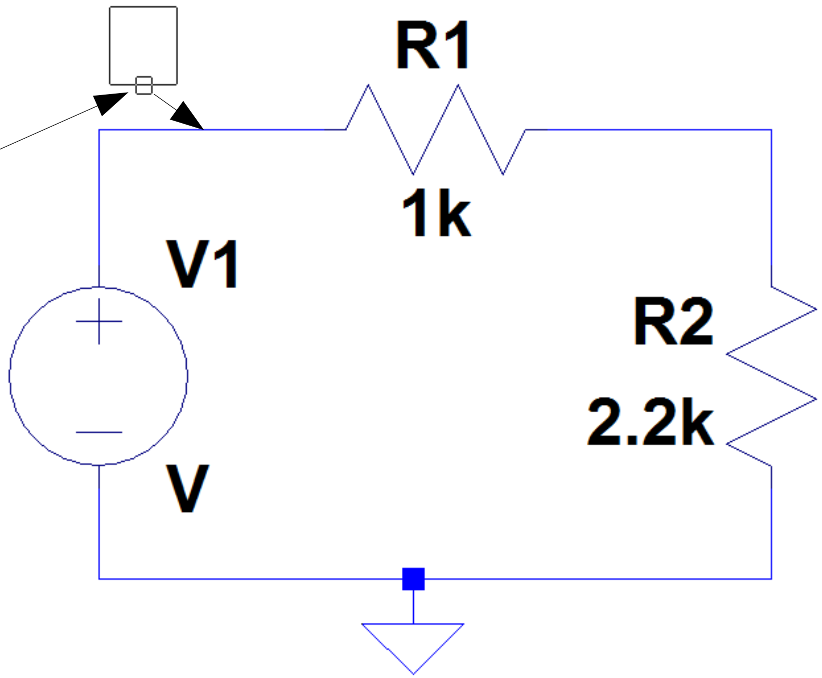
Drawing schematic

Place label

Press « F4 » key or click on  to select label tools.



Place the node on the wire on what you want give name.

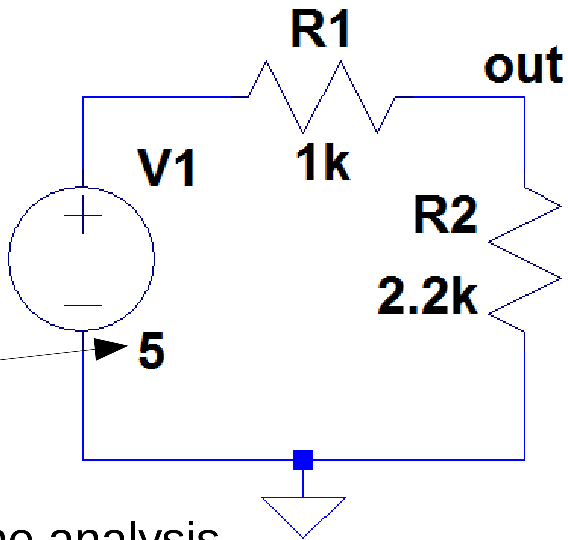


Comment : label can also be use to connect 2 nodes. Just give the same name at the 2 nodes

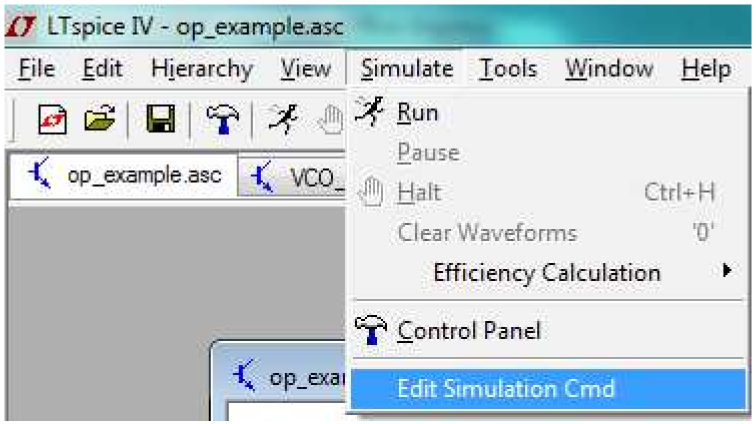
.OP analysis

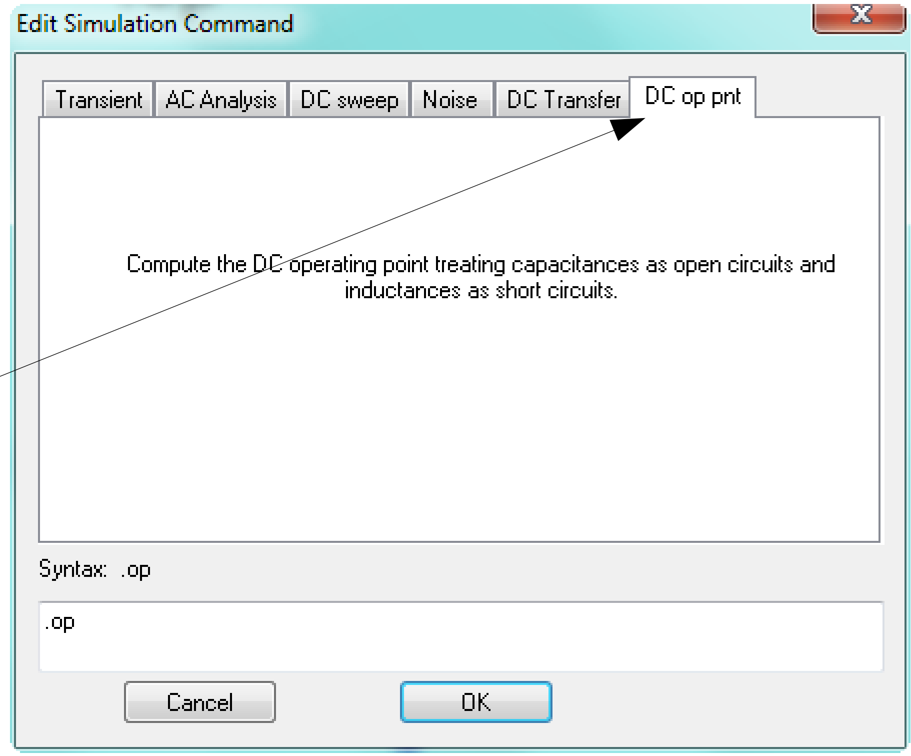
1. Draw the schematic study, and modifie (Right click on V) the DC voltage of V1

DC value



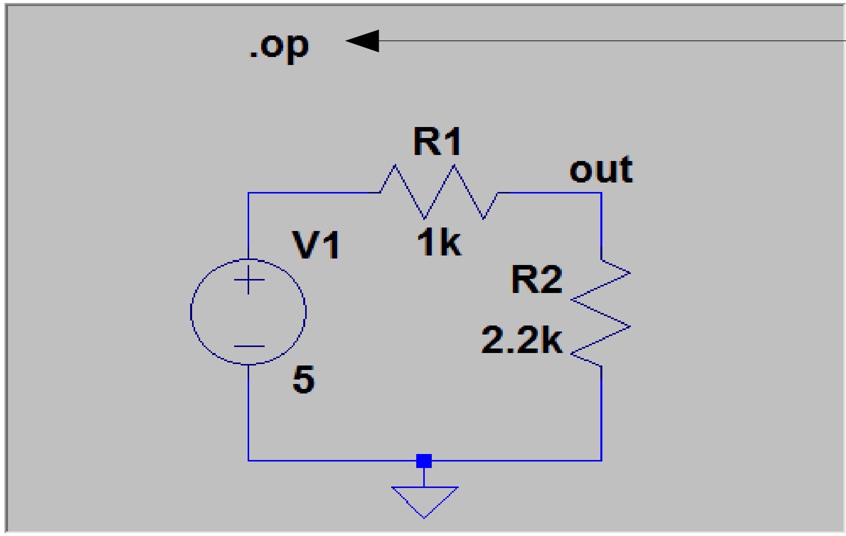
2. Setting the analysis





.OP analysis

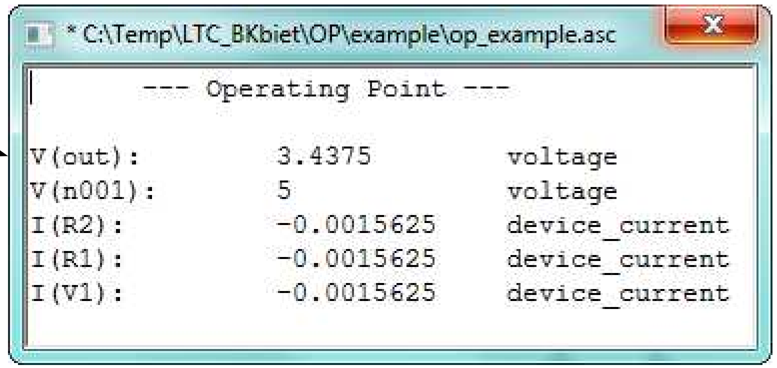
3. Click on schematic to valid your choice



left click on the schematic to put it

4. Run the analysis 

A new window appears and contains the value of the operating point


* C:\Temp\LTC_BKbiet\OP\example\op_example.asc

--- Operating Point ---

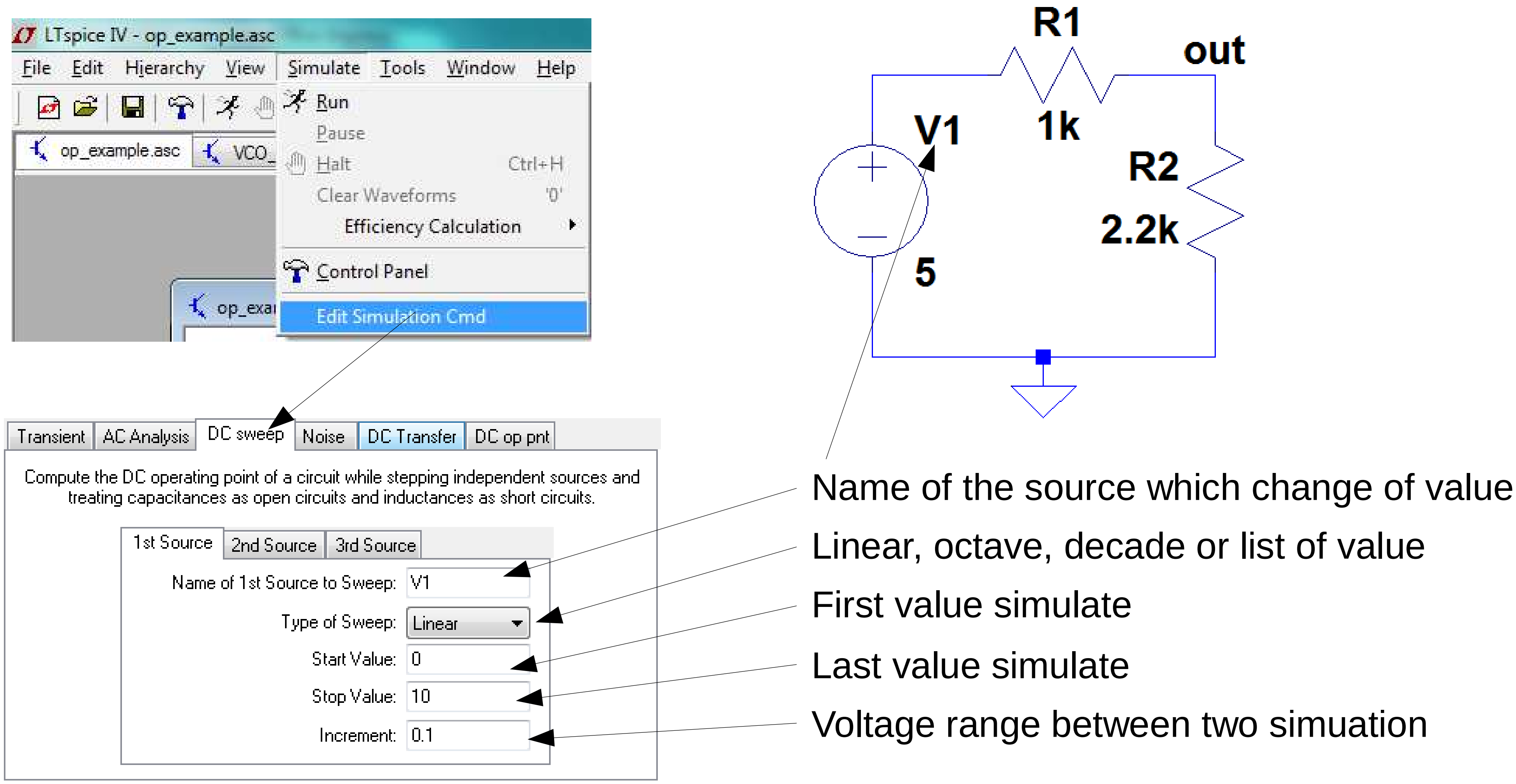
V(out):	3.4375	voltage
V(n001):	5	voltage
I(R2):	-0.0015625	device_current
I(R1):	-0.0015625	device_current
I(V1):	-0.0015625	device_current

.DC analysis

- Perform .op analysis for any value of one (or more) voltage (or current) source(s).
- Example: with the same circuit,we want to know how change the output voltage, if the input change.
- You need only change the analysis setting by choosing DC sweep

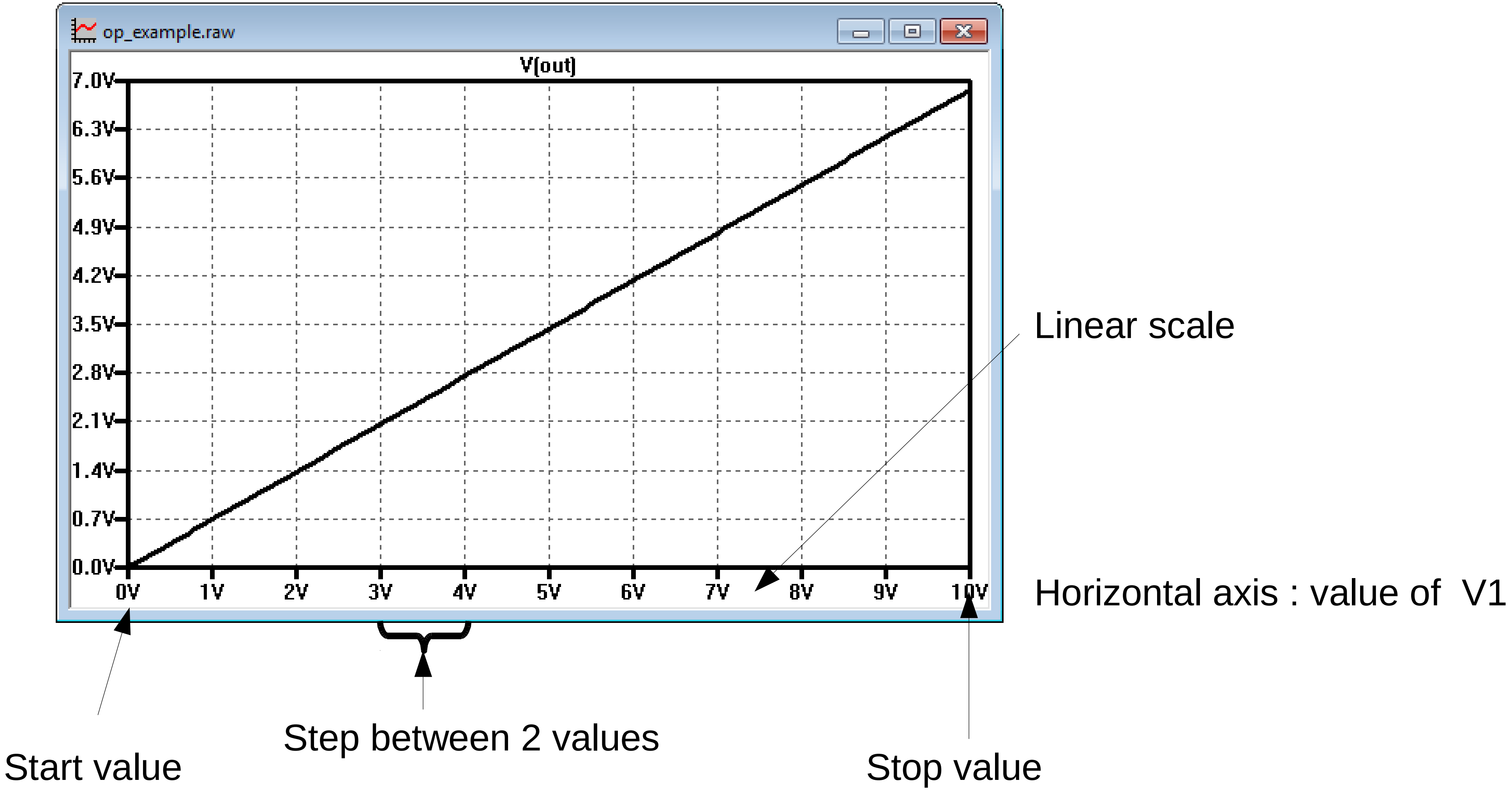
.DC analysis
Setting

- Setting the DC sweep analysis



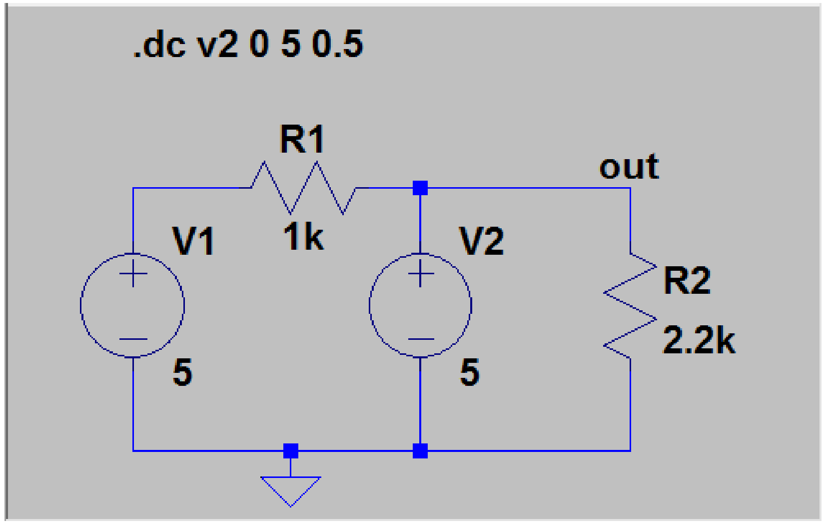
.DC analysis
Result

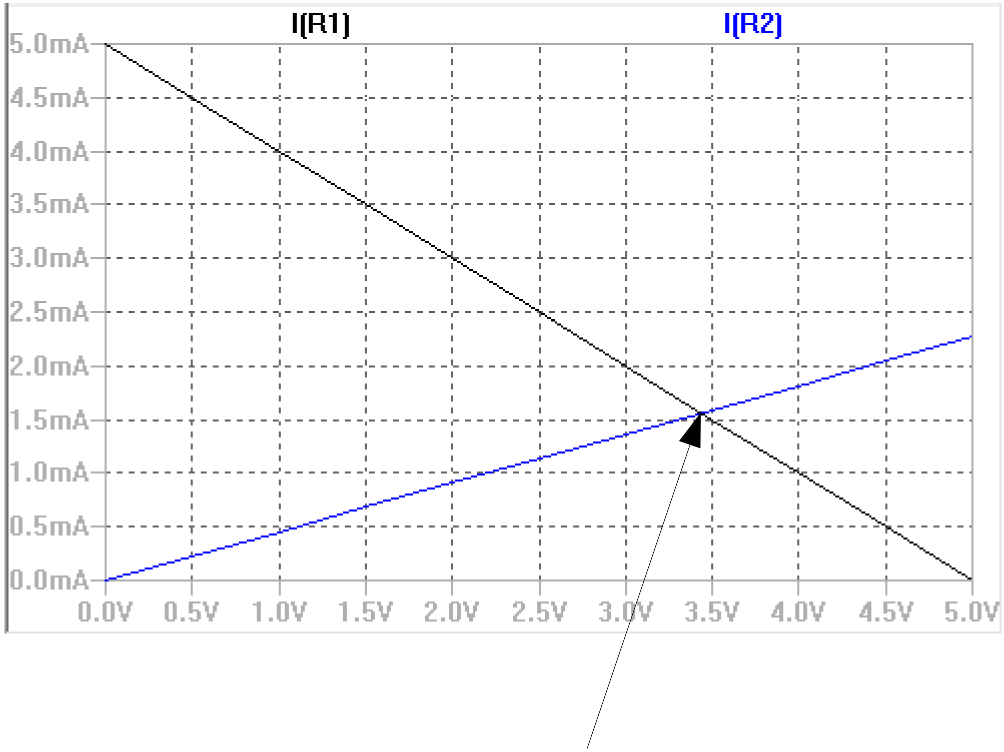
- Result is a graph :



.DC analysis
Example

- Load line

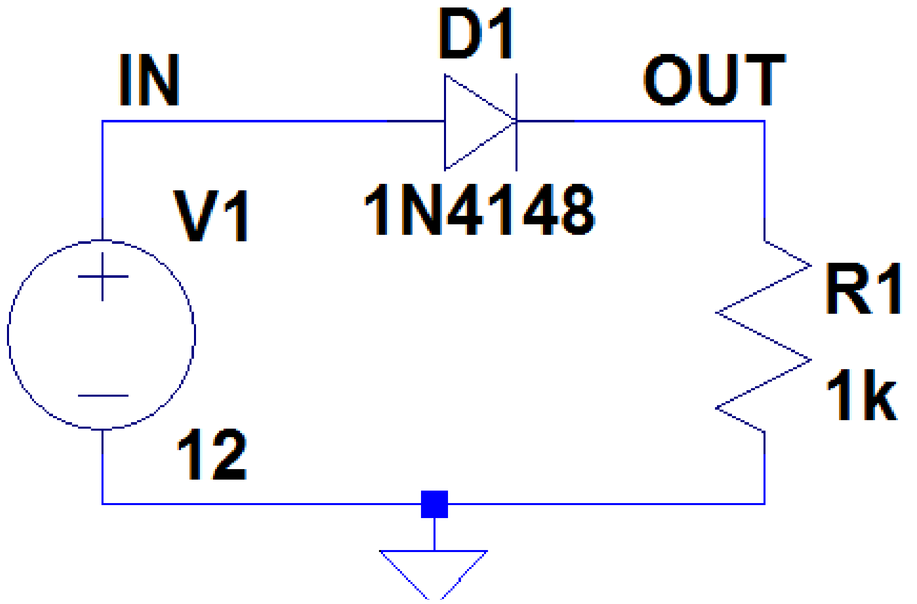




Operating point

.OP and .DC
Trainning

1.Propose a simulation which give the operating point of the following circuit



2.Propose a new simulation which allows to have the same result by tracing diode and « generator+resistor » caracteristics

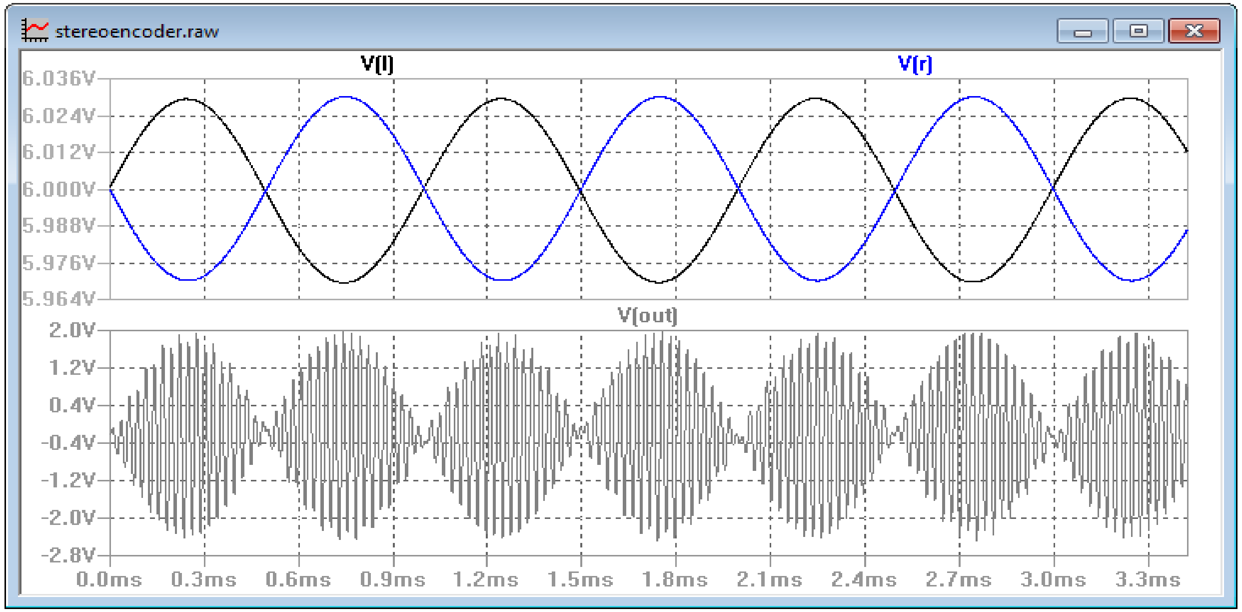
.DC sweep

Trainning

- Propose analysis which allow you to plot the 3 caractericts of BJT (Bipolar Junction Transistor),

  1. Ib = f(Vbe) for Vce = 6V
  2. Ic = f(Ib) for Vce = 6V
  3. Ic=f(Vce) for 0<Vce<12V pour ib= 0 to 100uA by step 10uA

.TRAN analysis



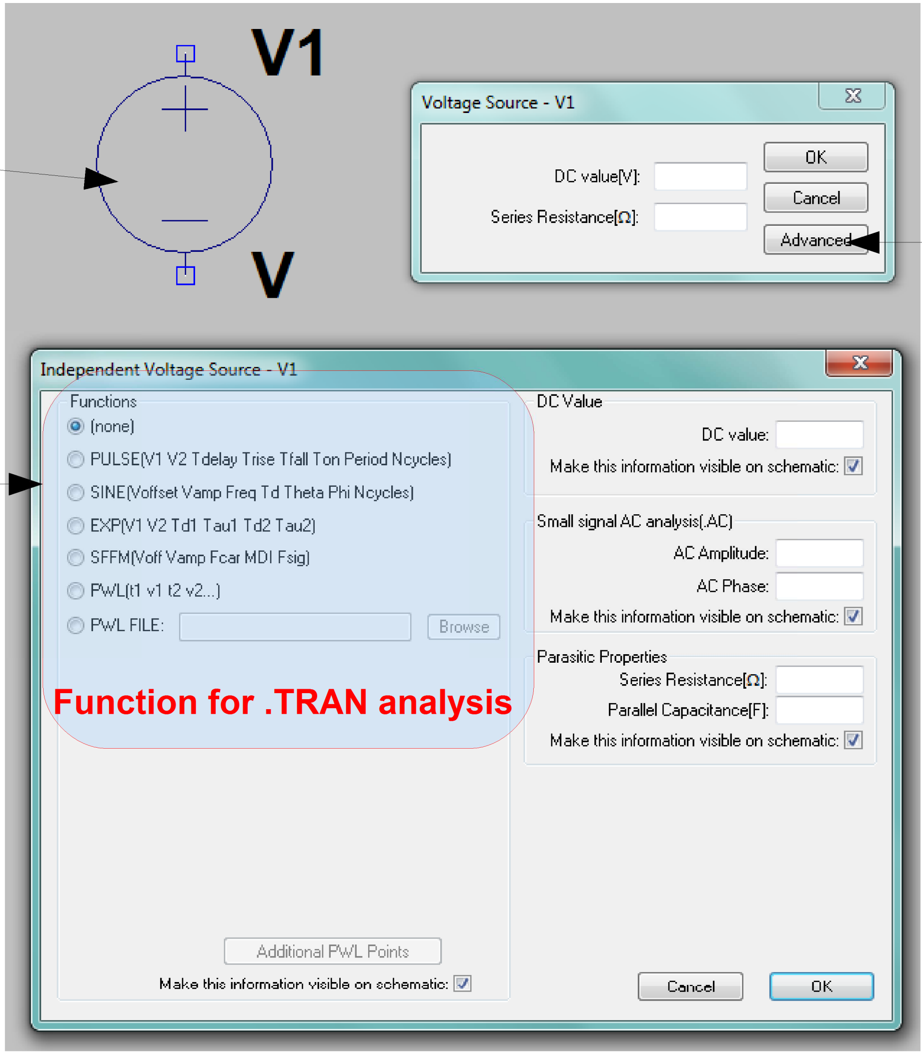
- Transient analysis is used when you want to see the timing diagram of the waveform (voltage and/or current) of a circuit.
- In practical, it's the same result as an scope

.TRAN analysis

Set up the voltage or current source

1

Right on the source symbol



2

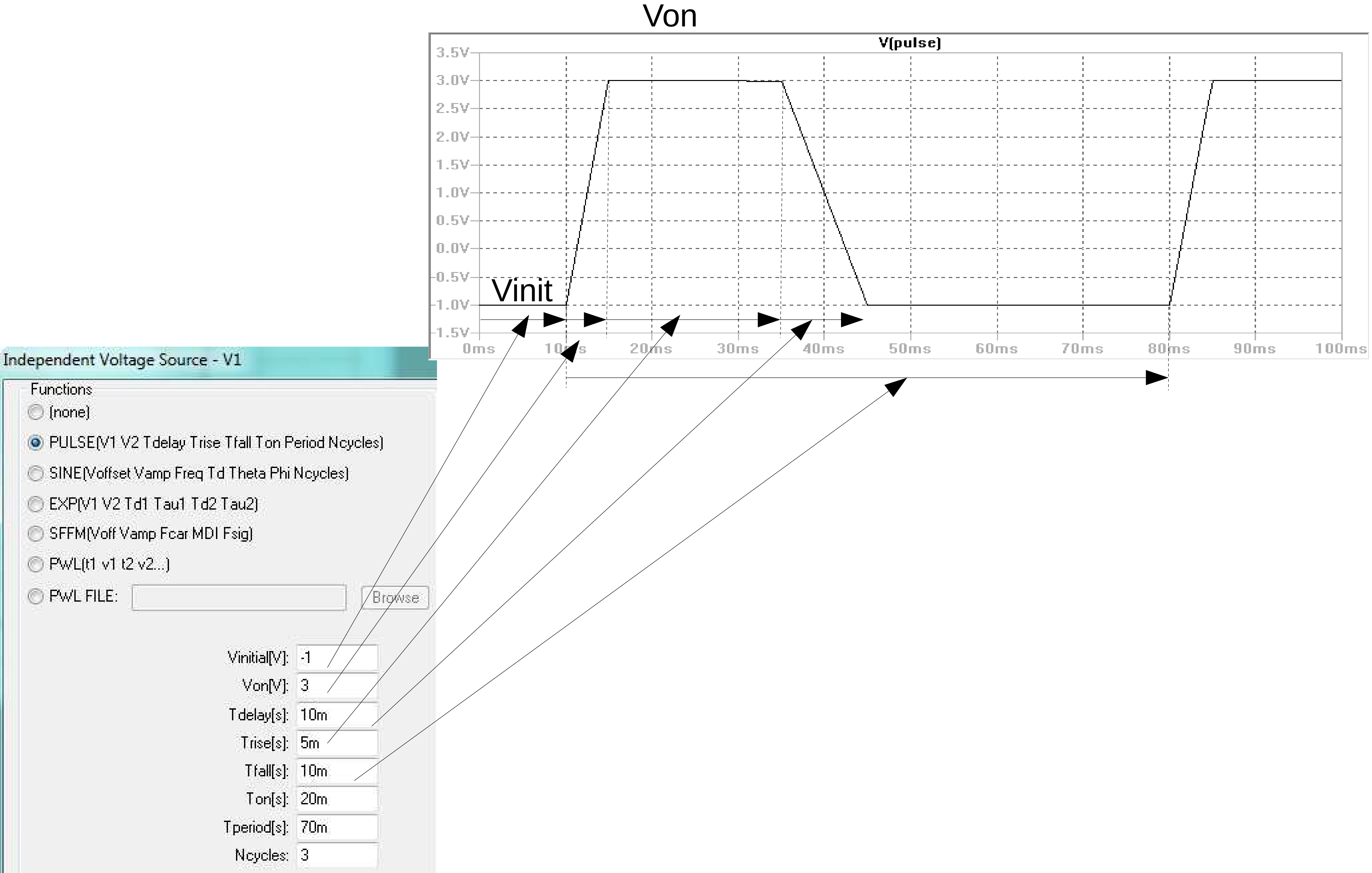
Click on Advanced button

3

Choose one of this function. Each are discribe below.

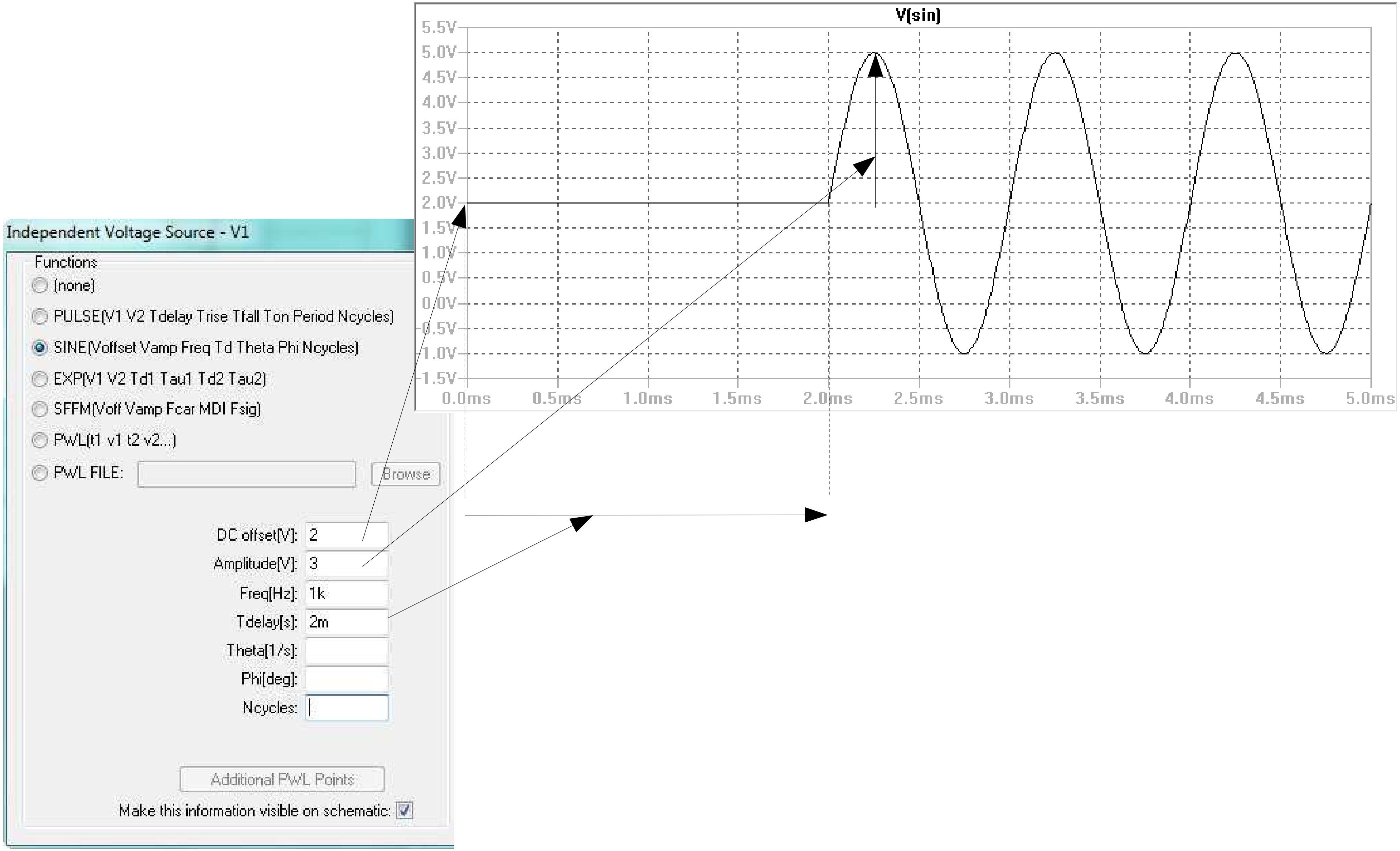
.TRAN analysis

PULSE DEFINITION



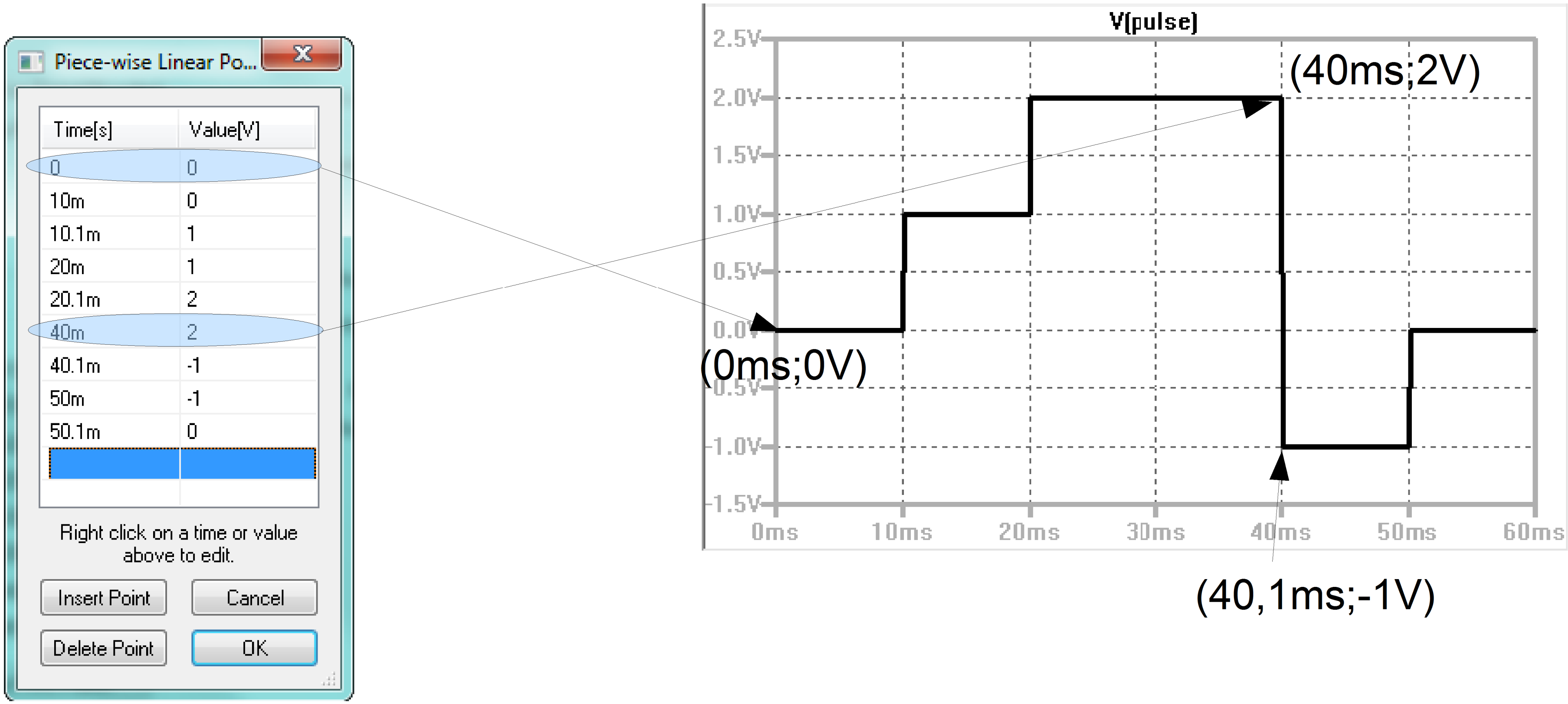
.TRAN analysis

SINUS DEFINTION



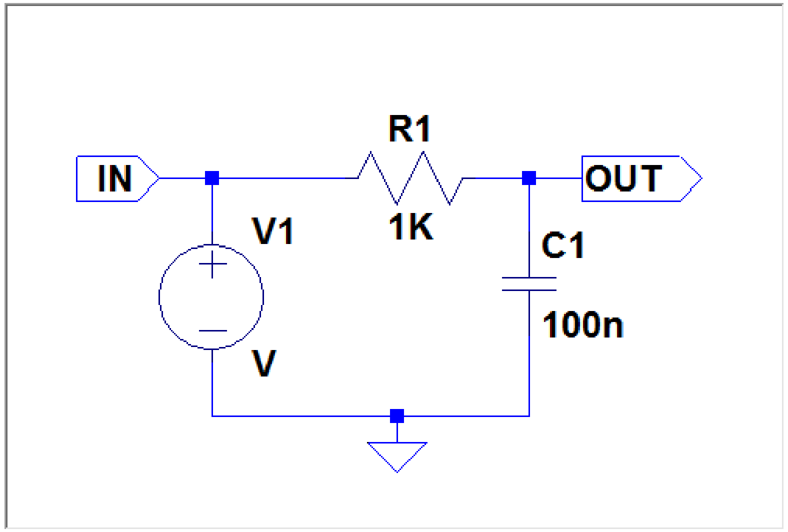
.TRAN analysis

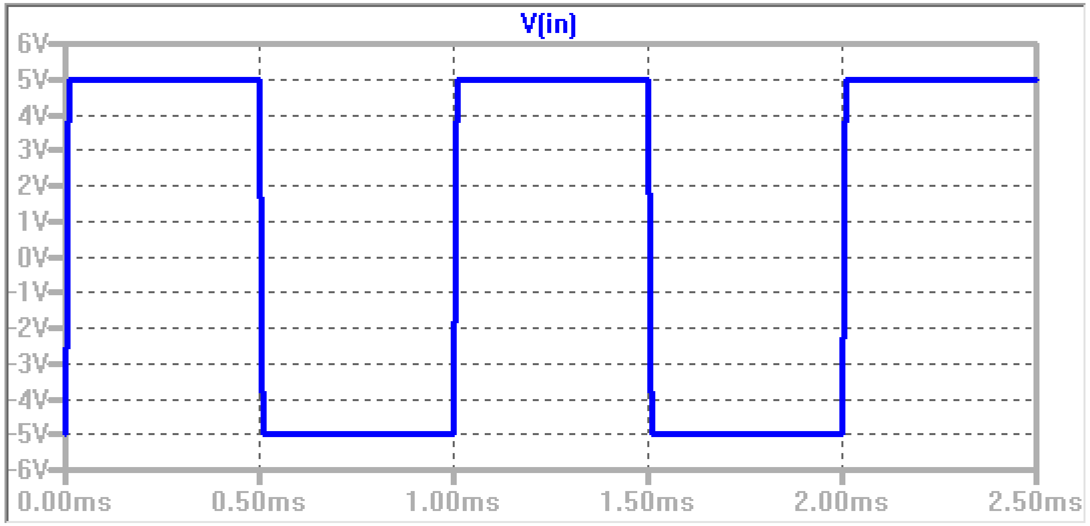
PWL (Piece Wire Linear) DEFINITION



.TRAN analysis
Example : RC low pass filter

- Draw the circuit below :



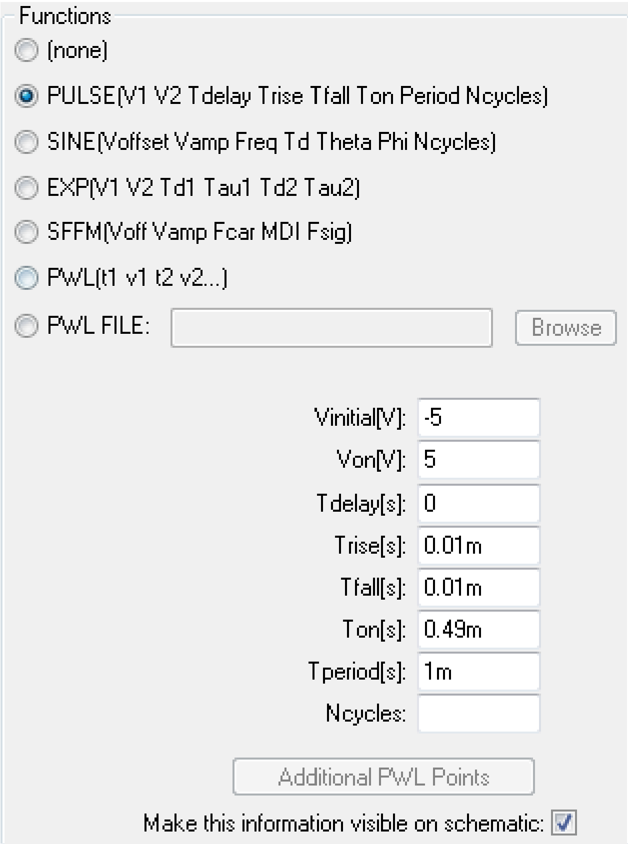


- Set the voltage source as it provides a square wave as defined above (at right)

- Set transcient analysis to observe 10 periodes of V(IN) and V(out).

.TRAN analysis
Example : RC low pass filter

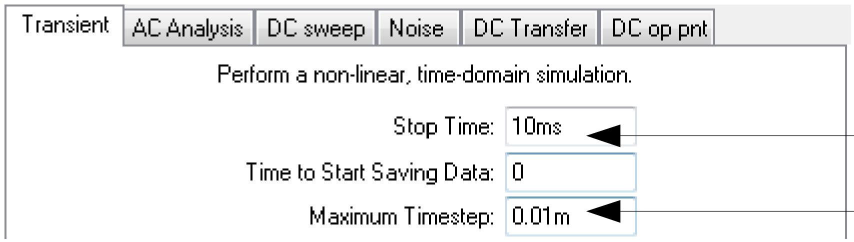
- After drawing schematic, right click on the voltage source and set as below :



.TRAN analysis

Example : RC low pass filter

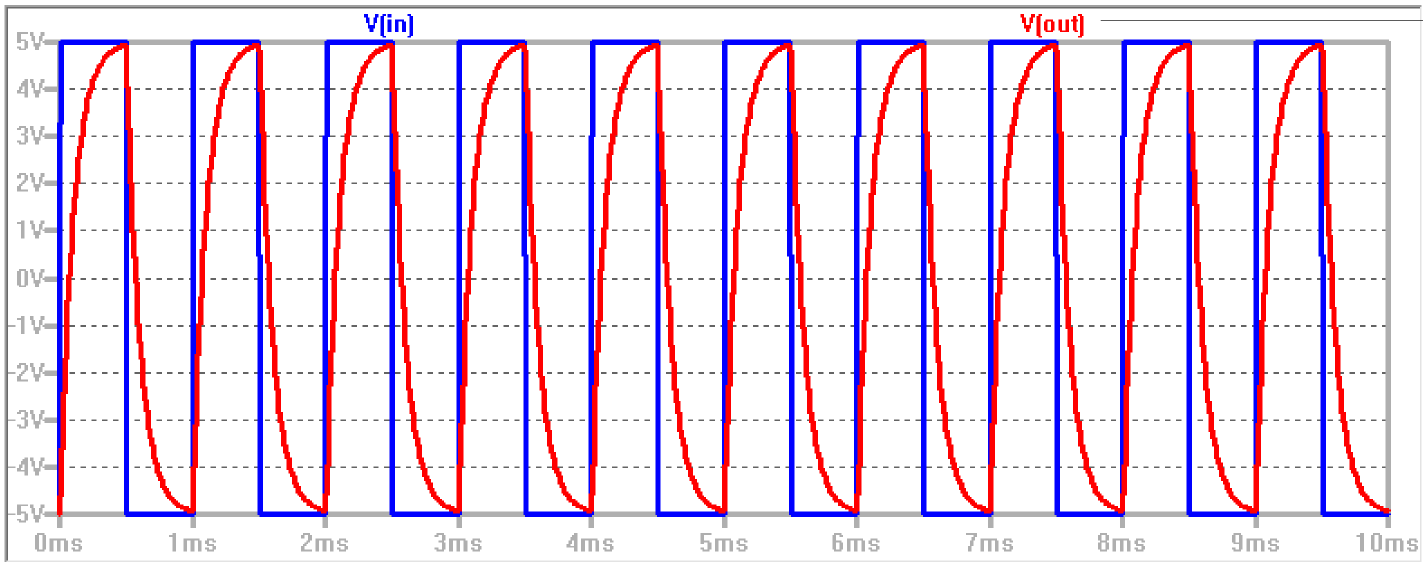
- After setting the source, set the analysis as below :



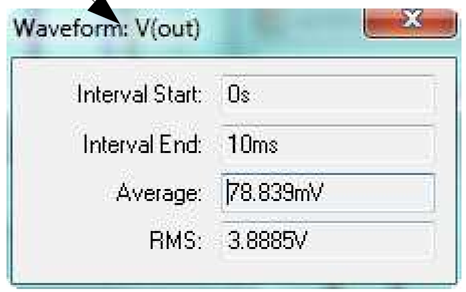
10periode => 10*1ms

As the rise and fall time in pulse definition

- Run analysis and display v(IN) and V(out)

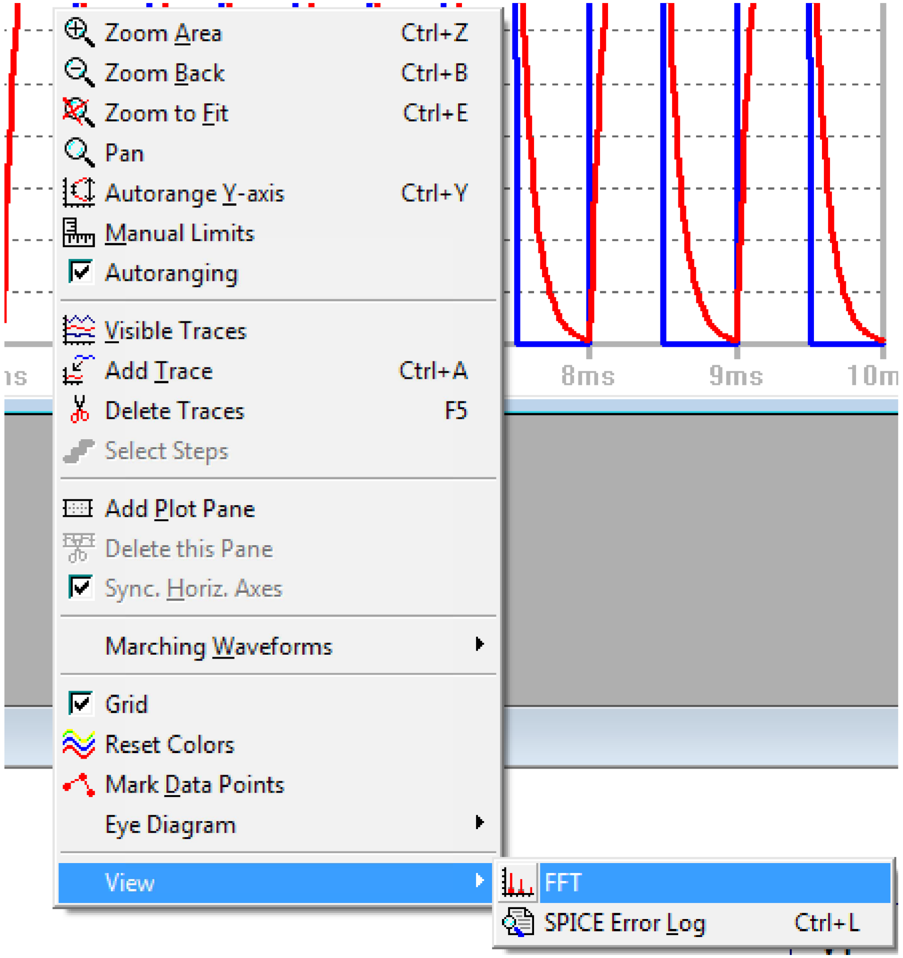


CRTL+click onthe name



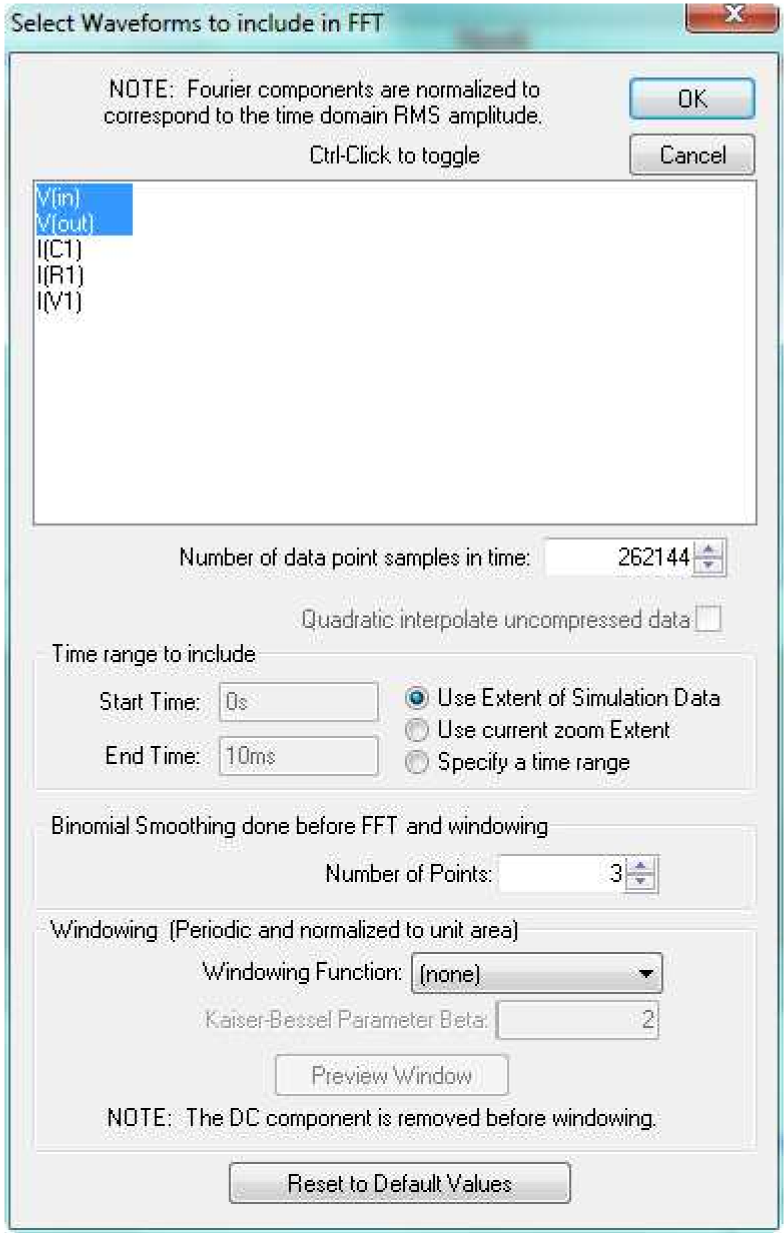
.TRAN analysis
FFT

- You can see the spectrum of signal by right clik on the probe windows and in the list choose « view », the « FFT » :



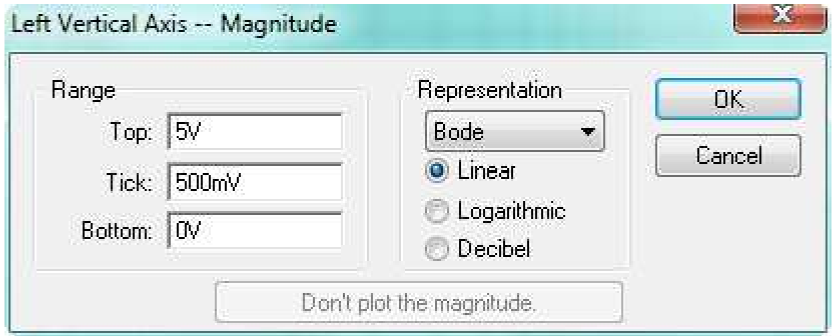
.TRAN analysis
FFT

- Select V(IN) and V(OUT) and click « OK » .
- In the new window select the both V(IN) and V(OUT) again and click Ok.

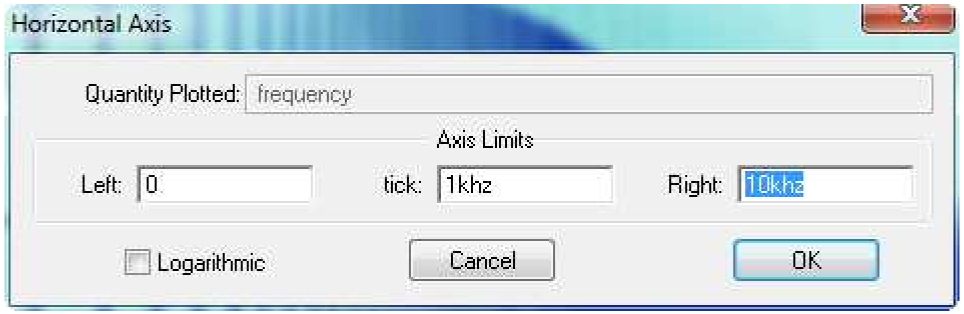


.TRAN analysis
FFT

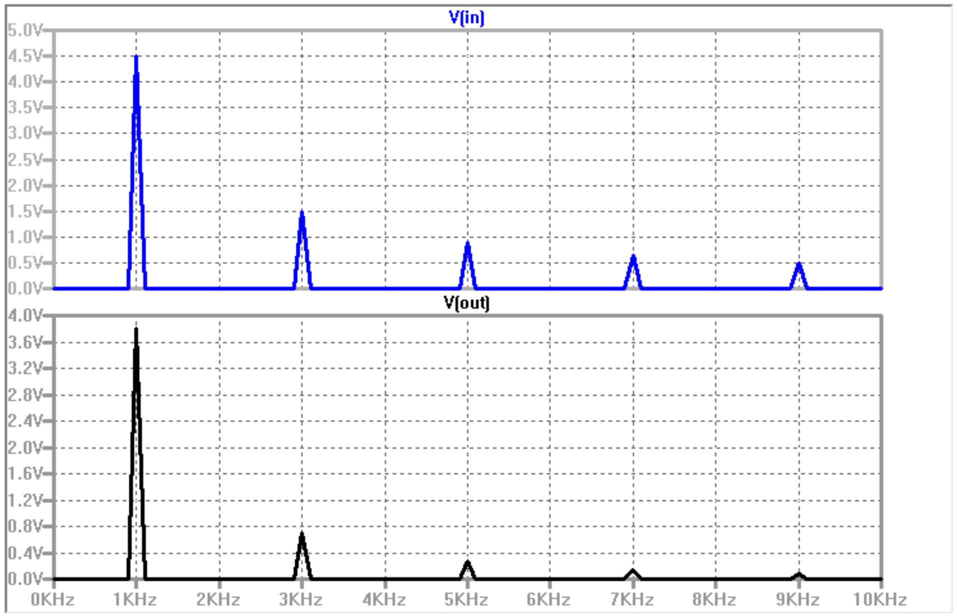
Modifie Y-Axis.



Modifie X-Axis.



- You must have :

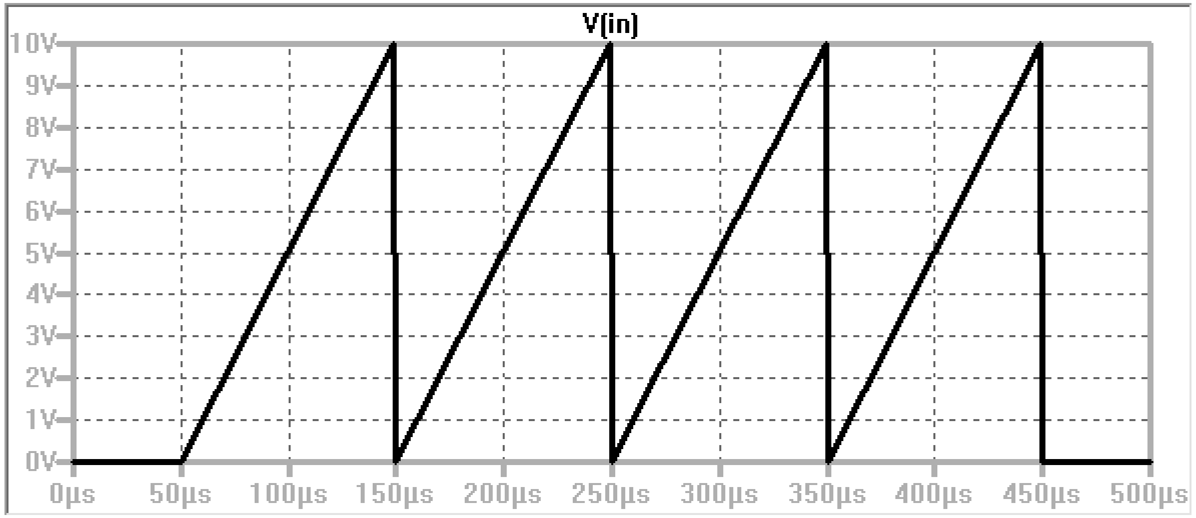


.TRAN analysis

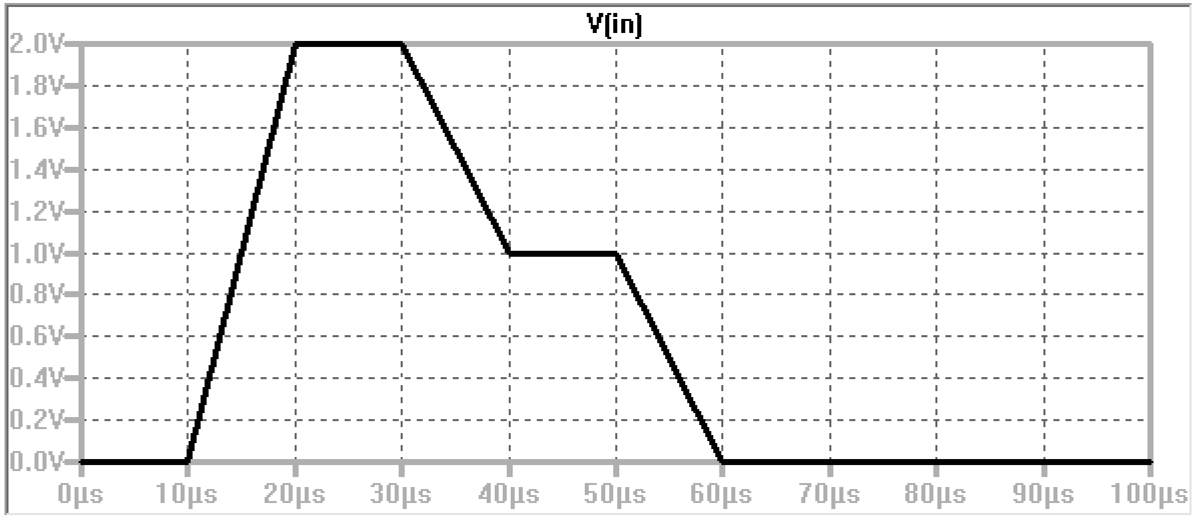
Trainning : setting source

Place voltage source on a new schematic project, and set to the waveform as follow :

1



2

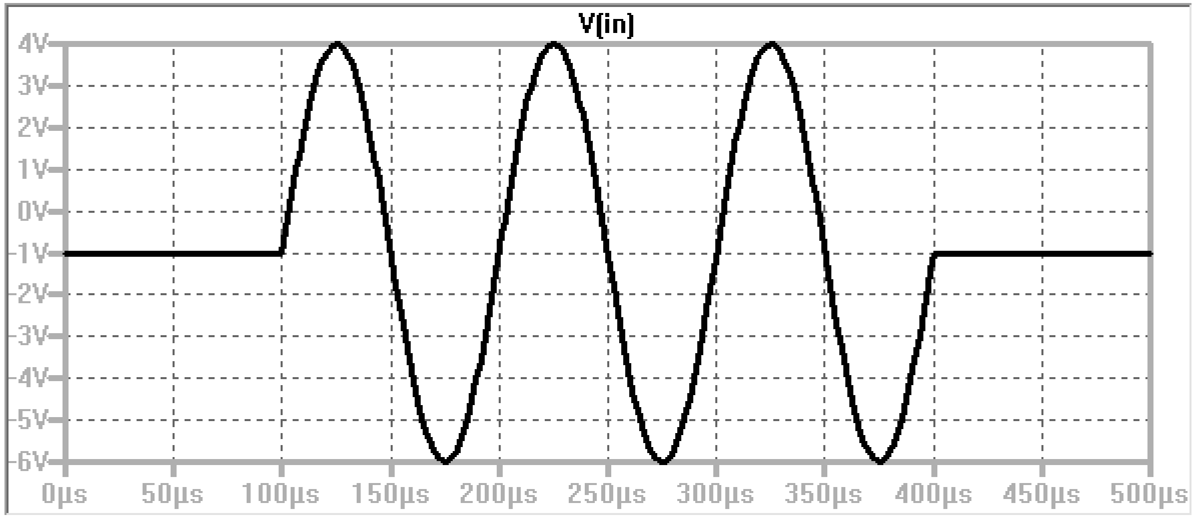


.TRAN analysis

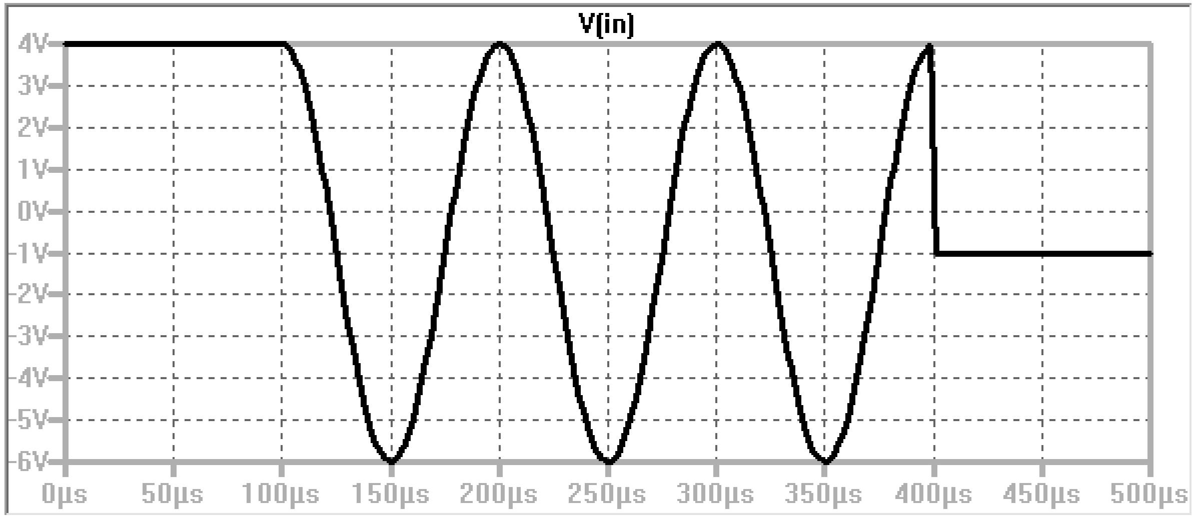
Trainning : setting source

Place voltage source on a new schematic project, and set to the waveform as follow :

3

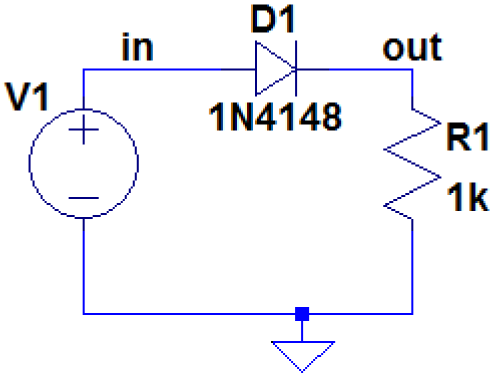


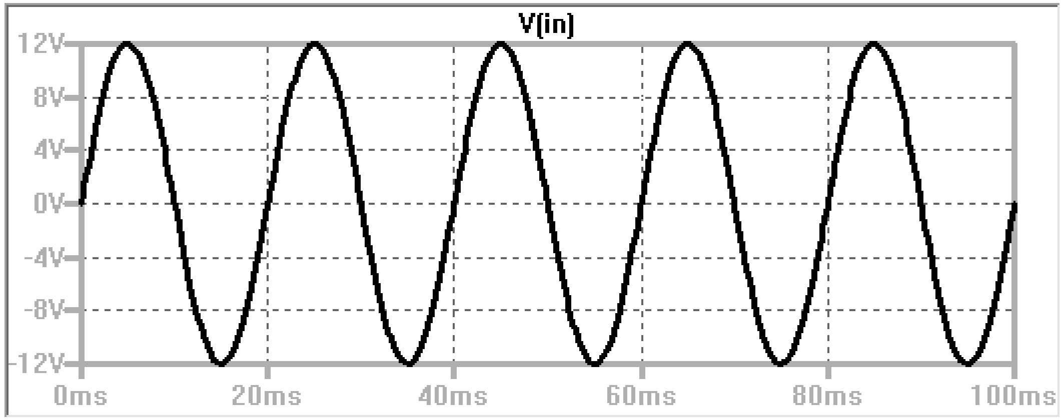
4



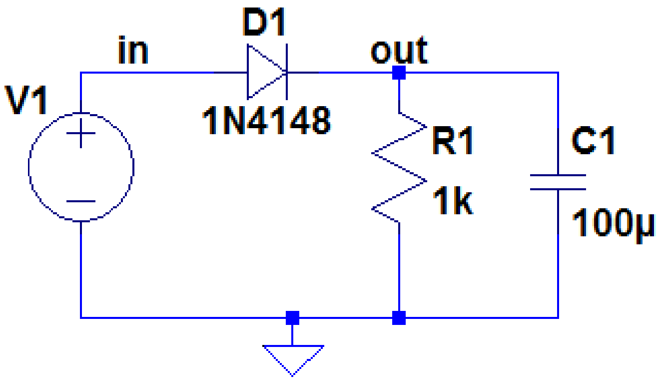
.TRAN analysis
Trainning : rectifier

- Draw the schematic as below and perform a transient analysis for an sinus wave input as below, display V(in) and V(out), and I(D1) in an other pane:

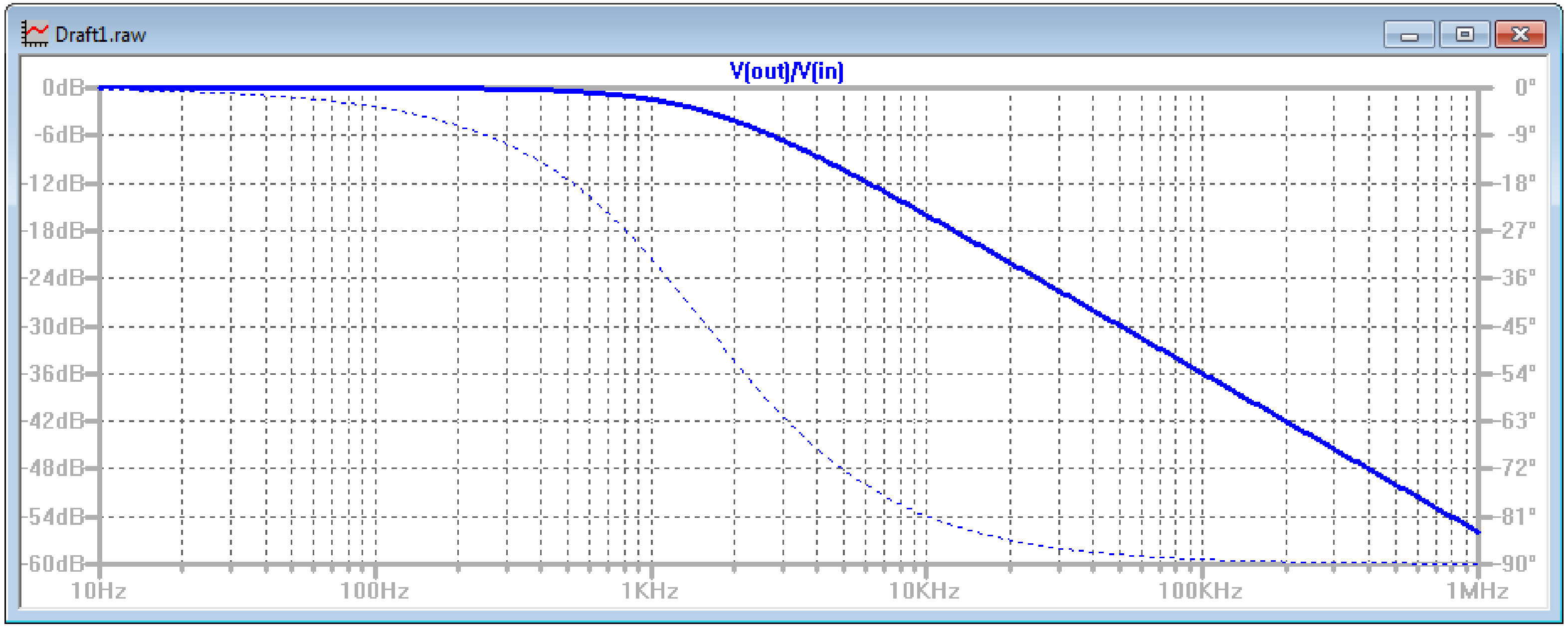




- Modify the schematic as below and run again



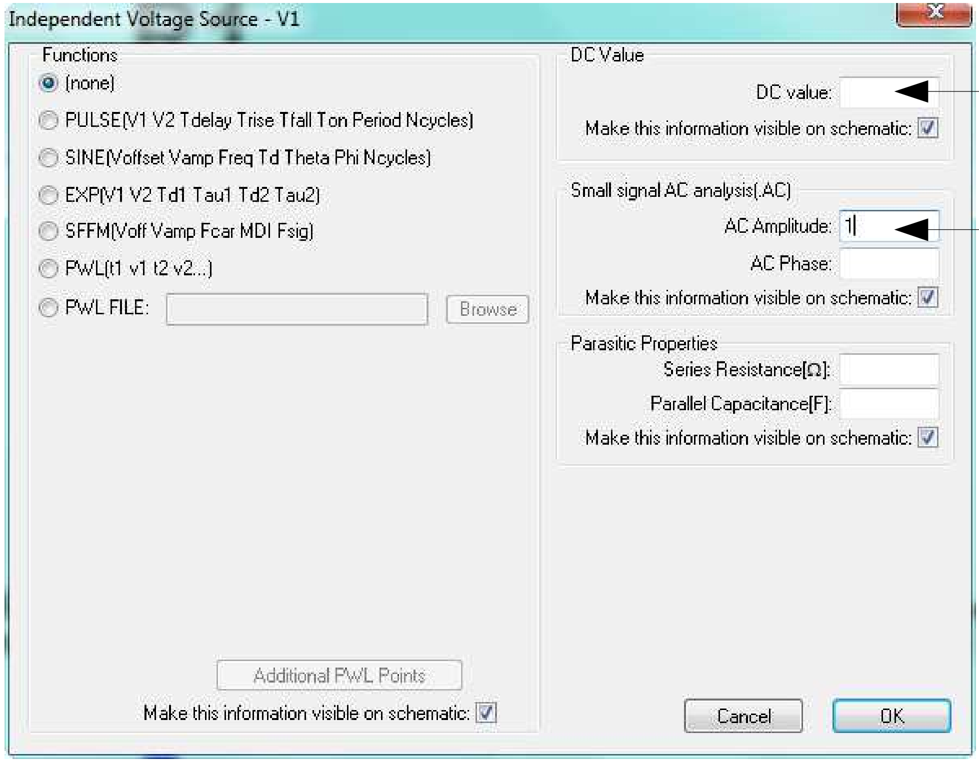
.AC analysis



- Perform an Small Signal AC Analysis Linearized arround the DC Operating Point and display the diagram of magnitude and phase functions of frequency, impedance versus frequency,...
- This mode of analysis is useful for filters, networks, stability analyses, and noise considerations.

.AC analysis
Example : RC low pass filter

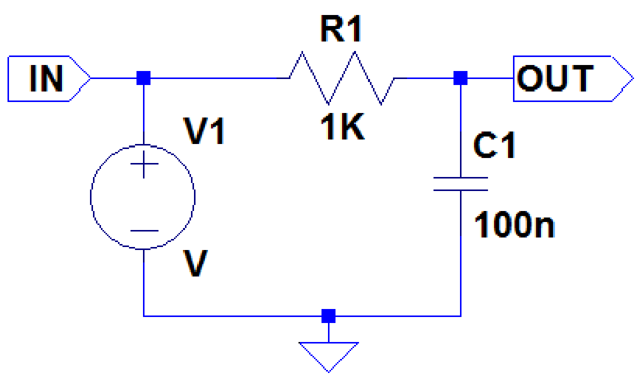
- After drawing schematic, right click on the voltage source and set as below :



Use by the software to calculate the operating point before linearization (0 if empty)

Amplitude of the sine wave used to drive the circuit

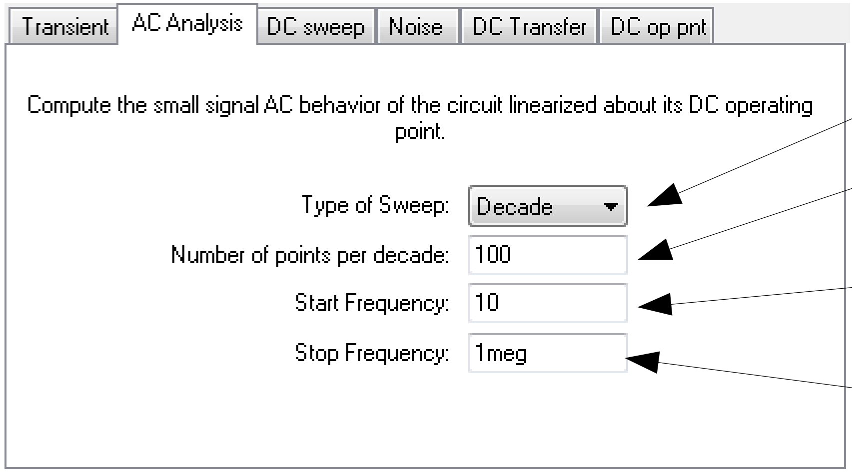
Circuit study



.AC analysis

Example : RC low pass filter

- After set voltage source, set analysis as below :



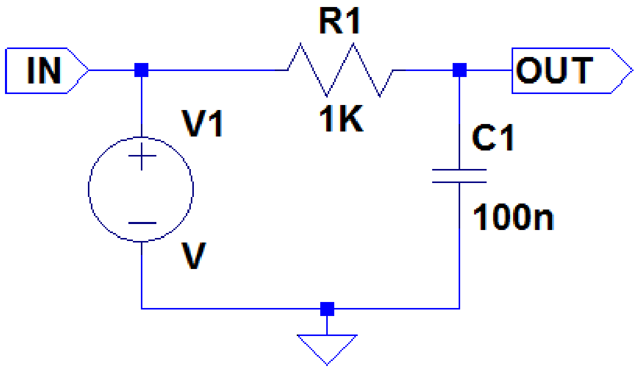
Decade, octave or linear

Take suffisant point if you want smooth curves

At least 1 decade lower than the theoretical low cut off frequency

At least 1 decade higher than the theoretical low cut off frequency

Reminder :



$$\frac{V_{out}}{V_{¿}} = \frac{1}{1 + r_1 C_1 s}$$

$$f_{cutoff} = \frac{1}{2 * \pi * R_1 * C_1}$$

- Run analysis

.AC analysis

Example : RC low pass filter

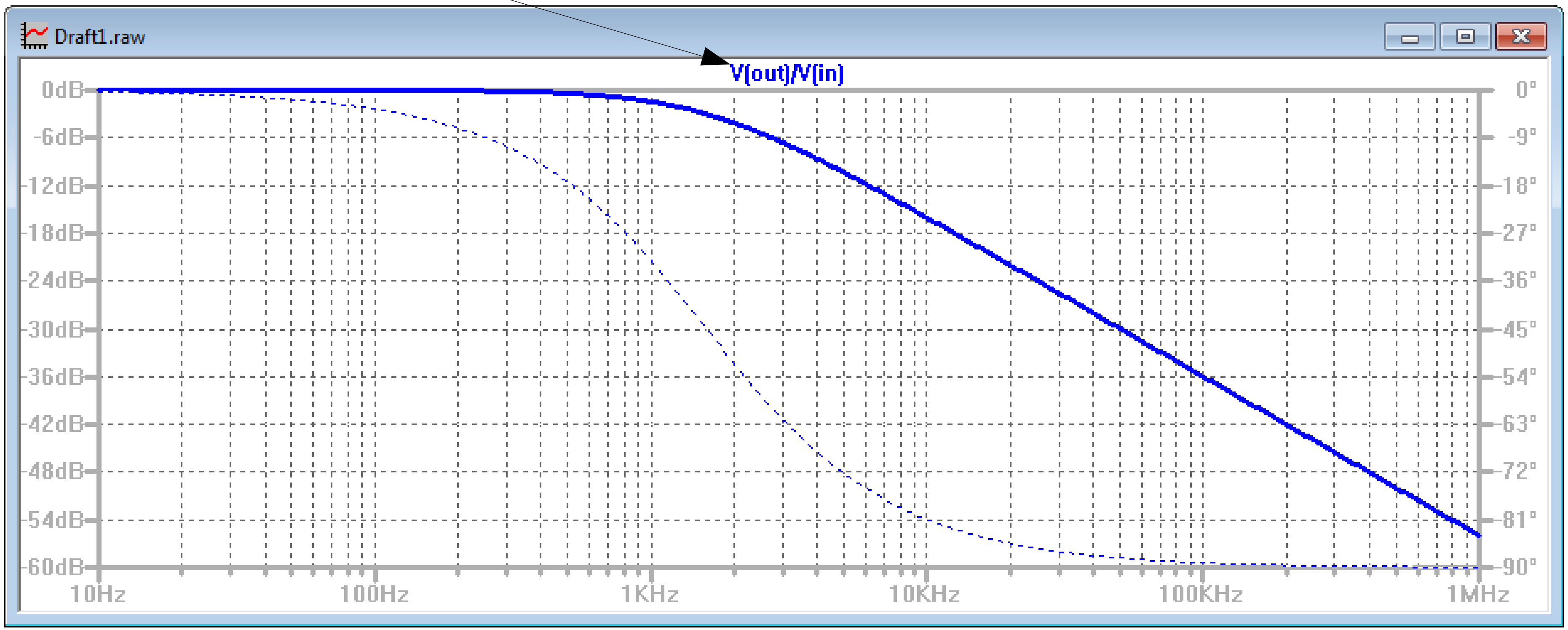
- result :

Change color

To use cursor on the curve

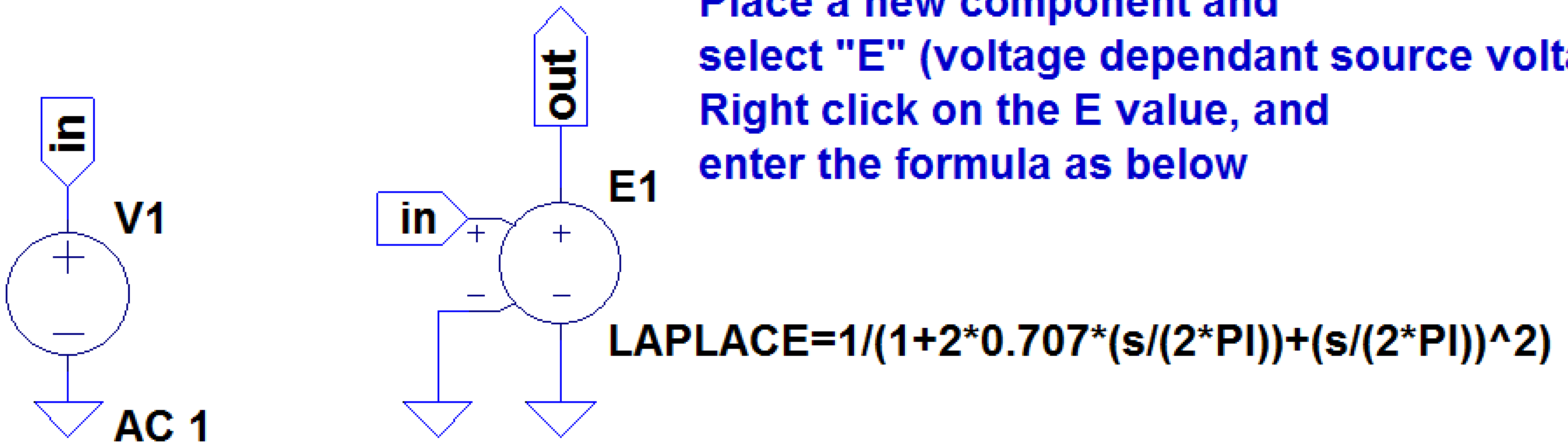
Left click on the name to modify

Enter the expression you want to display



.AC analysis
Trainning : laplace function

- Create a new schematic and draw as follow

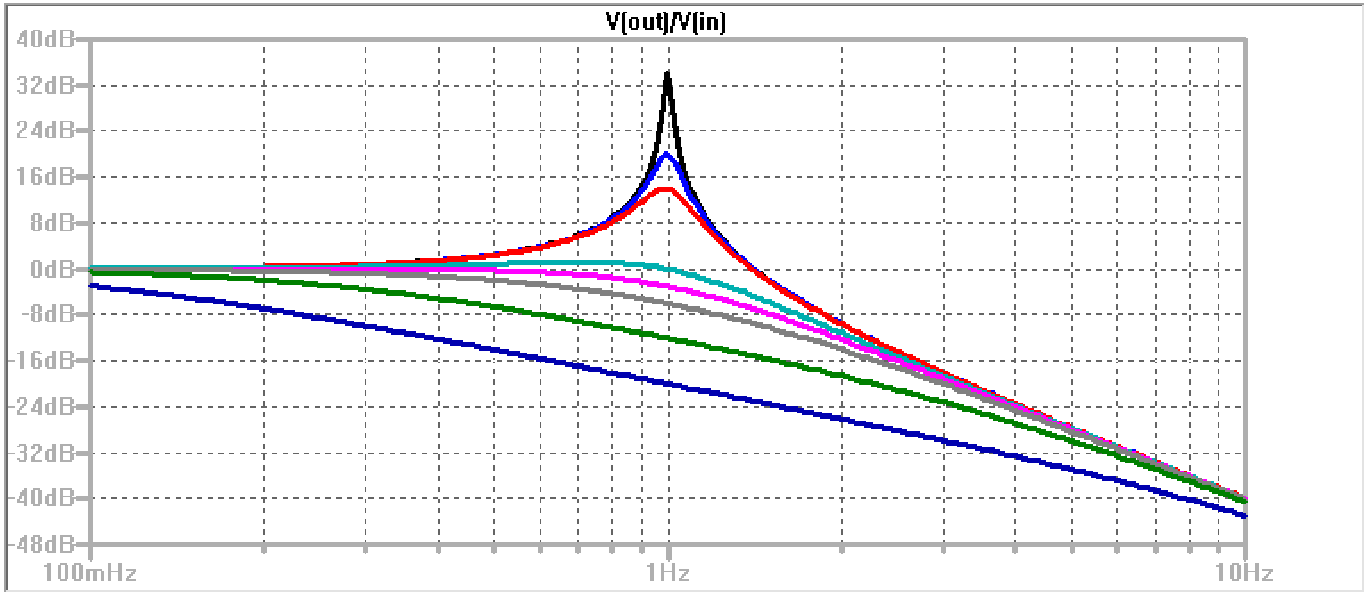


Place a new component and
select "E" (voltage dependant source voltage),
Right click on the E value, and
enter the formula as below

- Set an AC analysis in decade between 0.1 and 10 hz, and run simulation.(you can change the value 0,707 of the damping factor to see its effect)

Additionnal tools

.PARAM .STEP



- Directive .PARAM allow you to create one or any user defined variable that you can use in place of componant value, or for calculate an other parameter.
- Directive .STEP allow you to change the value of parmeter step by step, then allow to repeat analysis for various values of parameters or component.

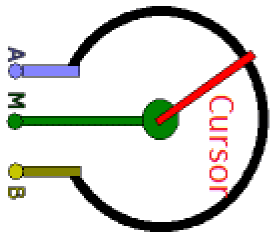
Additionnal tools

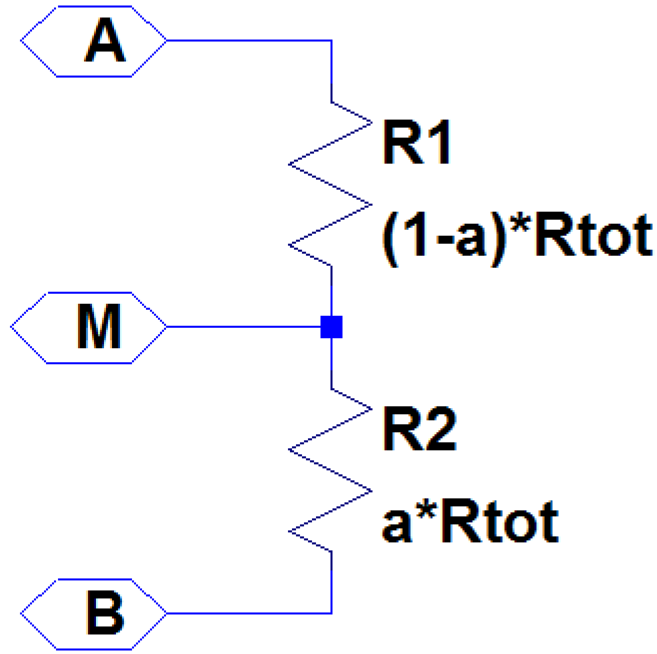
.PARAM .STEP : how to use

- How to simulate Potentiometer

Remider : what is potentiometer ?

Rtot {





Rtot : value of the total resitor measure between nodes A en B. When you move cursor, resistor between Aen M, and between M and B varied, with this relation :
Rtot= R_{AM}+R_{MB}

a : parameter which represent the cursor position :
0<a<1

We need 2 parameters :
- **Rtot : total resistor between nodes A and B**
- **a : cursor position**

Additionnal tools

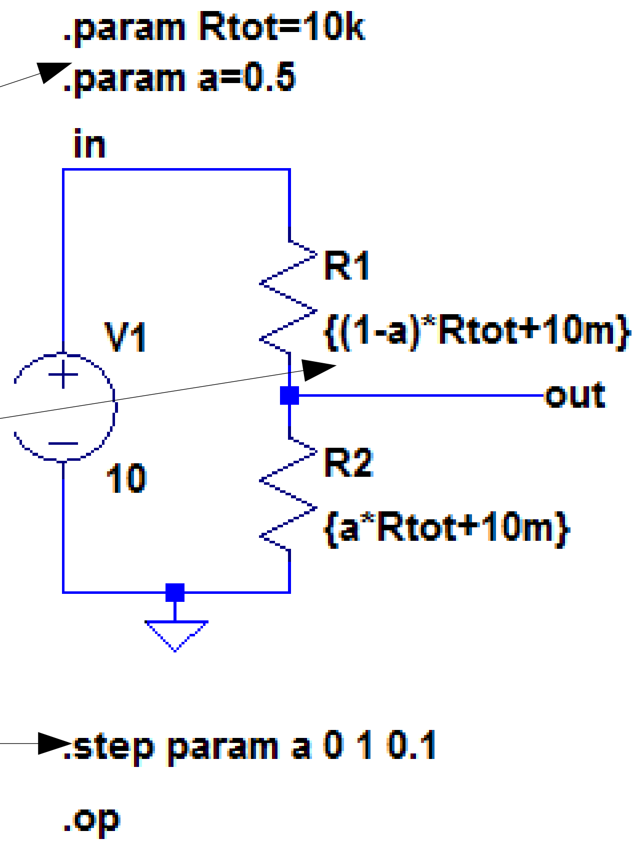
.PARAM .STEP : how to use

- Create parameter : click on .op and write in the text box.

  Syntax : .param param_name=default_value

  Ex : .param Rtot=10k,   .param a=0.5

•Affect to a component : Replace the value of the component by a relation between curly braces.
Ex : 10k => {(1-a)*Rtot+10m}



•Do vary : use the .STEP directive as follow :
.STEP param param_name init Stop Step

Name of parameter

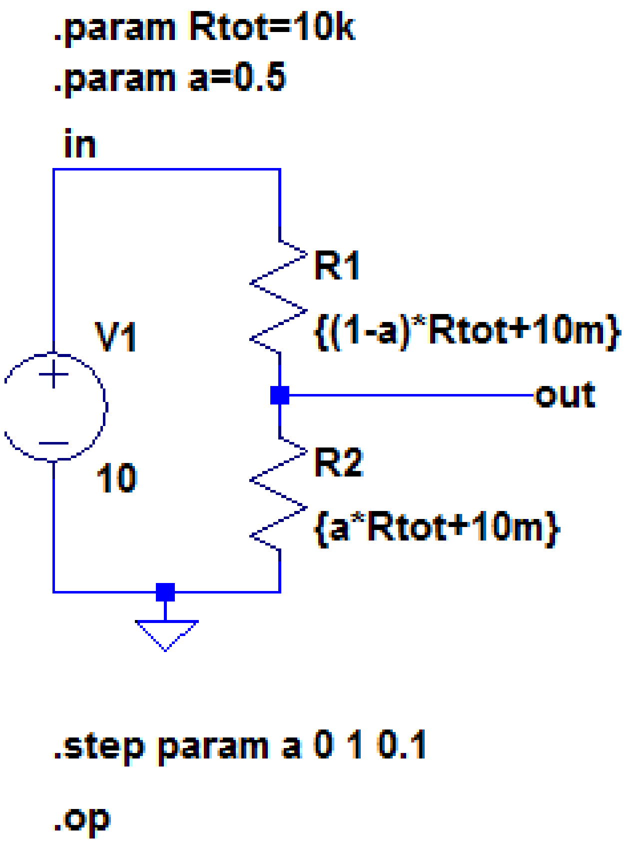
Start value

Stop value

Step between 2 values

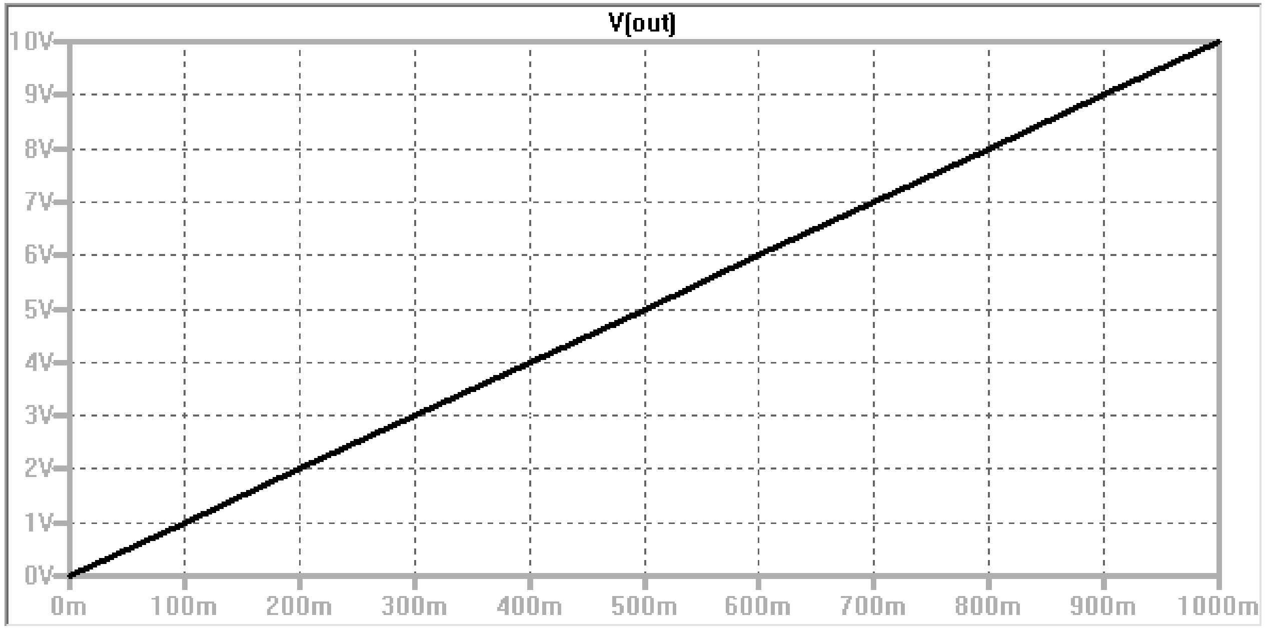
Additionnal tools

.PARAM .STEP : how to use



The result is a curev which give the .OP analysis result for each value of the parameter a,

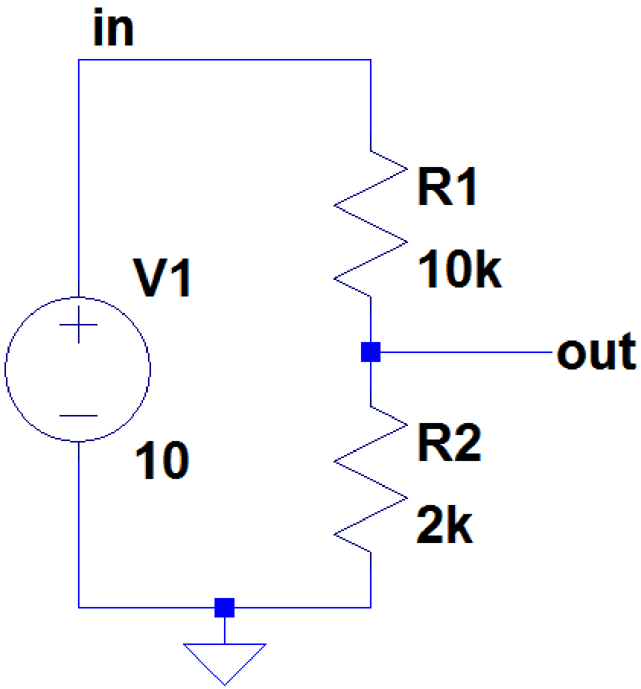
Vout=f(a)



xaxis=>a parameter

Additionnal tools

.PARAM .STEP : try yourself



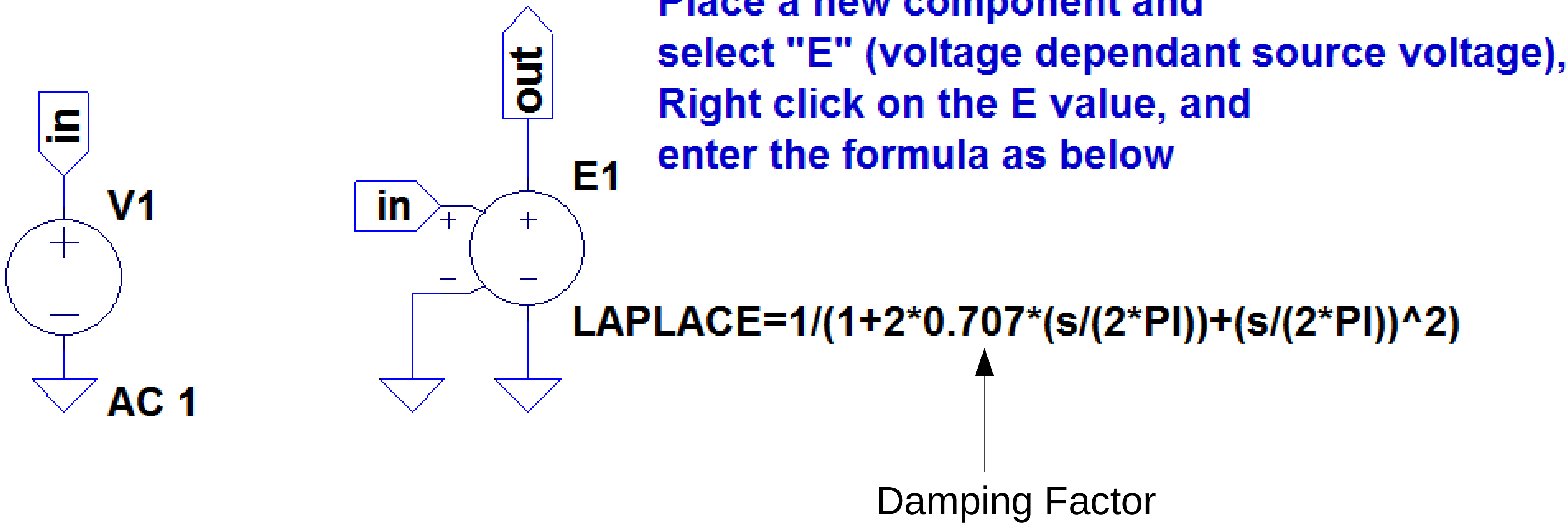
By using PARAM and STEP propose an analysis which allow to see how vary power consumed by R2 in foncion of R2.

For what value of R2 the power is max ?

Additionnal tools

.PARAM .STEP : try yourself

- Resume this schematic and analysis



- Propose modification to see the influence of the damping factor (0.707 in the formula) on the magnitude curve.

Additionnal tools

.MEAS

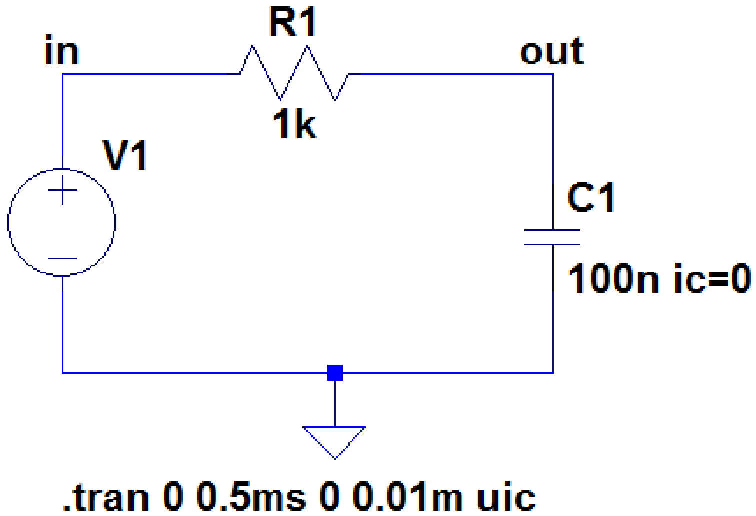
- When simultion is completed, it's possible to perform measurement on data point file using the directive .MEAS.
- MEAS allow :
  - to find a particular point at particular value of the x axis, or when a condition is met...
  - to find maximum value, minimum value, peak to peak value,RMS value, ...
  - To measure a range on a x axis between to particular point,...

Additionnal tools

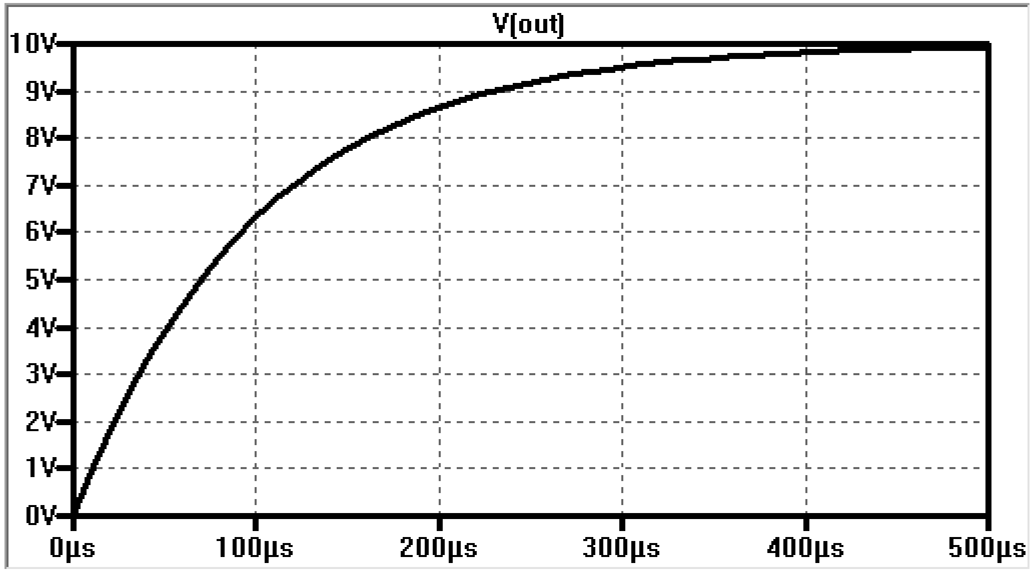
.MEAS - Example

- RC circuit : Step response

Schematic :



Result :



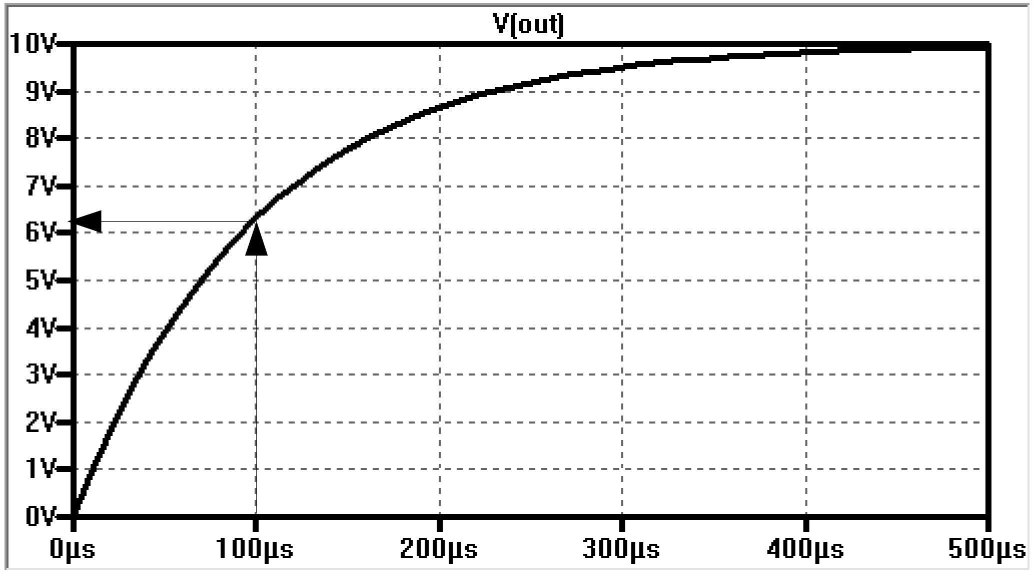
We want to know : what's the value of V(out) for time = R1*C1=100µs

Additionnal tools

.MEAS - Example

- RC circuit : Step response

We want to know : what's the value of V(out) for time = R1*C1=100µs



Using the  button, we add on the schematic the follow directive :

.meas tran vrc find v(out) AT=100u

Measure on the result of the transient analysis

Name of the user defined param it receive the result

On which measure is done

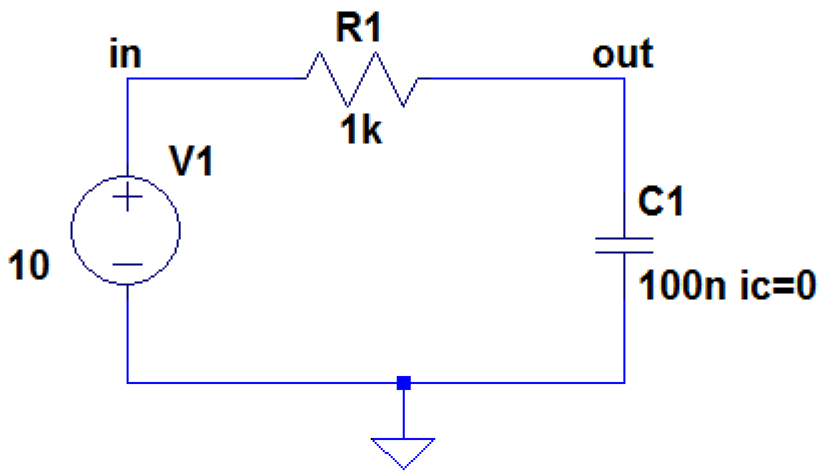
At t=100us

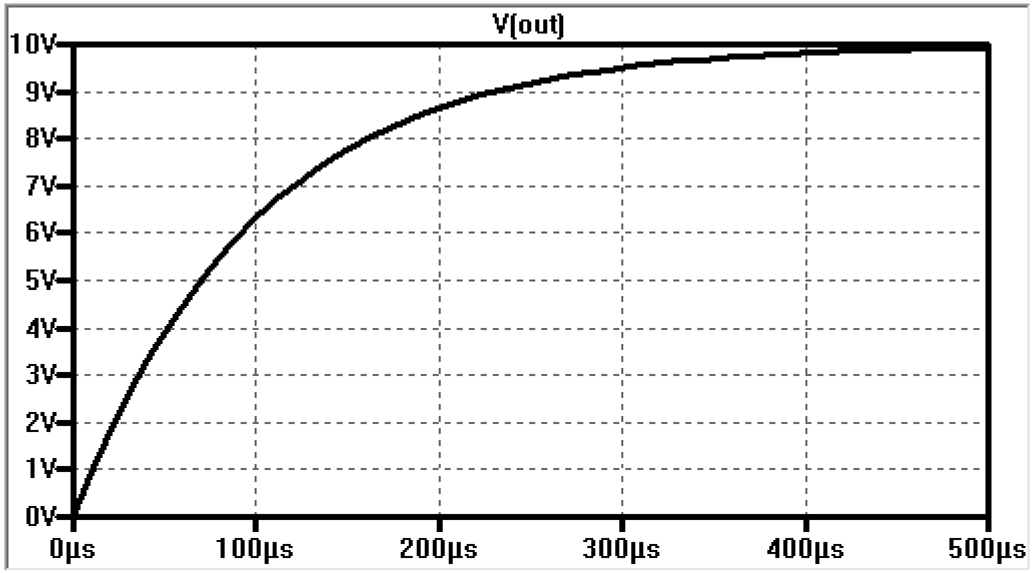
Vsr recieve the value of v(out) at t=100us

Additionnal tools

.MEAS - Example

- RC circuit : Step response

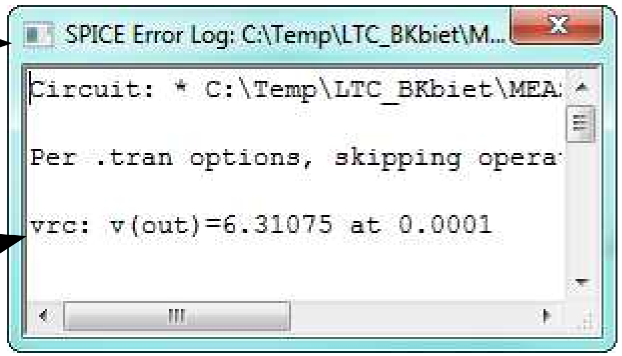




.tran 0 0.5ms 0 0.01m uic

.meas tran vrc find v(out) AT=100u

View => spice error log



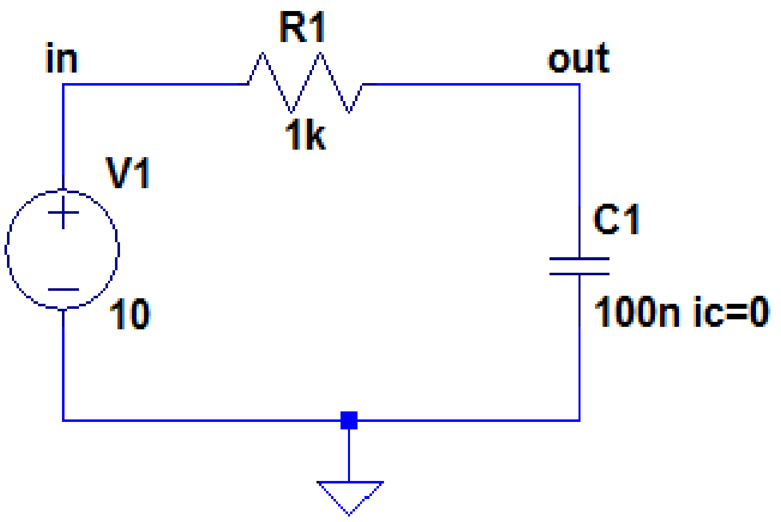
Vsr recieve the value of
v(out) at t=100us

Additionnal tools

.MEAS - Example

- RC circuit : Step response

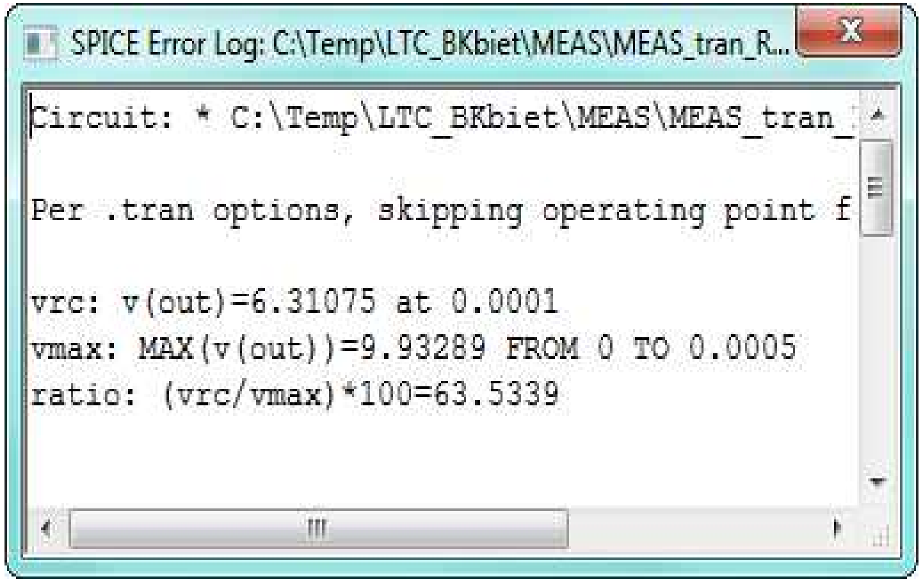
Schematic :




```
.tran 0 0.5ms 0 0.01m uic
```

```
.meas tran vrc find v(out) AT=100u
.meas vmax MAX v(out)
.meas tran ratio param (vrc/vmax)*100
```

View => spice error log



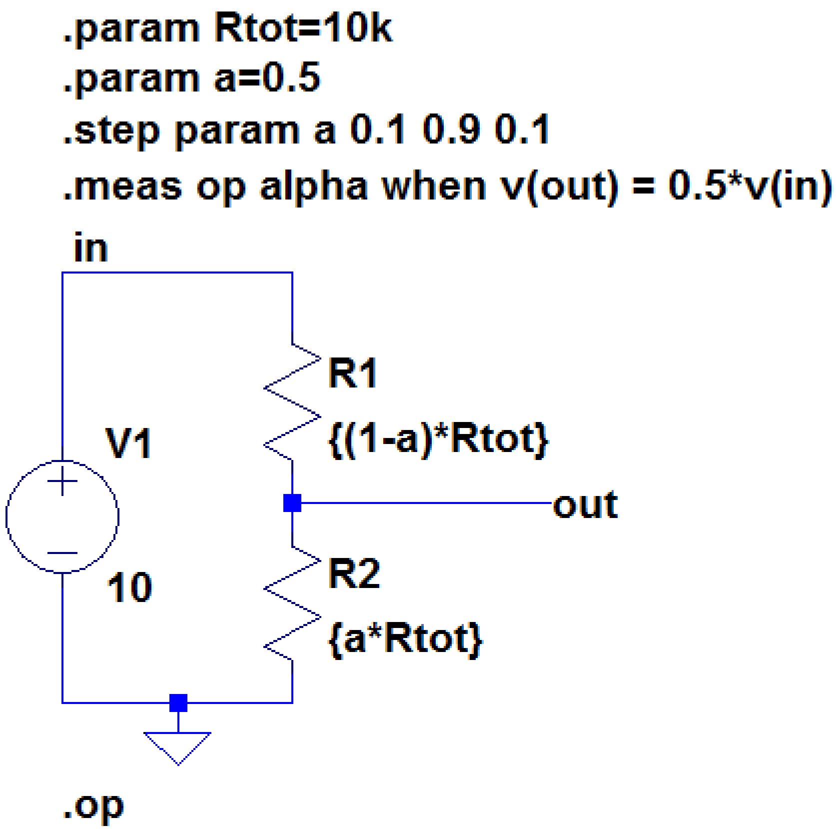
Additionnal tools

.MEAS - Example

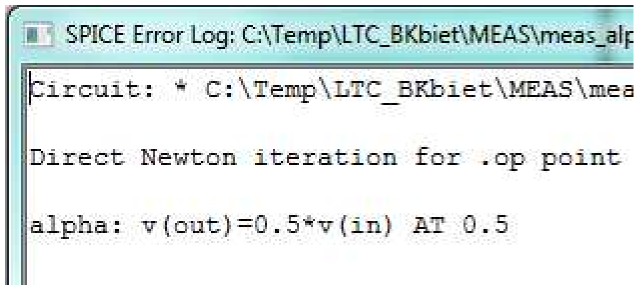
- Potentiometer : find the a value for which

  v(out) = 0,5*V(in)

Schematic :



View => spice error log


SPICE Error Log: C:\Temp\LTC_BKbiet\MEAS\meas_alp

Circuit: * C:\Temp\LTC_BKbiet\MEAS\mea

Direct Newton iteration for .op point

alpha: v(out)=0.5*v(in) AT 0.5

Additionnal tools

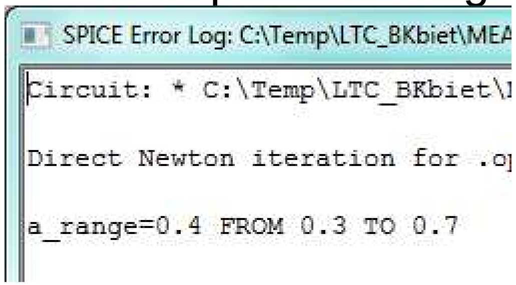
.MEAS - Example

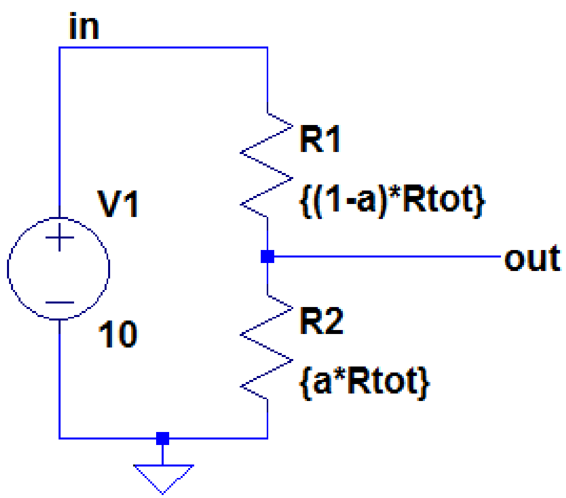
- Potentiometer : find the range on a for which 0,3*v(IN)<v(out) < 0,7*V(in)

Schematic :

```
.param Rtot=10k
.param a=0.5
.step param a 0.1 0.9 0.1
.meas a_range trig v(out) val=0.3*v(in)
+ targ v(out) val=0.7*v(in)
```

View => spice error log





.op

.meas a_range trig v(out) val=0.3*v(in) targ v(out) val=0.7*v(in)

From V(out) = 0,3*V(in)

To V(out) = 0,7*V(in)

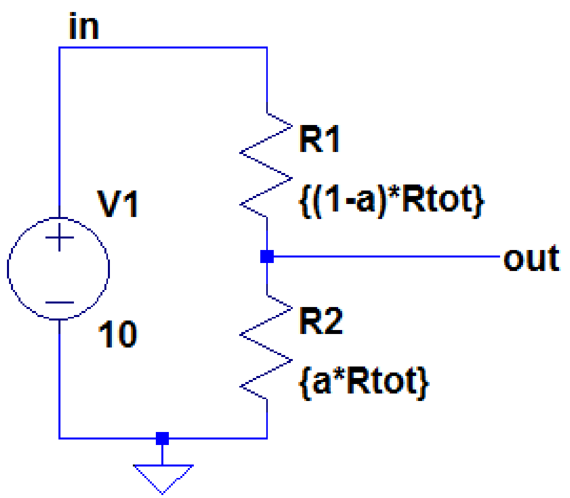
Additionnal tools

.MEAS - Example

- Potentiometer : find the range on a for which 0,3*v(IN)<v(out) < 0,7*V(in)

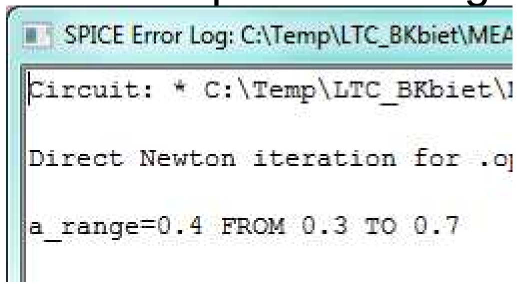
Schematic :

```
.param Rtot=10k
.param a=0.5
.step param a 0.1 0.9 0.1
.meas a_range trig v(out) val=0.3*v(in)
+ targ v(out) val=0.7*v(in)
```



.op

View => spice error log


SPICE Error Log: C:\Temp\LTC_BKbiet\MEA

Circuit: * C:\Temp\LTC_BKbiet\I

Direct Newton iteration for .o]

a_range=0.4 FROM 0.3 TO 0.7

.meas a_range trig v(out) val=0.3*v(in) targ v(out) val=0.7*v(in)

From V(out) = 0,3*V(in)

To V(out) = 0,7*V(in)

Additionnal tools

.MEAS - Example

- Amplifier : find the bandwidth
- Definition : bandwidth is the frequency range for wich magnitude G is between G_{max} and $\frac{G_{max}}{\sqrt{2}}$

.measure tmp max mag(V(out))

Tmp receive the max value mag(V(out)) where mag(V(out)) is the real part of V(out)

.measure BW **trig** mag(V(out))=tmp/sqrt(2) rise=1 **targ** mag(V(out))=tmp/sqrt(2) fall=last

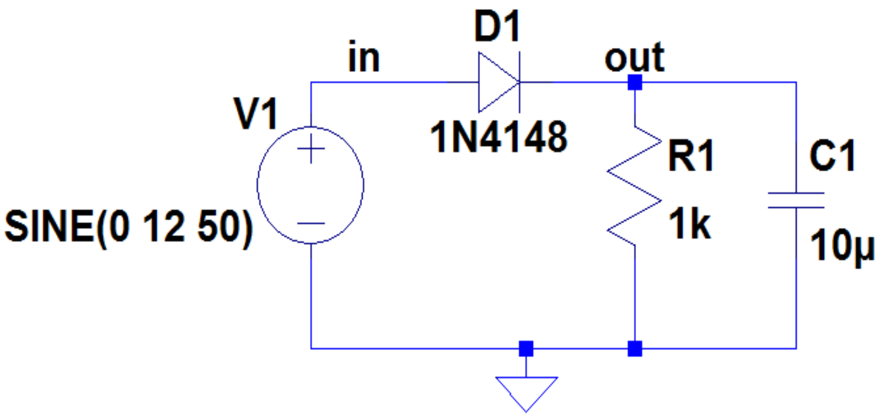
BW receive the frequency range **FROM** which mag(V(out)) cross the value $\frac{G_{max}}{\sqrt{2}}$ for the first time by rising (rise =1) **TO** mag(V(out)) cross the value $\frac{G_{max}}{\sqrt{2}}$ for the last time by falling (fall =1).

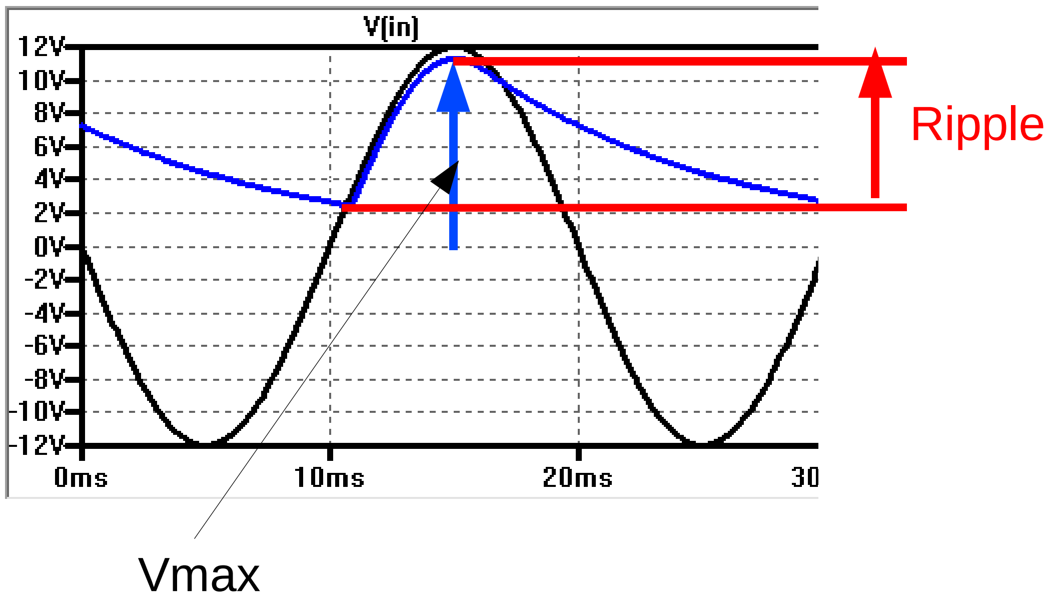
Comment : for more examples, see the help file, press F1 keys.

Summary

Trainning 1

- Take again and modify the rectifier analysis to perform the measurement of the ripple ratio in fonction of the value of the capacitor



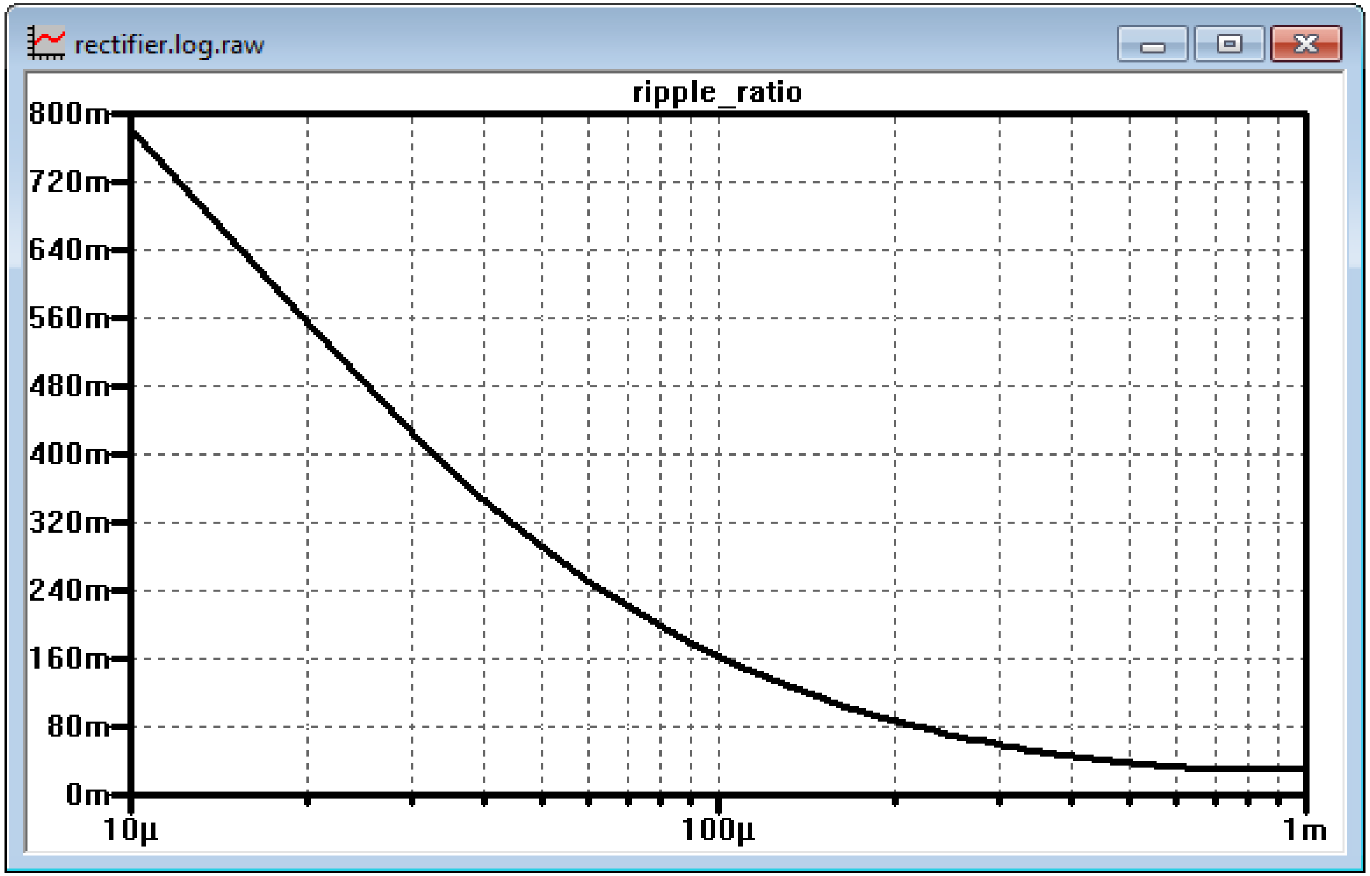


$$Riplle\ ratio = \frac{Ripple}{V_{max}}$$

Summary

Trainning 1

Expected result :



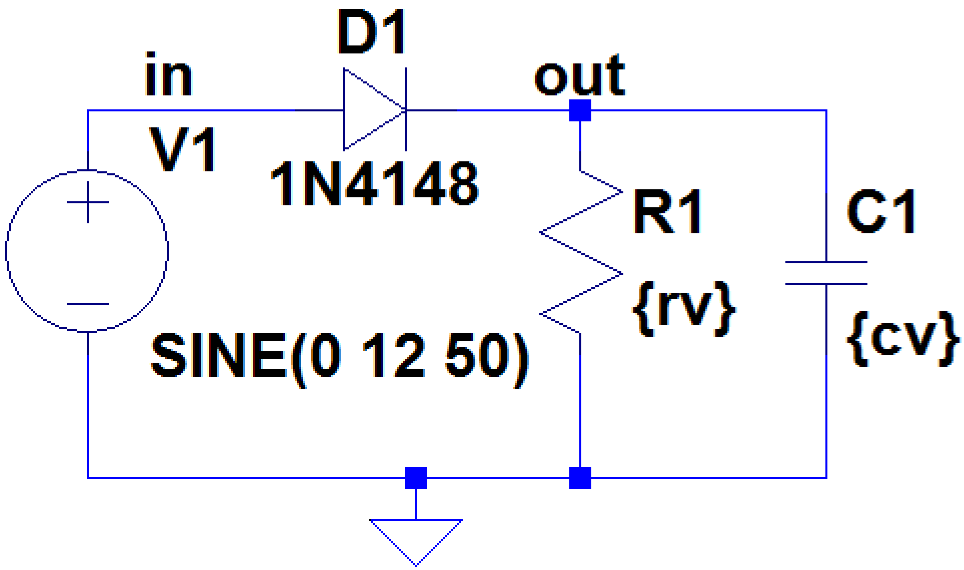
Valeur de C1

Summary

Trainning 1

Correct :

```
.param cv 100n
.param rv 1k
.step param cv 10u 1000u 10u
```




```
.tran 0 200m 10m 0.1m

.meas vmax MAX(v(out))
.meas ripple pp v(out)
.meas ripple_ratio param ripple/vmax
```

Summary

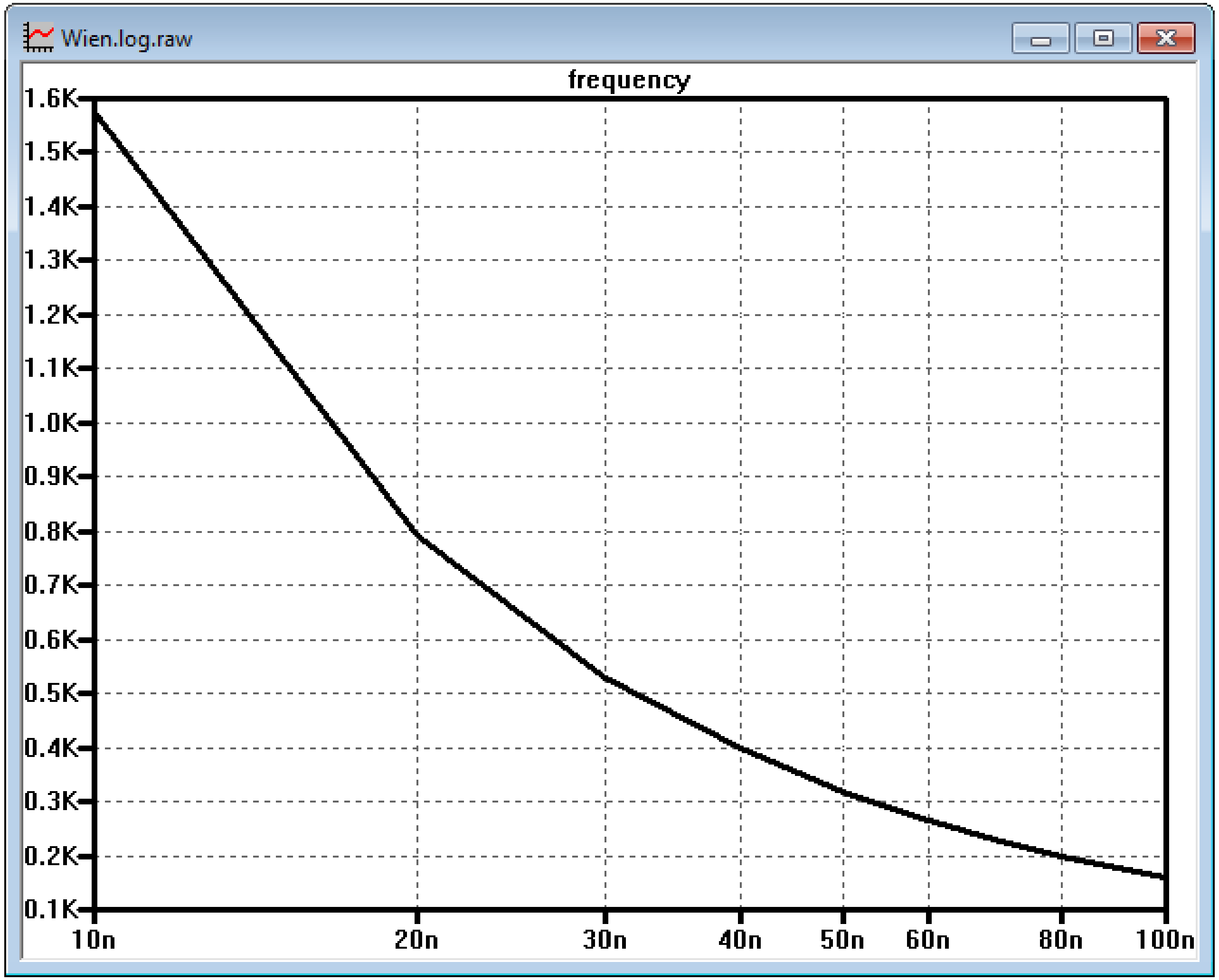
Trainning 2 : frequency measurement

- Open the file « c:\programmeFiles\LTC\example\educational\Wien.asc » and run it.
- Modify the schematic for change the value of C1 and C2 from 0,01uF to 0,1uT by step of 0,01uF
- Add .measure statement to measure the frequency of the signal at output of the Opamp
- Trace the graph frequency versus the value of the 2 capacitor.

Summary

Trainning 2 : frequency measurement

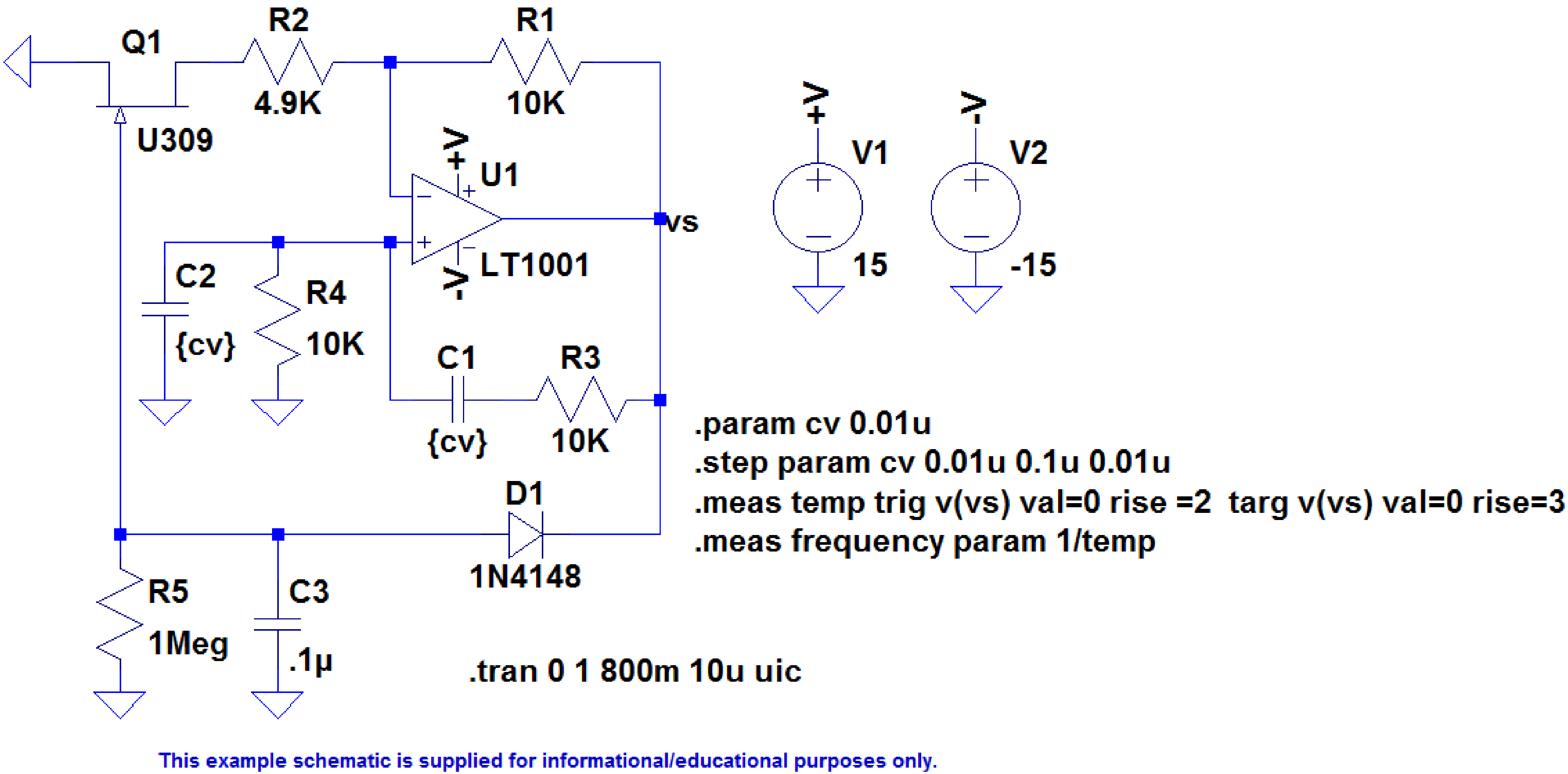
- Expected result



Summary

Trainning 2 : frequency measurement

- Correct :



Web Link

- Power Point tutorial

  www.ltspice.linear.com/software/LTspiceGettingStartedGuide.pdf

- Complete documentation

  www.ltspice.linear.com/software/scad3.pdf