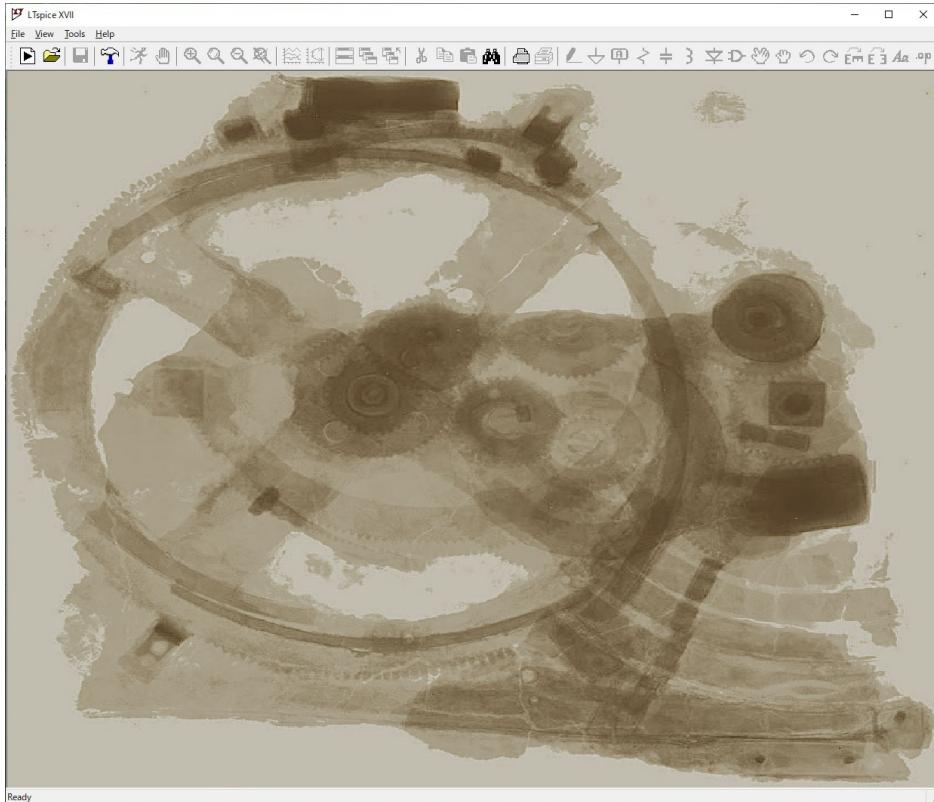
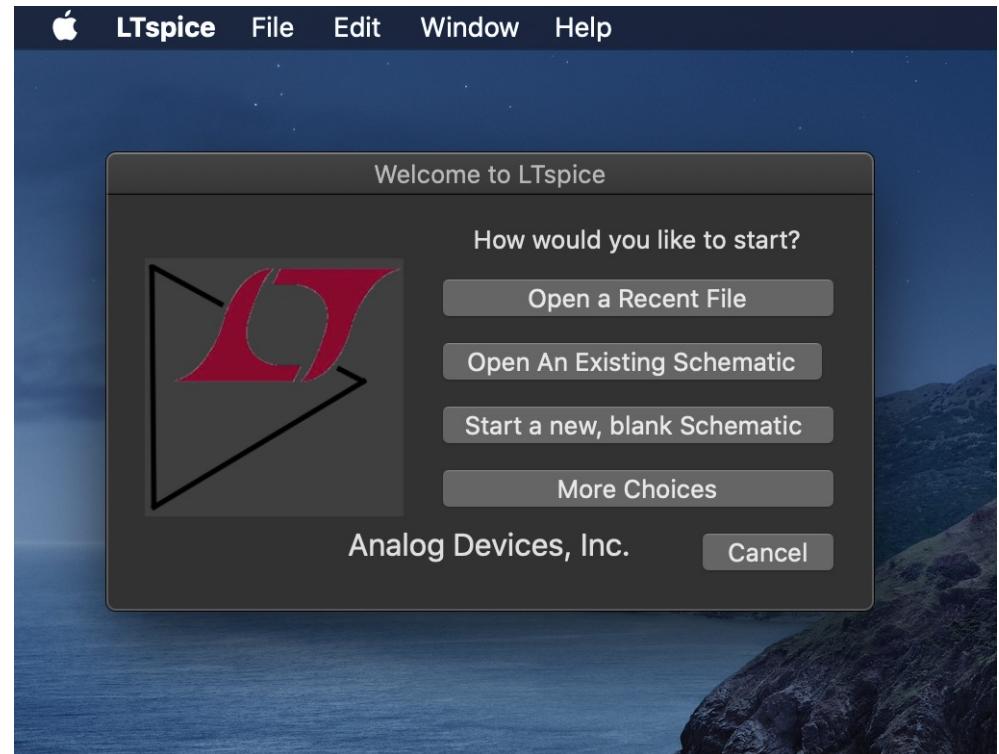


About basic analysis in LTspice circuit simulator

Startup window of LTspice



LTspice application
after booting the Windows version

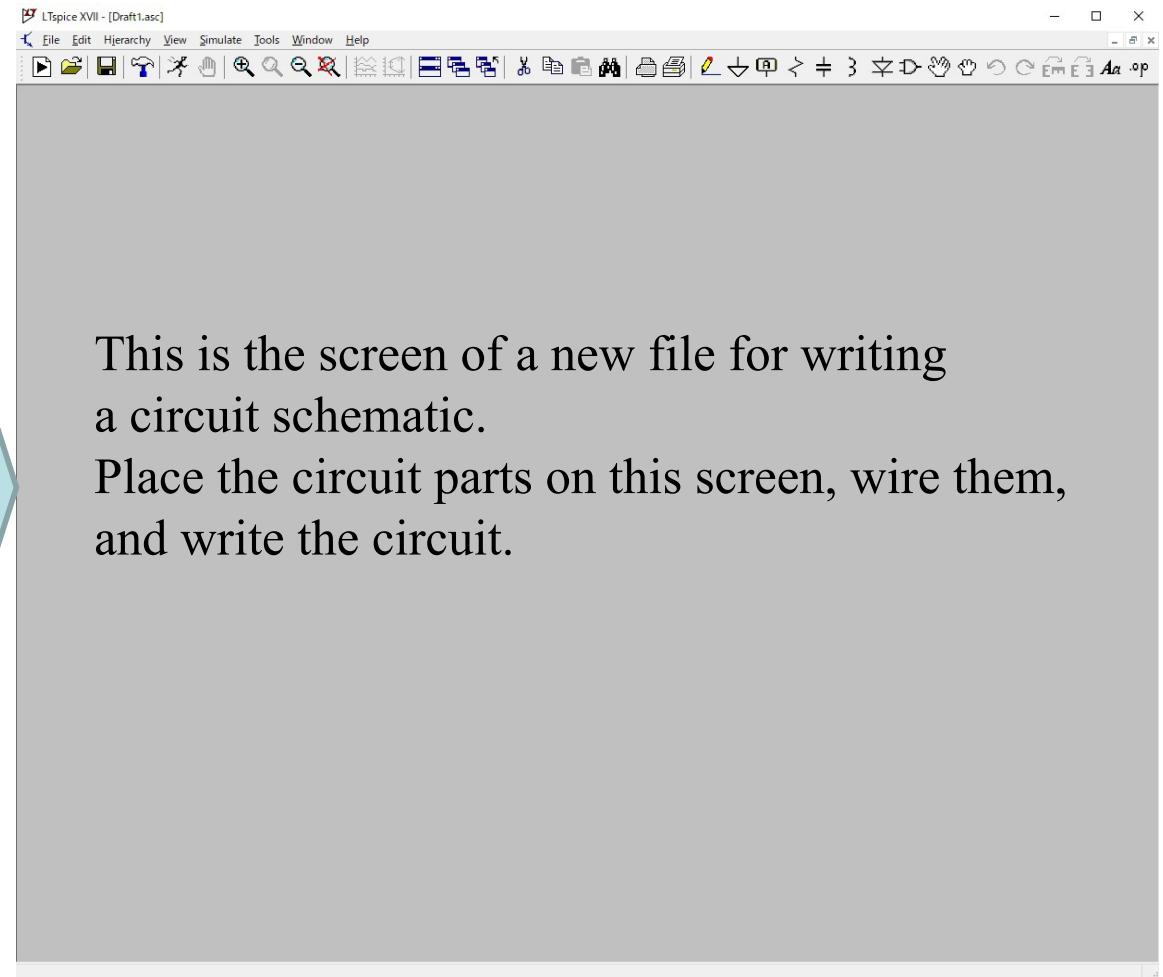
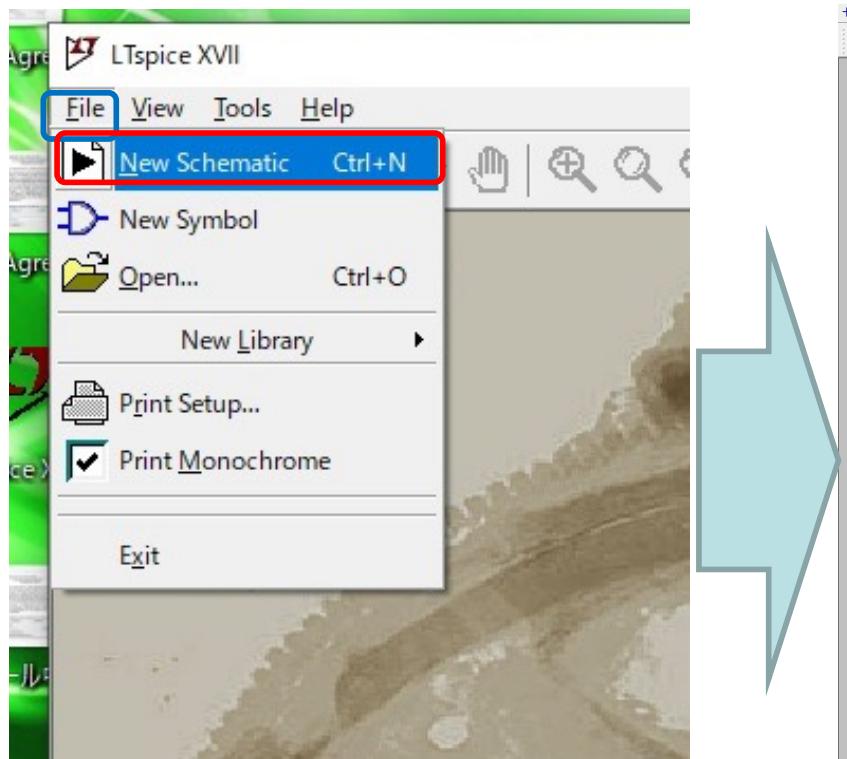


LTspice application window
after booting the Mac version

- In the Mac version, it starts with a menu for opening a file.
- After selecting new schematic or selecting an existing file, the circuit diagram creation window appears.

New Schematic(Windows Ver.)

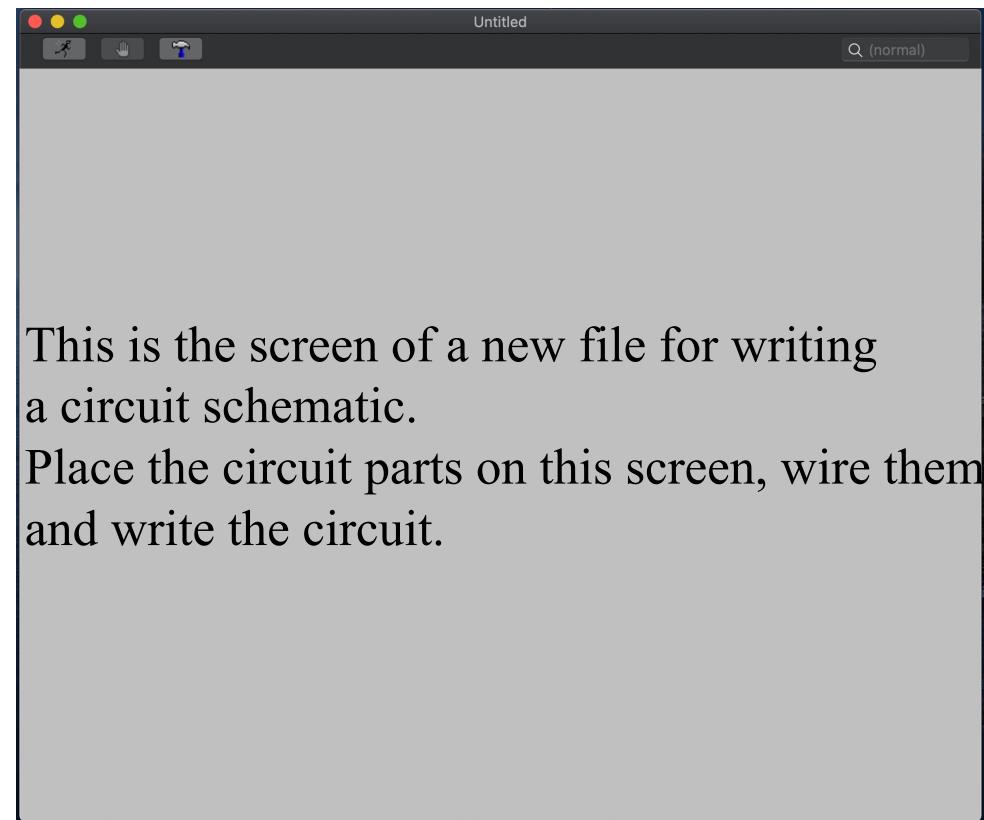
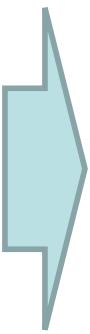
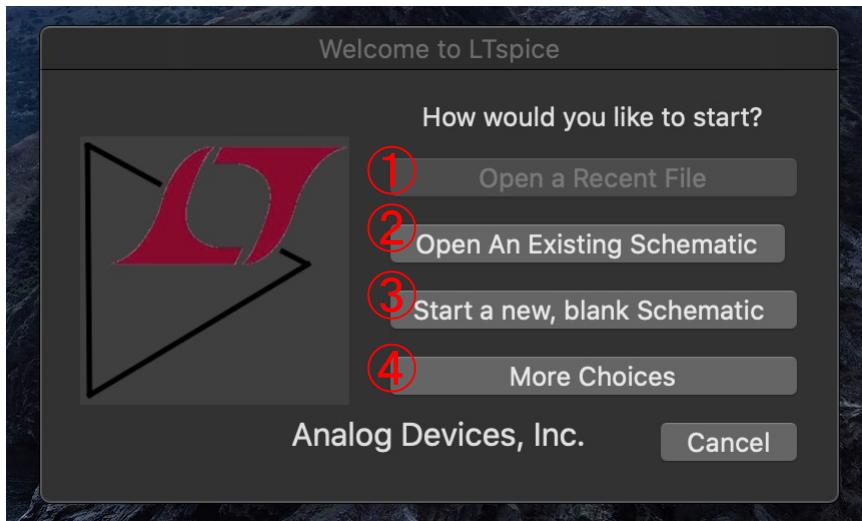
You can open a new file to write a schematic (Circuit Schematic) by selecting **New Schematic** from the **File** menu of LTspice.



This is the screen of a new file for writing a circuit schematic. Place the circuit parts on this screen, wire them, and write the circuit.

New Schematic(Mac Ver.)

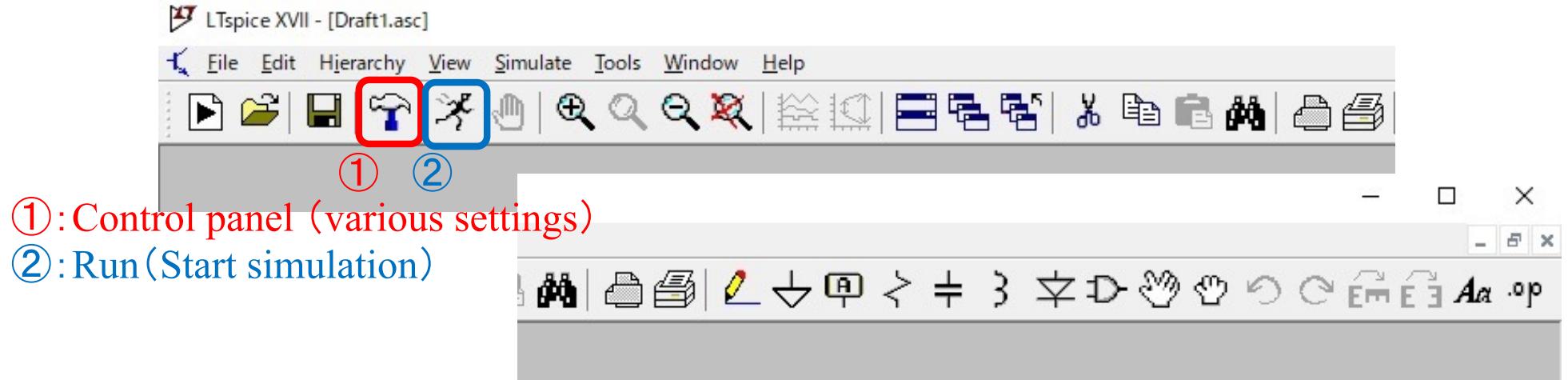
Select "Start a new, blank Schematic" to display a new circuit schematic window from the file selection menu that appears after startup.



This is the screen of a new file for writing a circuit schematic.
Place the circuit parts on this screen, wire them, and write the circuit.

- ① Open the file that was opened last time
- ② Open the previously created file
- ③ Create a new file
- ④ Select another method

Menu bar and Tools bar

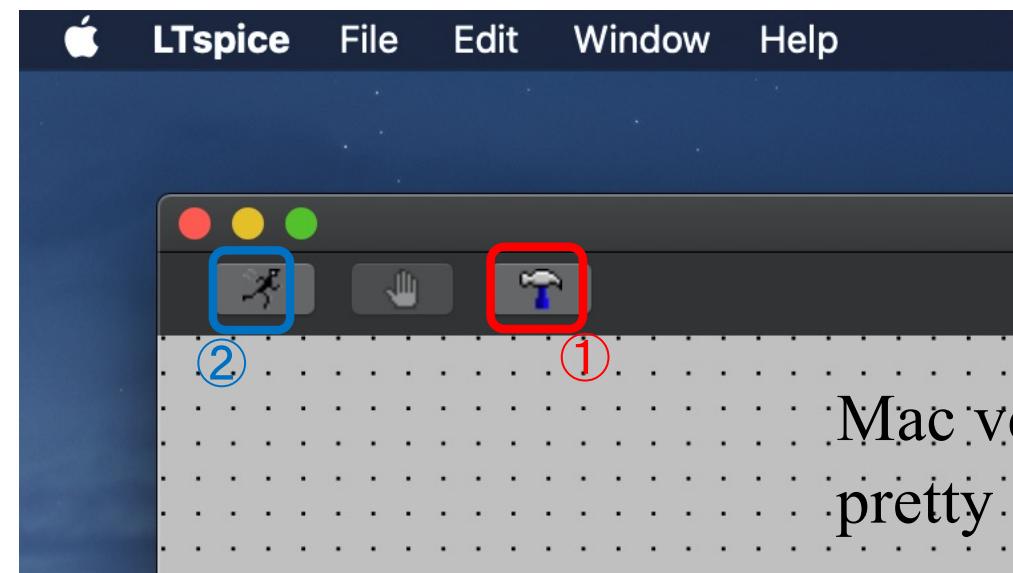


①: Control panel (various settings)

②: Run (Start simulation)

Menu bar and Tools bar of Windows Ver.

Most operations can be done with the menu (next page) that appears by right clicking on the window.



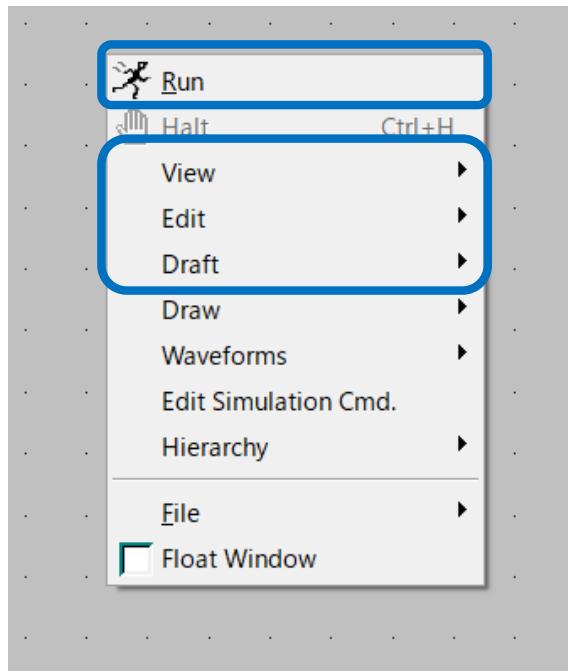
Menu bar and Tools bar of Mac Ver.

Operation Menu

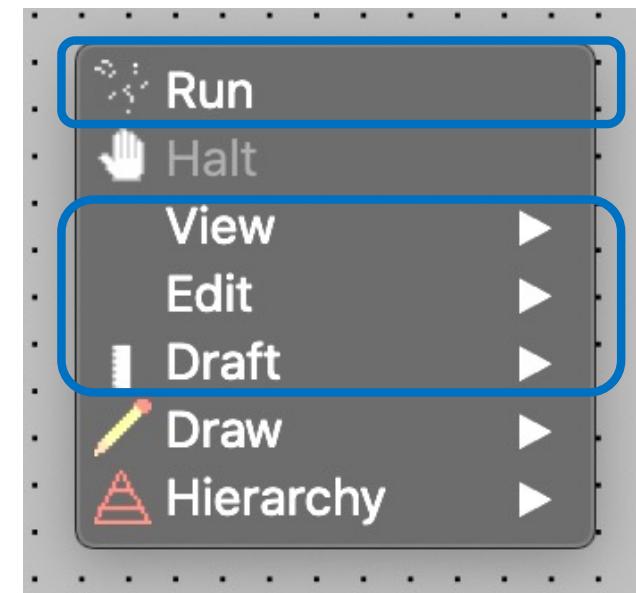
The operation menu is a menu that appears by right-clicking on the circuit schematic window (same for the graph window) for both Windows and Mac.

Most of the operation is done using this menu (or shortcut).

The important items are “Run”,
and three menus surrounded by a blue frame .

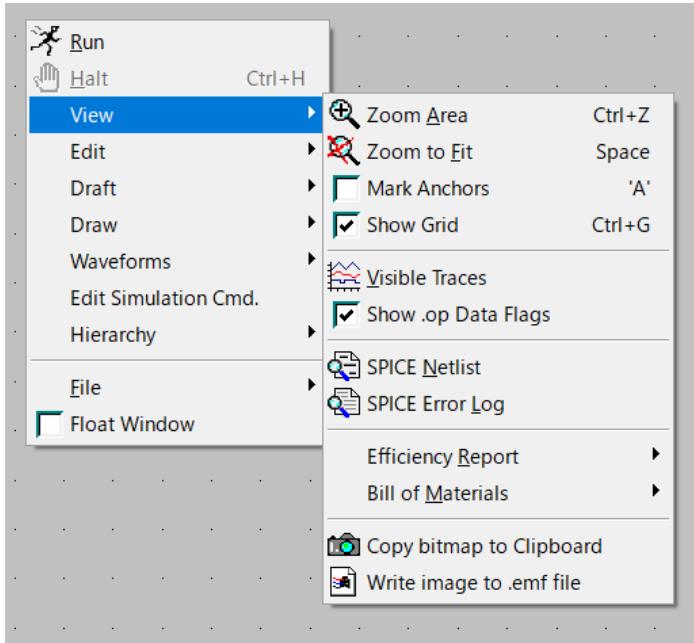


Windows Ver.

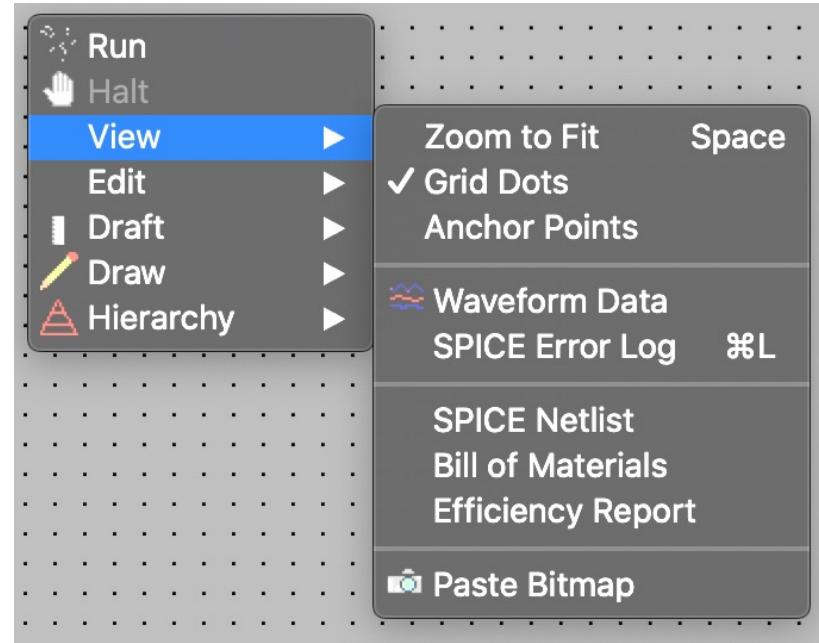


Mac Ver.

“View” menu



Windows Ver.



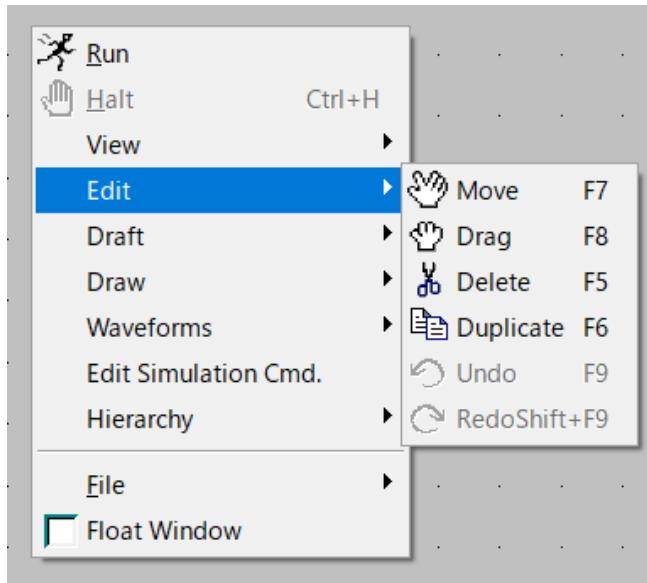
Mac Ver.

After all, the Mac version is simple and slightly different, but the menus that are often used are the same. "Show Grid" on Windows and "Grid Dots" on Mac have different notations, but both have the function of "displaying the grid on the circuit diagram creation screen".
(It is easier to determine the position of circuit parts if the grid is displayed, so check this item.)

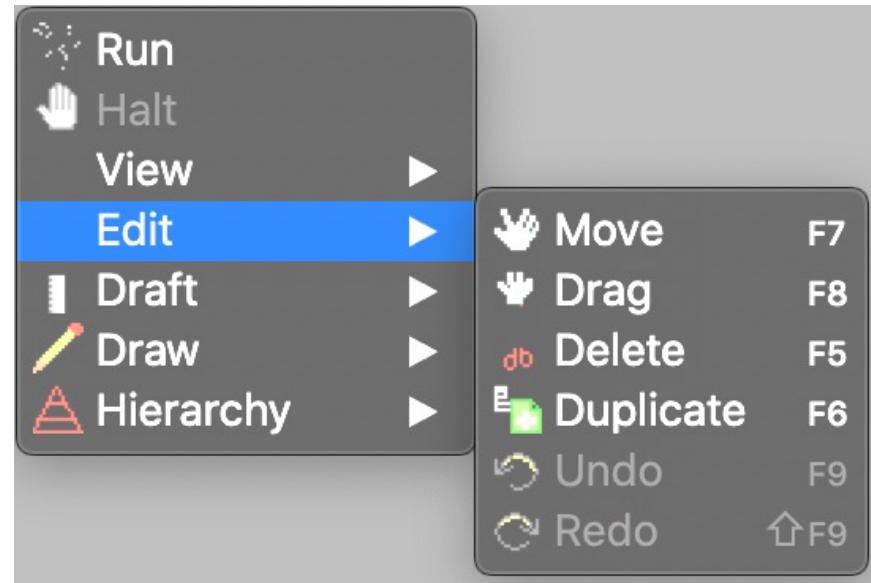
[Frequently used items in the “View” menu]

- Win “Visible Traces”, Mac “Waveform Data”: If the simulation is finished, the result is displayed.
 - “SPICE Netlist”: Circuit expression in SPICE. Output the textual representation of the circuit connection.
 - Win “Copy bitmap to Clipboard”: Created circuit diagram as a bitmap file to the clipboard *
 - Win “Write image to .emf file”: Save the created circuit diagram as an EMF format image file *
- * Functions not available on Mac: For Mac, only screenshots are available to use the circuit diagram with other software.

Edit menu



Windows版



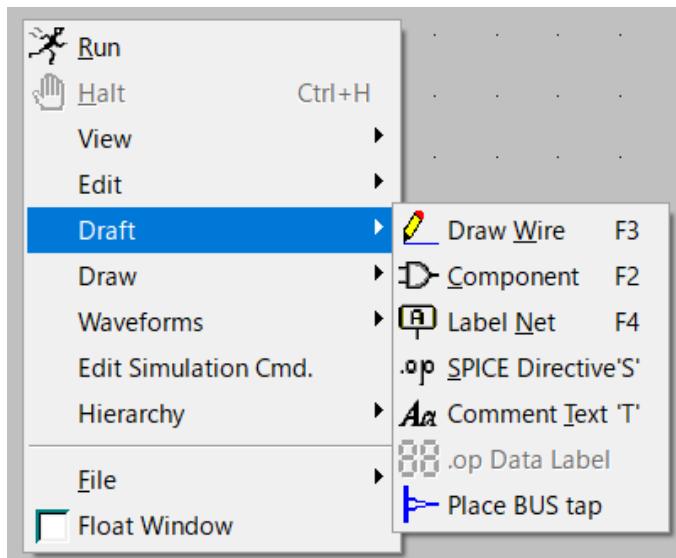
Mac版

“Edit” menu is exactly the same, including shortcuts.

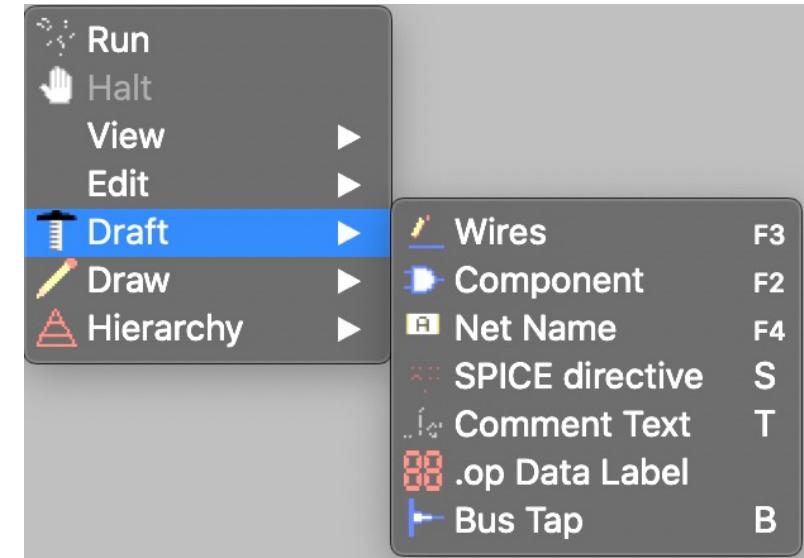
[“Edit” menu]

- "Move": Movement of parts or the entire circuit. However, if the parts are moved after wiring, only the parts will move.
- "Drag": Same as moving, but if you move the part after wiring, you can move the part while the wiring is connected.
- "Delete": Delete parts and circuits.
- "Duplicate": Duplicate of parts and circuits. At this time, the symbols representing the parts are automatically changed so that they do not overlap.
- "Undo, Redo": Redo the operation. Note that it is not "Ctrl + z (Shift + Ctrl + z)" unlike the normal case.

Draft menu



Windows版



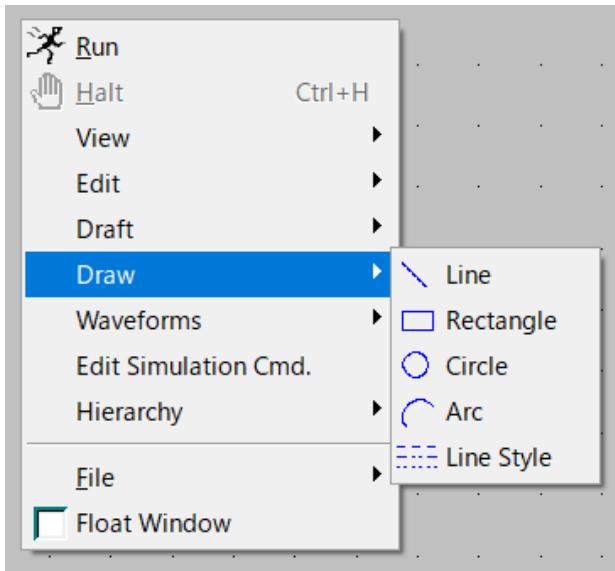
Mac版

“Draft” menu is almost the same including shortcuts.

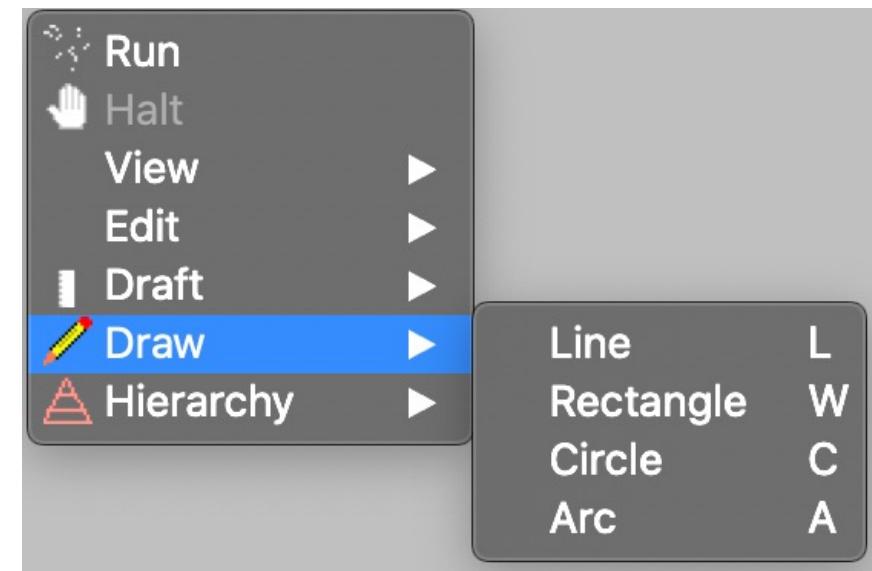
[“Draft” menu]

- Win “Draw Wire”, Mac “Wires”: Wire. -Component: Select parts.
- Win “Label Net”, Mac “Net Name”: Select the ground symbol, name the wiring, and define the input / output points.
- “SPICE directive”: Defines an analysis command.
- “Comment Text”: Enter a comment in the circuit schematics.
- “.Op Data Label”: Displays the DC response value during simulation.
- Win “Place BUS tap”, Mac “Bus Tap”: Collecting multiple bits in digital circuit data, etc. Make multiple wirings (bus wiring).

Draw menu



Windows Ver.



Mac Ver.

The Draw menu is almost the same.

It is used when adding new parts and preparing a circuit symbol by yourself.

- In the Windows version, the line style can also be changed from the menu.
- For Mac version, right-click the drawn line and change it with the menu that appears.

Hierarchy menu



Mac Ver.

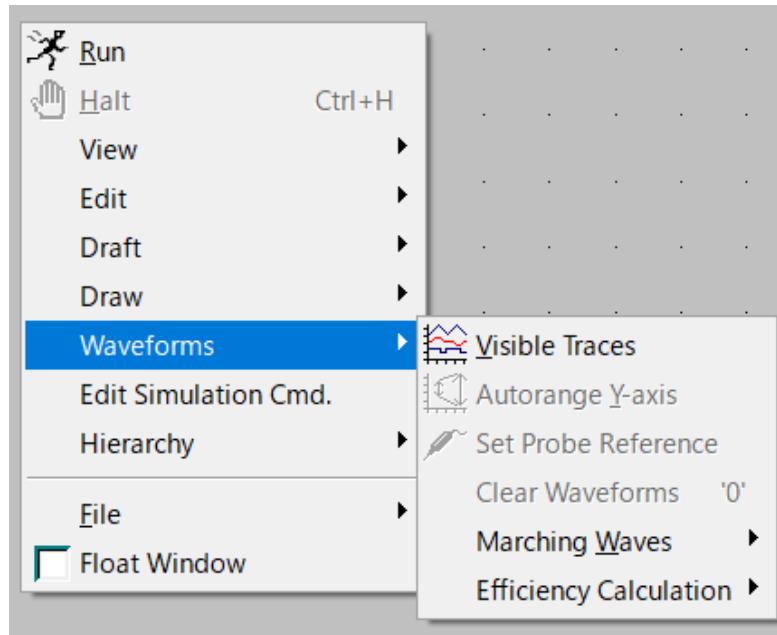
The same applies to the Windows version.

As the circuit scale increases, the number of commonly used circuit elements increases.

This commonly used circuit element can be represented by Symbol and treated like a component. It is used to display this symbol.

Waveforms menu

Windows Ver. only.



The Windows version has a Waveforms menu that allows you to display simulation results and operate the result display.

It's not in the Mac version, but you can do the same in other ways.

Basics of SPICE circuit analysis

- Transient analysis
- AC analysis

Example of circuit analysis implemented in the circuit simulator

Basic analysis

Transient analysis:

A method of analyzing changes in the voltage at each node in the circuit and the current flowing through each element after the circuit is turned on.

AC analysis:

A method for analyzing the frequency characteristics of a circuit. Simulation of the amplitude characteristic, which is the frequency dependence of the output signal amplitude, and the phase characteristic, which is the frequency dependence of the phase difference between the input signal and the output signal.

DC Sweep analysis:

A method of analyzing the voltage and current of each node with respect to the value of the DC power supply in the circuit as a parameter. It is used when analyzing DC (direct current) characteristics of diodes, transistors, operational amplifiers, etc.

Noise analysis:

A method of analyzing the frequency characteristics of noise in a circuit.

DC Operating Point analysis:

A method of analyzing the DC voltage and DC current of each node when the electronic circuit is in a steady state.

Example of circuit analysis implemented in the circuit simulator

Applied analysis

Parametric analysis:

A method for analyzing changes in circuit operation when the resistance value, capacitance value, power supply voltage, etc. are changed.

Temperature analysis:

A method for analyzing how a circuit is affected by temperature changes due to the environment in which the circuit is placed.

Monte Carlo analysis:

A method for analyzing how errors and variations in parts used in a circuit affect the circuit.

Transient analysis

Transient analysis

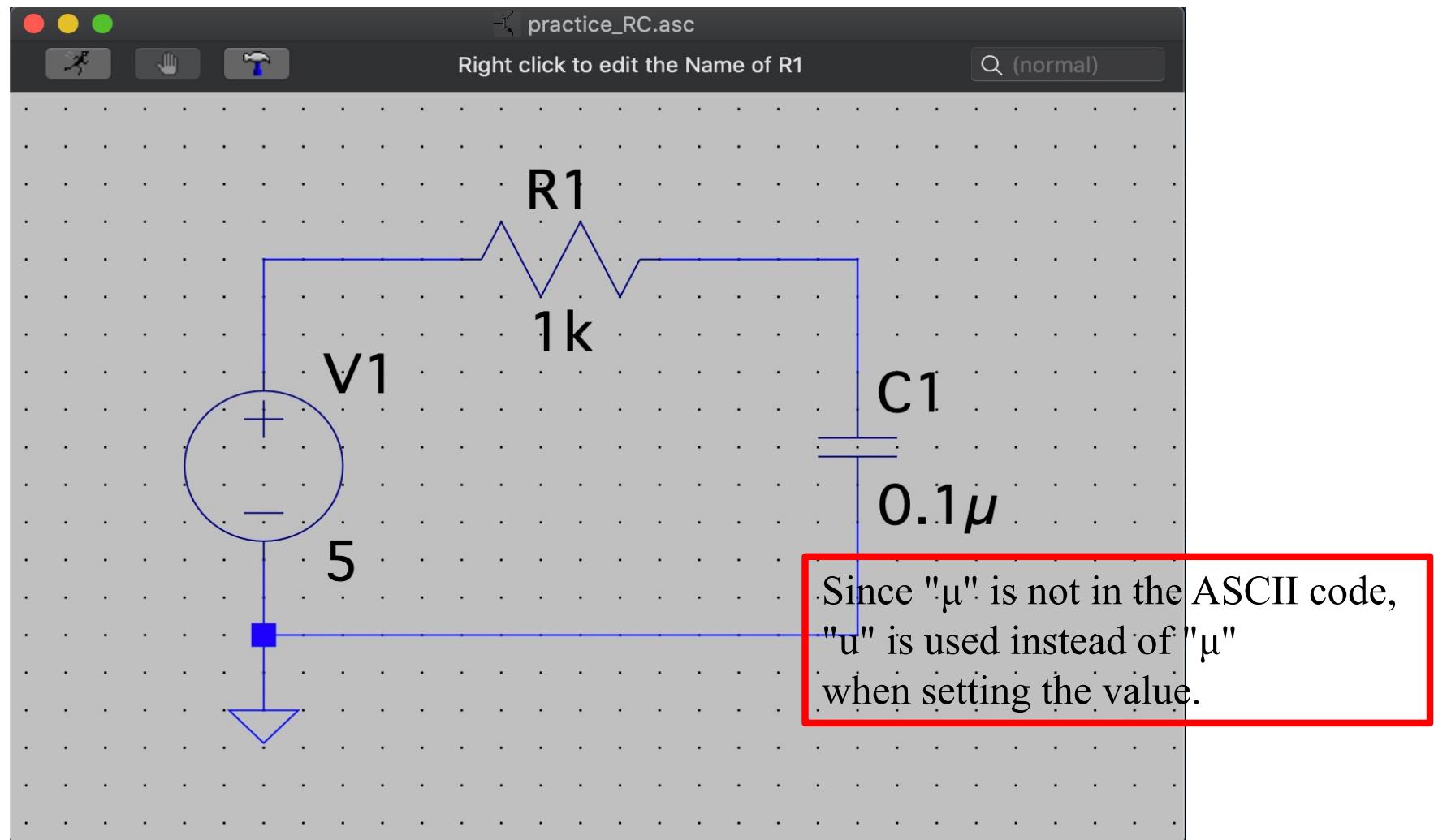
- A simulation method that analyzes changes over time in the voltage of each node and the current flowing through the element due to the input signal from the moment the power of the electronic circuit is turned on.
- It is used to simulate the behavior (transient state) of the circuit until the circuit operation becomes stable (steady state) after the power is turned on.
- It is used to simulate the actual signal observed with an oscilloscope.

That is, **the horizontal axis: time, the vertical axis: signal voltage** (in some cases, the current can represent the simulation result).

Test circuit for “Transient analysis

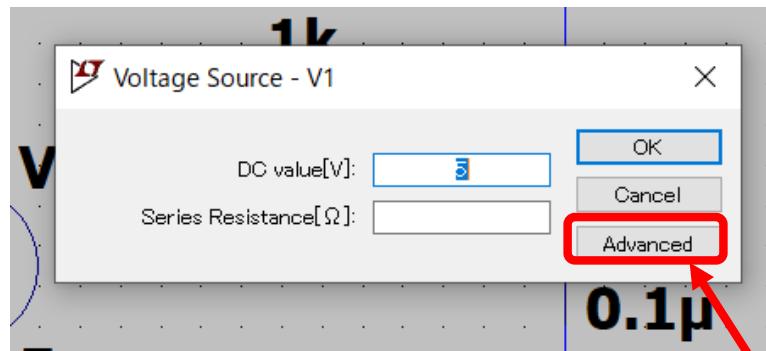
To practice transient analysis, create the circuit schematic shown in below.

* The document shows the operation on the Mac version, but it is almost the same on the Windows version.

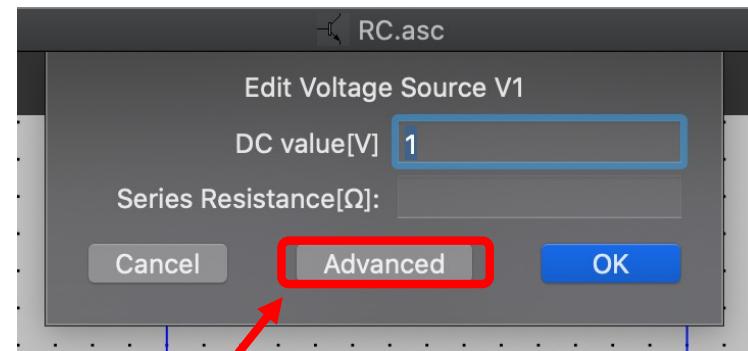


Change power supply to pulse (square wave) voltage source

- When the voltage source "Voltage" is placed, the default is the DC voltage source.
- It will be change that to a pulse (square wave) voltage source.
- When you move the mouse cursor over the voltage source symbol, the cursor changes to the shape of a pointing finger, so right-click in that state.
- After that, the menu like below will appear. Then select "Advanced" to proceed to the detailed setting screen of "Voltage" function.



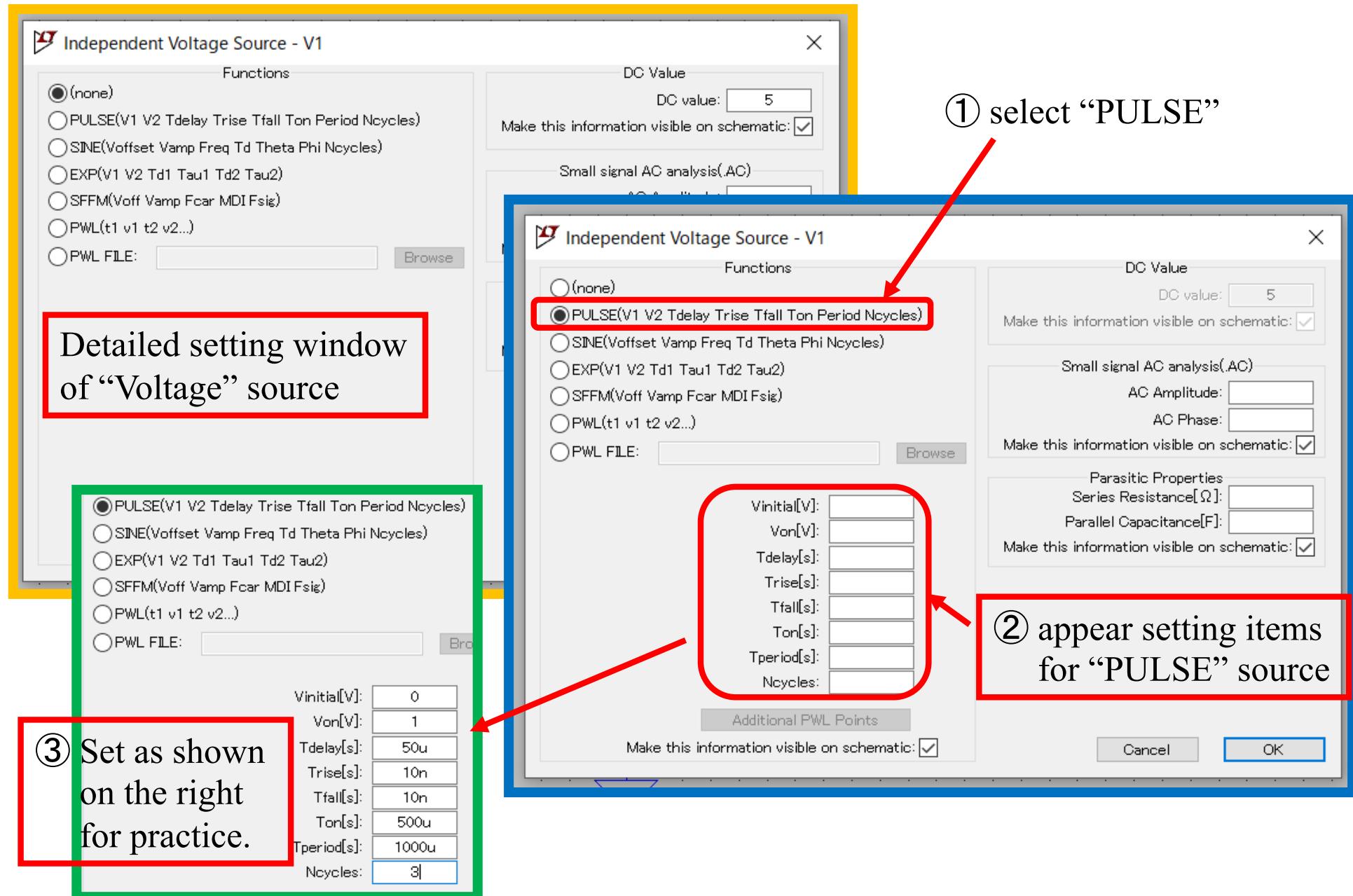
Windows Ver.



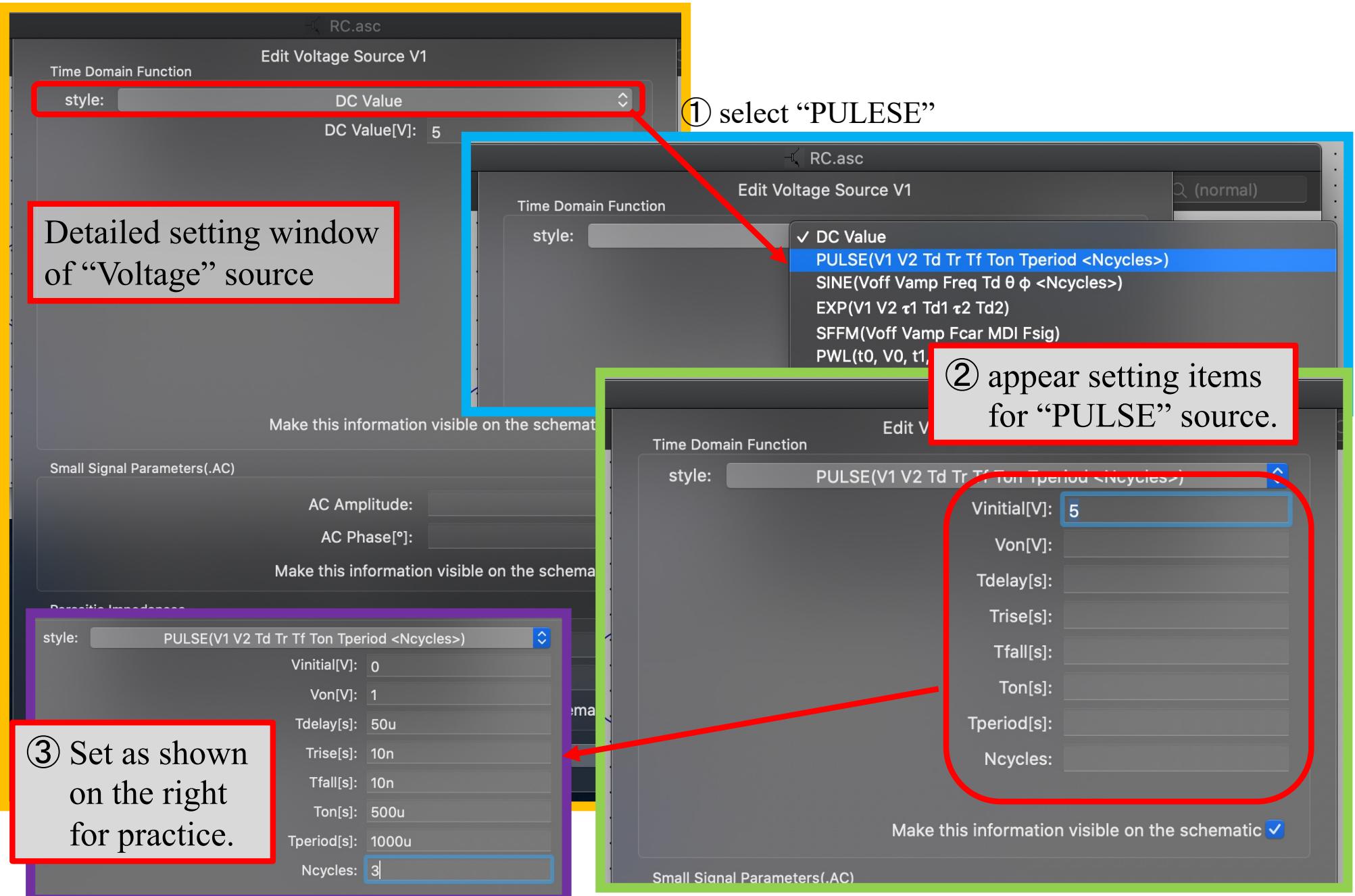
Mac Ver.

To proceed to detailed setting, press “Advanced” button.

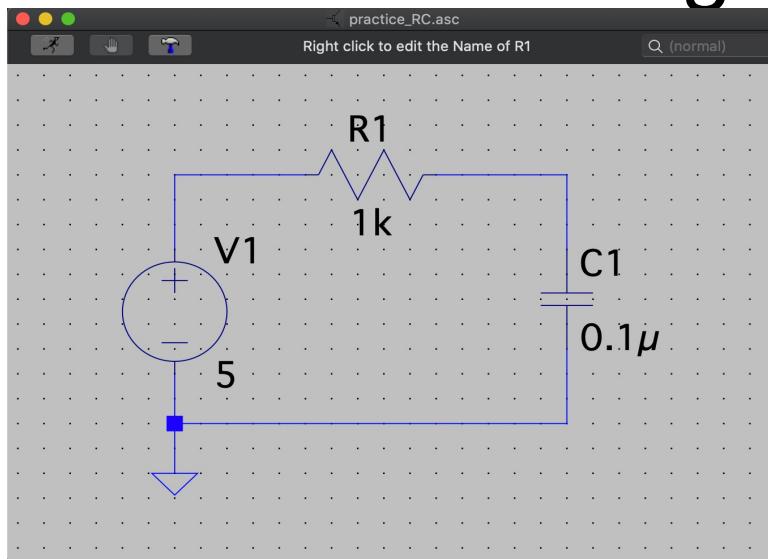
Detailed setting of “Voltage” source (Windows)



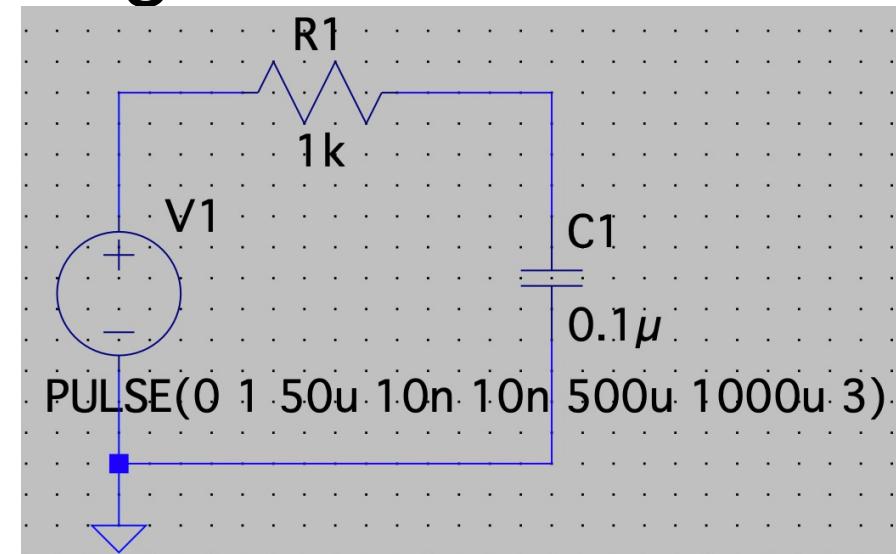
Detailed setting of “Voltage” source (Mac)



Comparison between before and after setting of "Voltage" source



Before setting



After setting

Description format of “PULSE” source

PULSE(V1 V2 Td Tr Tf Ton Tperiod <Ncycles>)

V1 (Vinitial[V]): Initial voltage. That is the voltage value at OFF (low voltage)

V2 (Von[V]): Voltage value at ON (high voltage)

Td (Tdelay[s]): Delay time to the first ON from the start of simulation

Tr (Trise[s]): Time it takes to rise from OFF to ON

Tf (Tfall[s]): Time it takes to fall from ON to OFF

Ton (Ton[s]): Duration time of ON (high voltage)

Tperiod (Tperiod[s]): Period of repetition of signal

Ncycles: The number of repetitions of cycle

SPICE command for Transient analysis

In general SPICE, it is necessary to input a command called SPICE command in order to execute the simulation.

In LTspice, enter the SPICE command on the schematic screen.

- The command for Transient analysis is ".tran".
- The ".tran" command also describes the parameters required for analysis.
- The format of the ".tran" command is as follows.

```
.tran 0 End time Start time for data saving Maximum step time
```

0: Simulation start time. This is **optional**, since the default is 0.

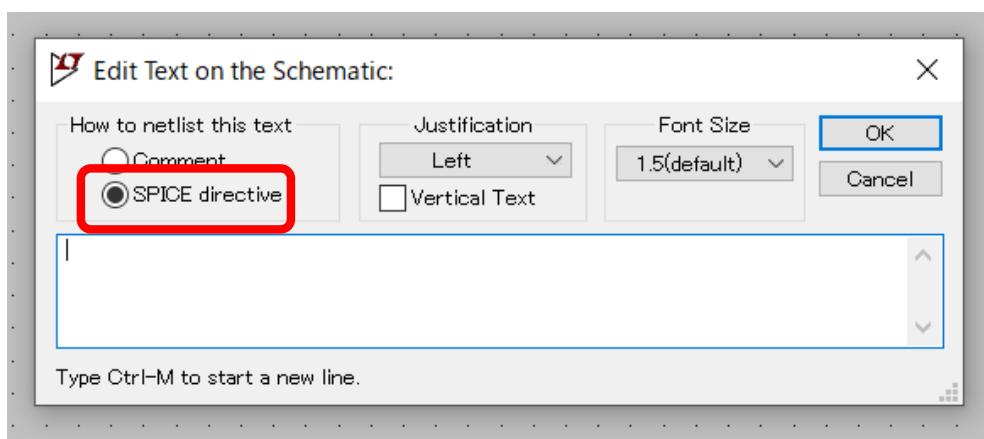
End time: Time to finish the simulation. Set according to the frequency of the input / output signal.

Start time for data saving: Set from what time the data obtained by the simulation is saved (also displayed) as the result. (**Optional**)

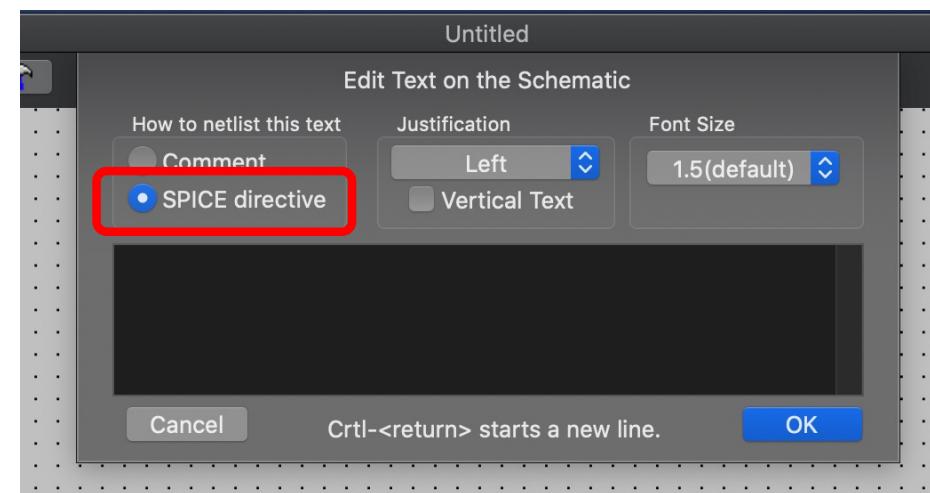
Maximum step time : In the simulator, the calculated time interval is automatically adjusted according to the change in the signal. You can specify the maximum value of this time interval. (**Optional**)

Entering “.tran” command

- Command input menu appears when you right-click on a circuit schematic window where there are no parts.
- Select "Draft" ⇒ "SPICE directive" from that menu.
 - * You may enter the shortcut key "s" on the schematic input screen.
- The following command input screen will appear, so enter the command here.
 - * Same flow for other command input



Windows Ver.



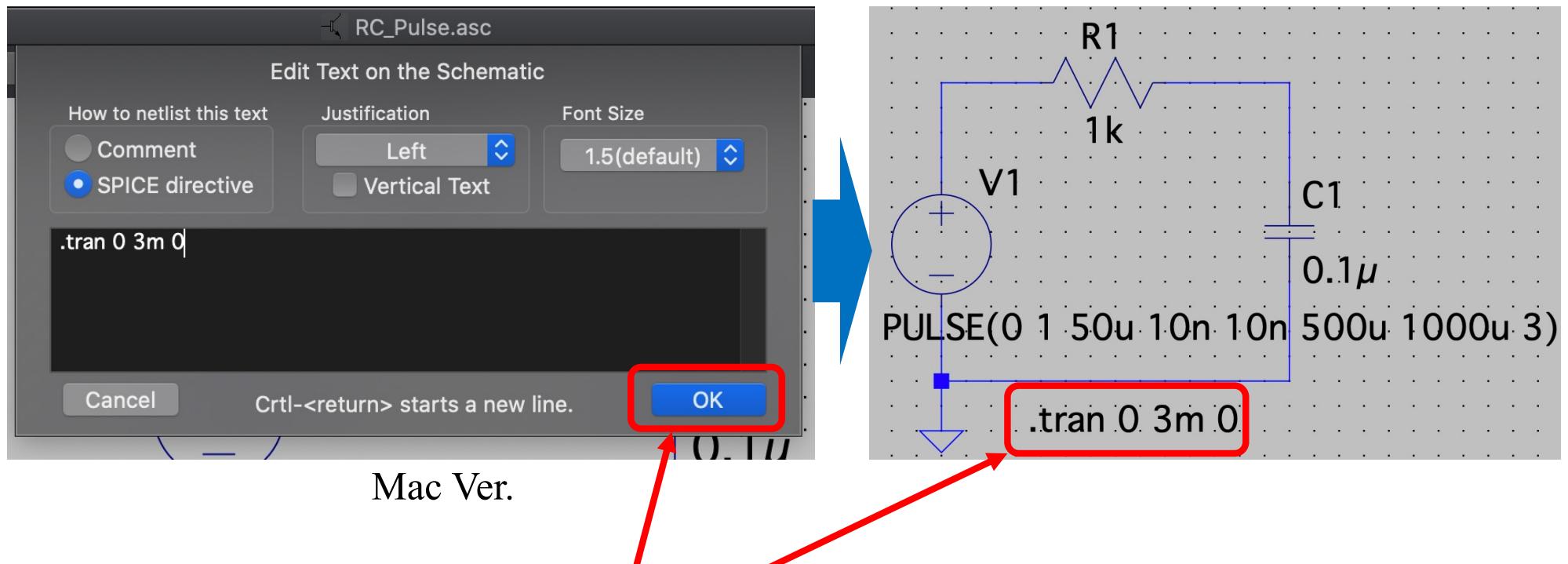
Mac Ver.

Make sure "**SPICE directive**" is selected and enter the command in the text box below.

* The same operation is used when entering text such as comments. You can enter comments by selecting "Comment" instead of "SPICE directive".

Entering “.tran” command (continued)

- In this practice, enter as follows,
.tran 0 3m 0

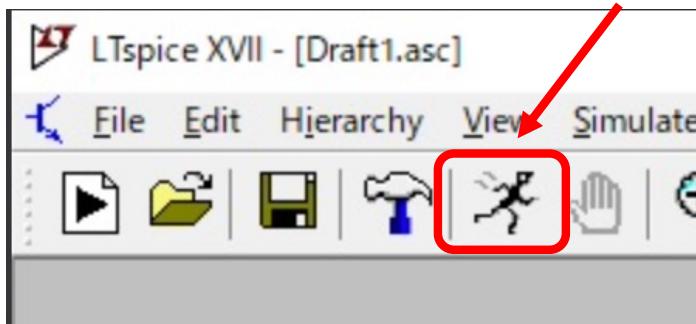


After confirming the input and pressing "OK", the menu disappears, and the entered command is displayed.

Since it can be moved with the mouse cursor, click it at an appropriate position to place it.

Execution of simulation

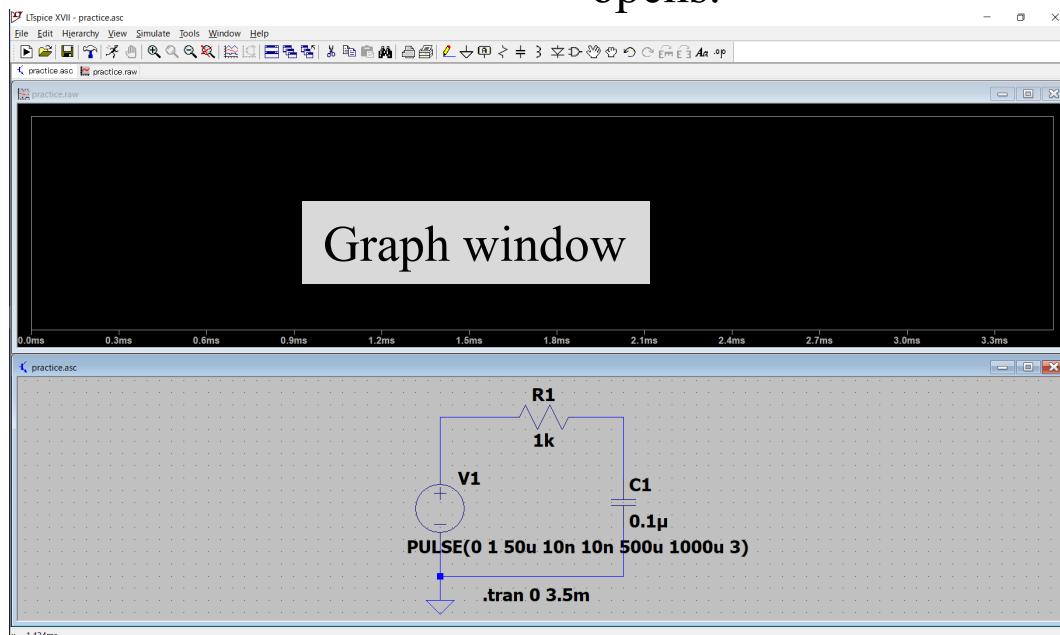
Just click this!!



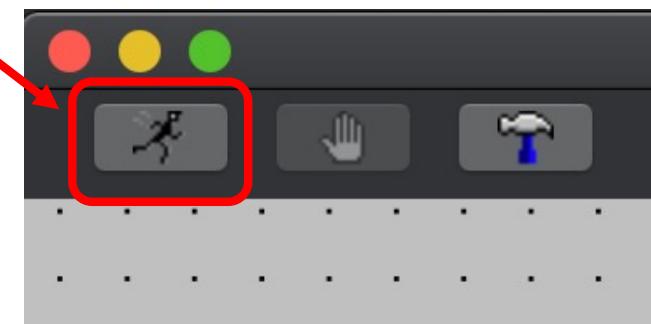
Windows Ver.



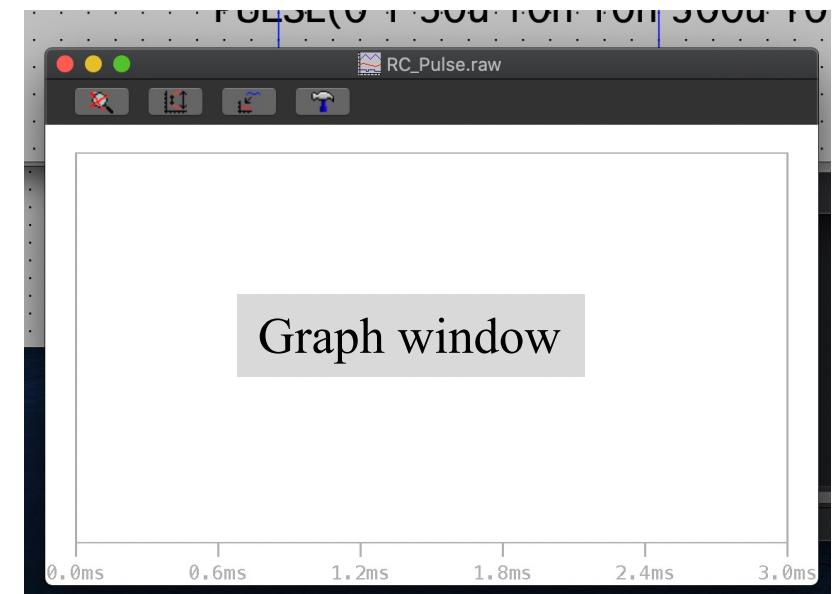
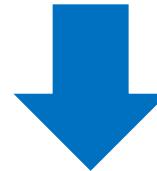
When the simulation is finished,
the graph window for displaying the result
opens.



Windows Ver.



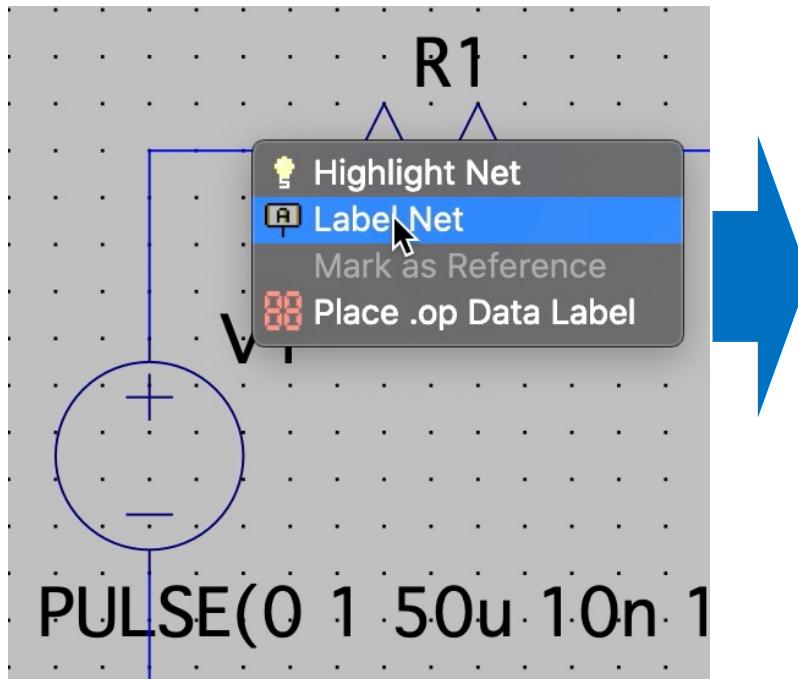
Mac Ver.



Mac Ver.

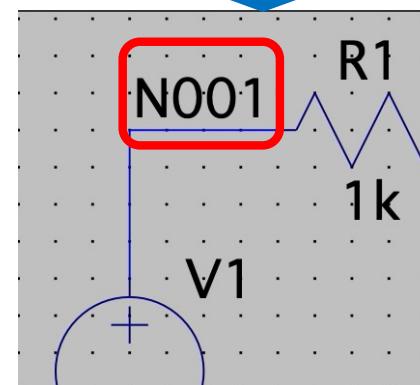
Display node's label

The node label is the name given to the node. Since the node voltage is specified by the name, the label of each node should be displayed after the simulation is completed.
* The node name can be changed.

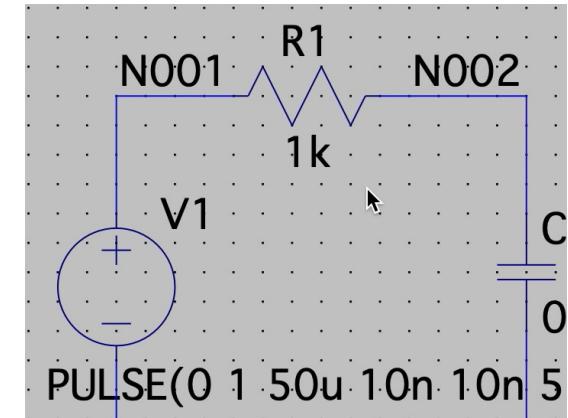


Press OK.
* If you want to change the name, new name is entered in text box.

After the simulation is completed, move the mouse cursor to any node and right-click. Select "Label Net" from the menu that appears.

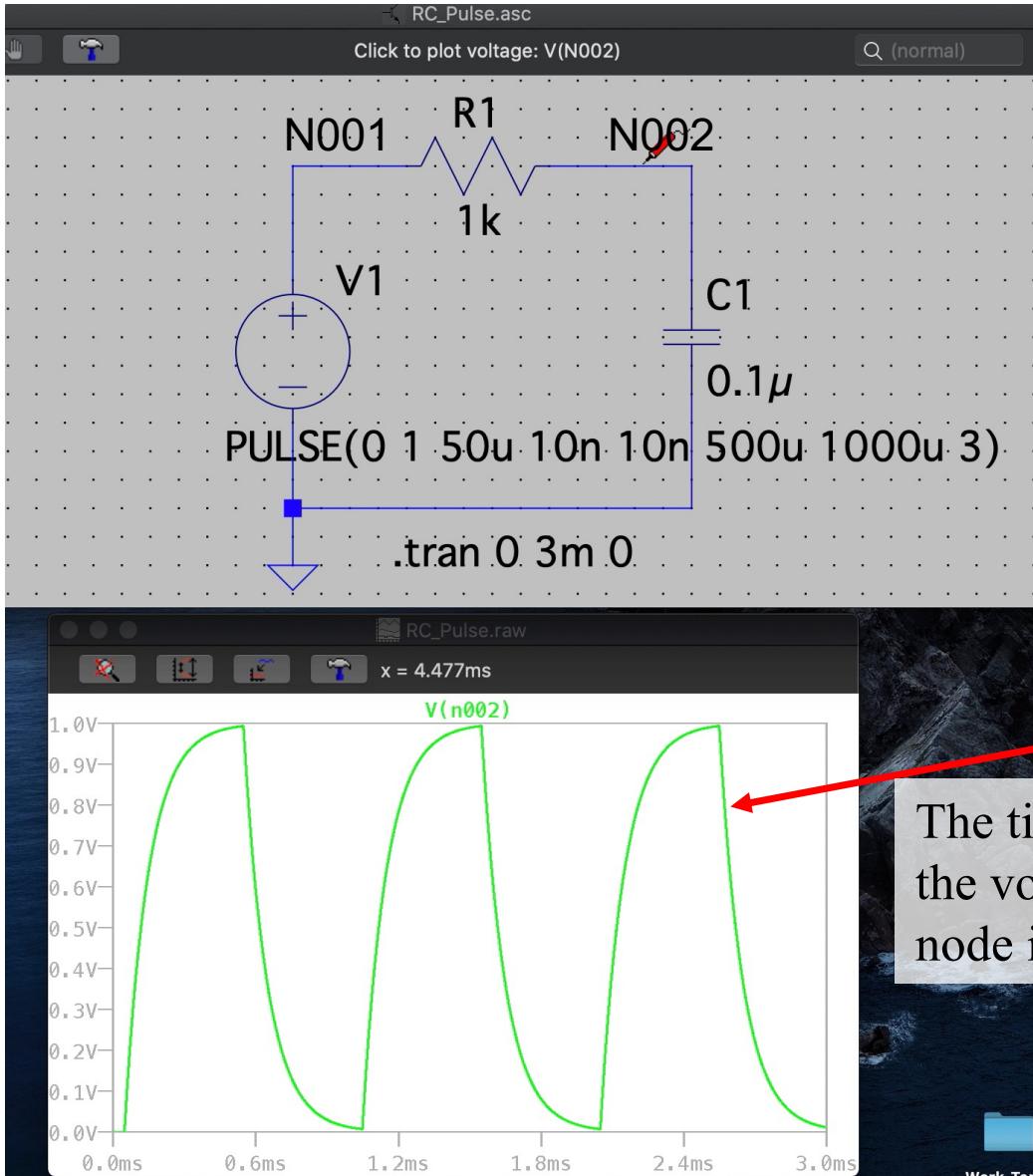


Node's label is displayed.

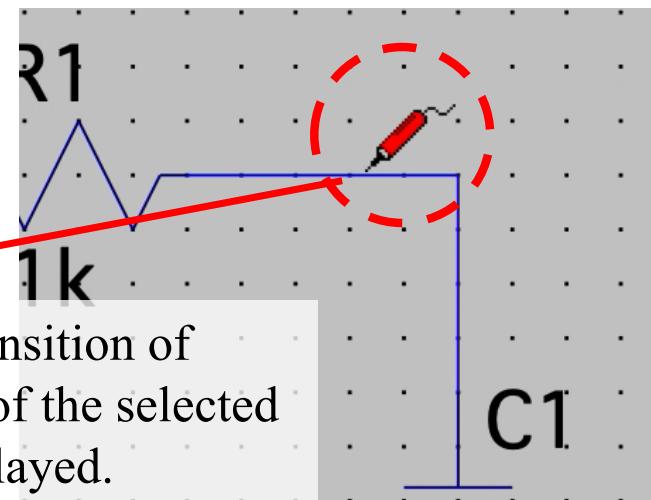


Display simulated result

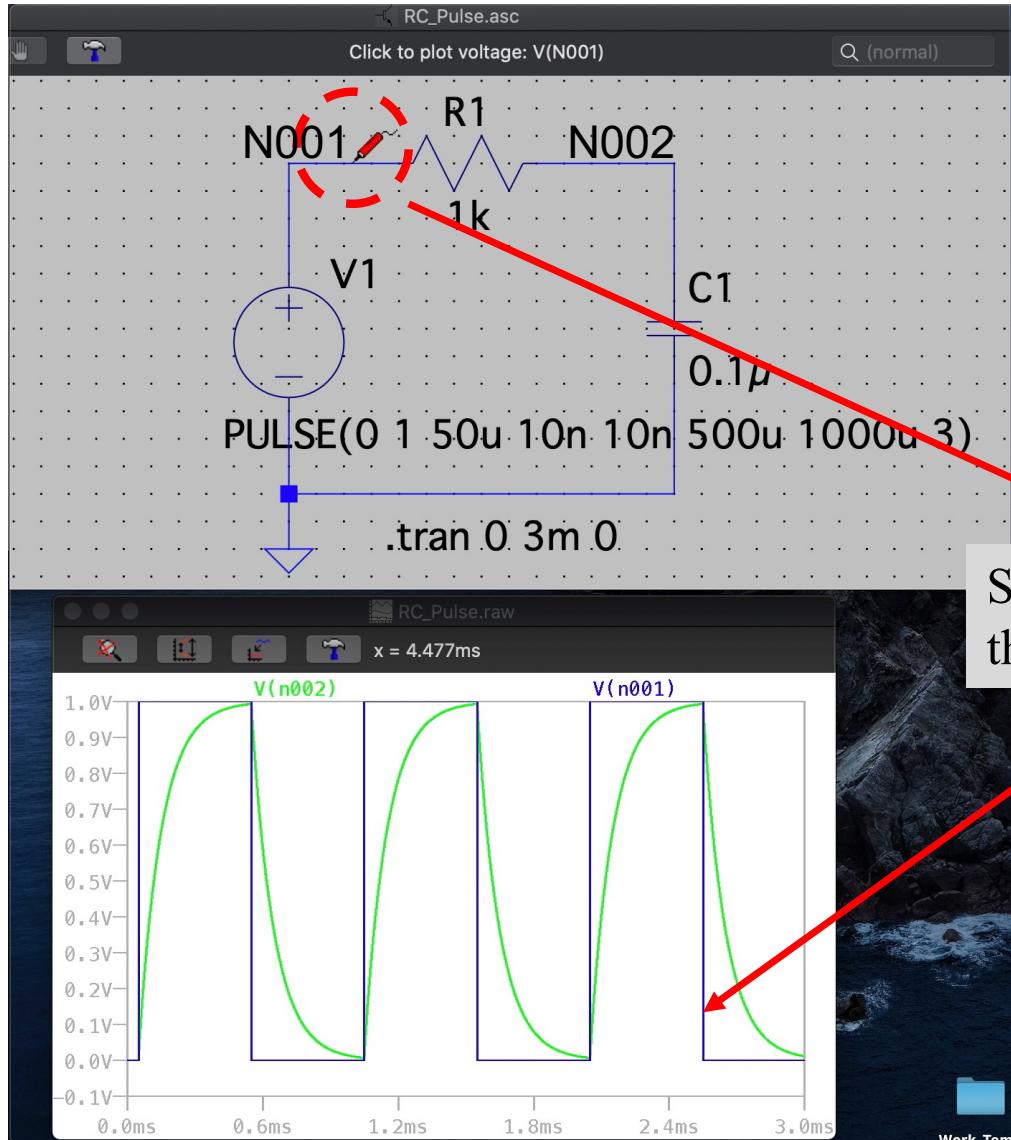
When the Transient analysis is completed, the voltage at each node of the circuit and the current flowing through each branch (each element) are calculated.



After the simulation, select the schematic screen and move the mouse cursor to any point to change the icon as shown in the figure below. In this state, click the node for which you want to check the analysis result, and the time transition of the voltage at that node will be displayed in the graph window.



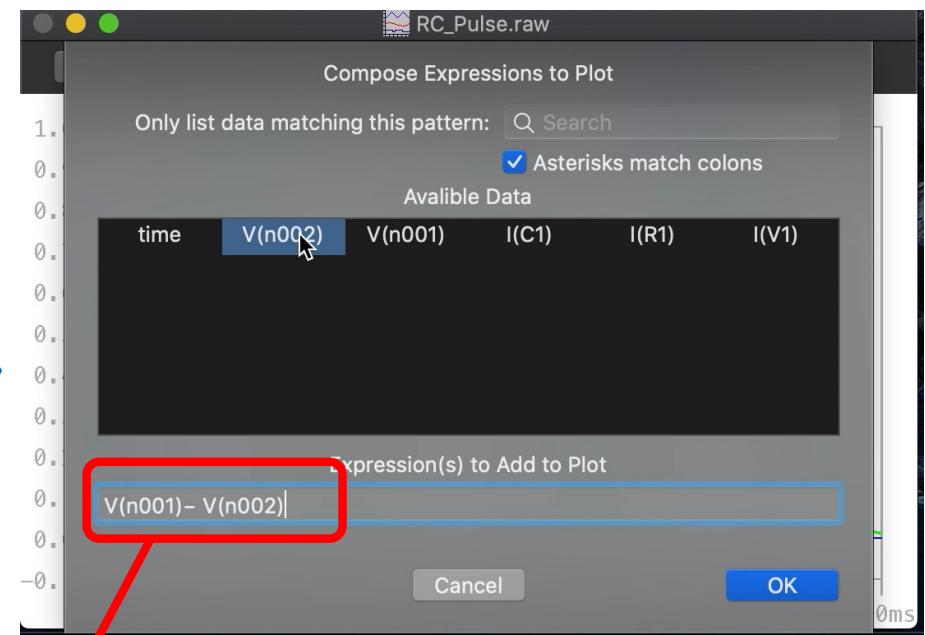
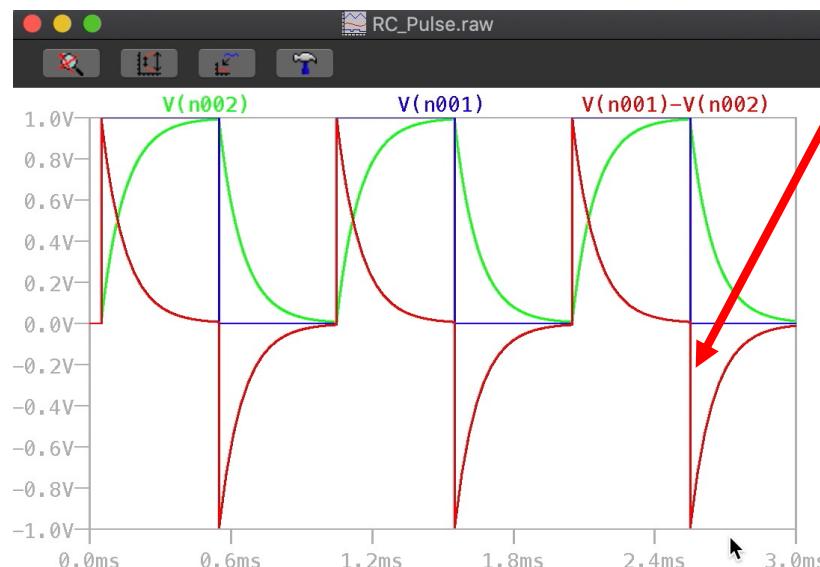
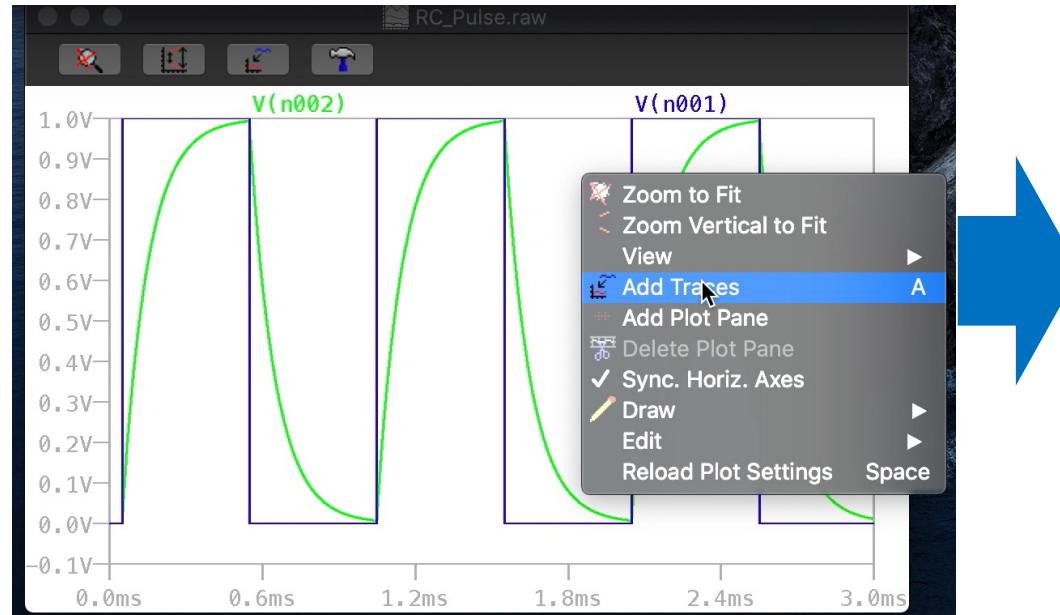
Additional display of simulated results



Selecting another node adds a plot of the simulation results for that node voltage.

It is also possible to calculate for simulation results

Right-click on the graph window to display the graph menu and select "Add Traces".

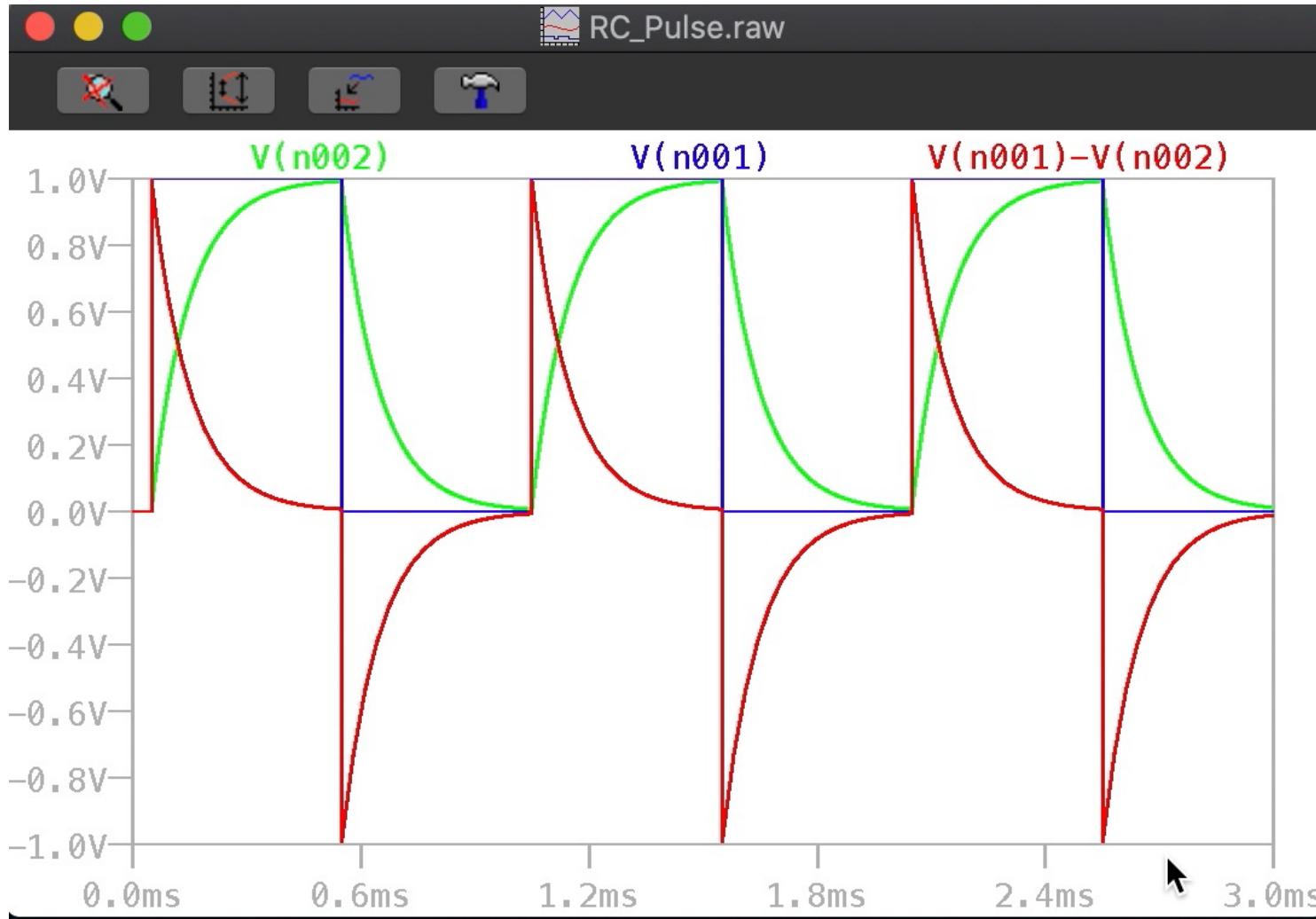


You can select the voltage or current you want to display from this window. Here, the calculation result can be displayed by setting the calculation formula in the text box such as taking the difference voltage between nodes.

In the above example,
The voltage $V(n002)$ at node n002 is
subtracted from the voltage $V(n001)$
at node n001.
⇒ Voltage between terminals of resistor R1

Simulated result for test circuit

Time series data at several node voltages obtained by Transient analysis



$V(n001)$: input signal

$V(n002)$: terminal voltage of C1

$V(n001) - V(n002)$: terminal voltage of R1

AC Analysis

AC analysis

- Analysis method to obtain the frequency characteristics of the circuit
- Simulate gain and phase for a signal of a certain frequency
- The amplitude and phase of the output signal is analyzed when a signal of each frequency is input to a circuit such as an amplifier or filter.
- Amplitude characteristics and phase characteristics are output.

Amplitude characteristics:

Horizontal axis is frequency, vertical axis is Amplitude.

* Sometimes input and output amplitude ratio.

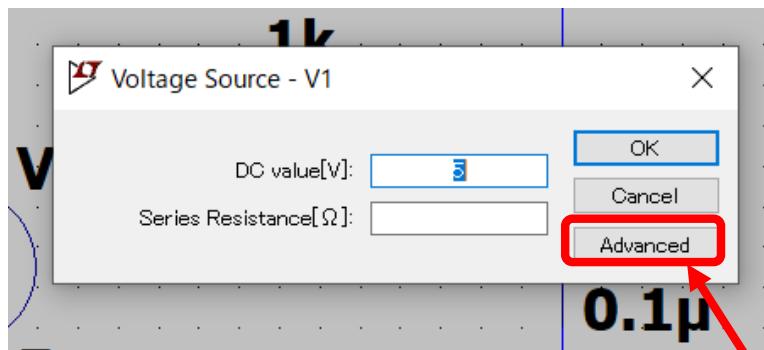
Phase characteristics:

Horizontal axis is frequency, vertical axis is phase difference between input signal and output signal.

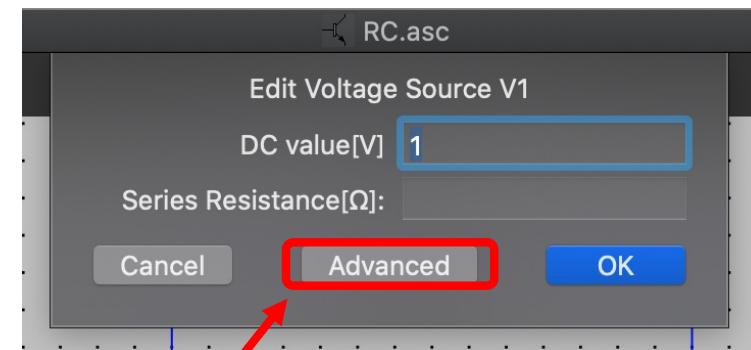
Change pulse voltage source to sinusoidal voltage source

- * The same circuit used in the practice of transient analysis is also used in AC analysis.
- * However, since the simulation contents are different, it is better to save it under a different folder.

- In AC analysis, a signal is input to the circuit while changing the frequency of the sine wave, and the output for that signal is analyzed. Therefore, it is necessary to use a sinusoidal voltage source as the signal source.
- When you move the mouse cursor to the voltage source symbol, the cursor changes to the shape of a pointing finger, so right-click in this state.
- After that, the following menu will appear. Then, select "Advanced" to proceed to the detailed setting window.



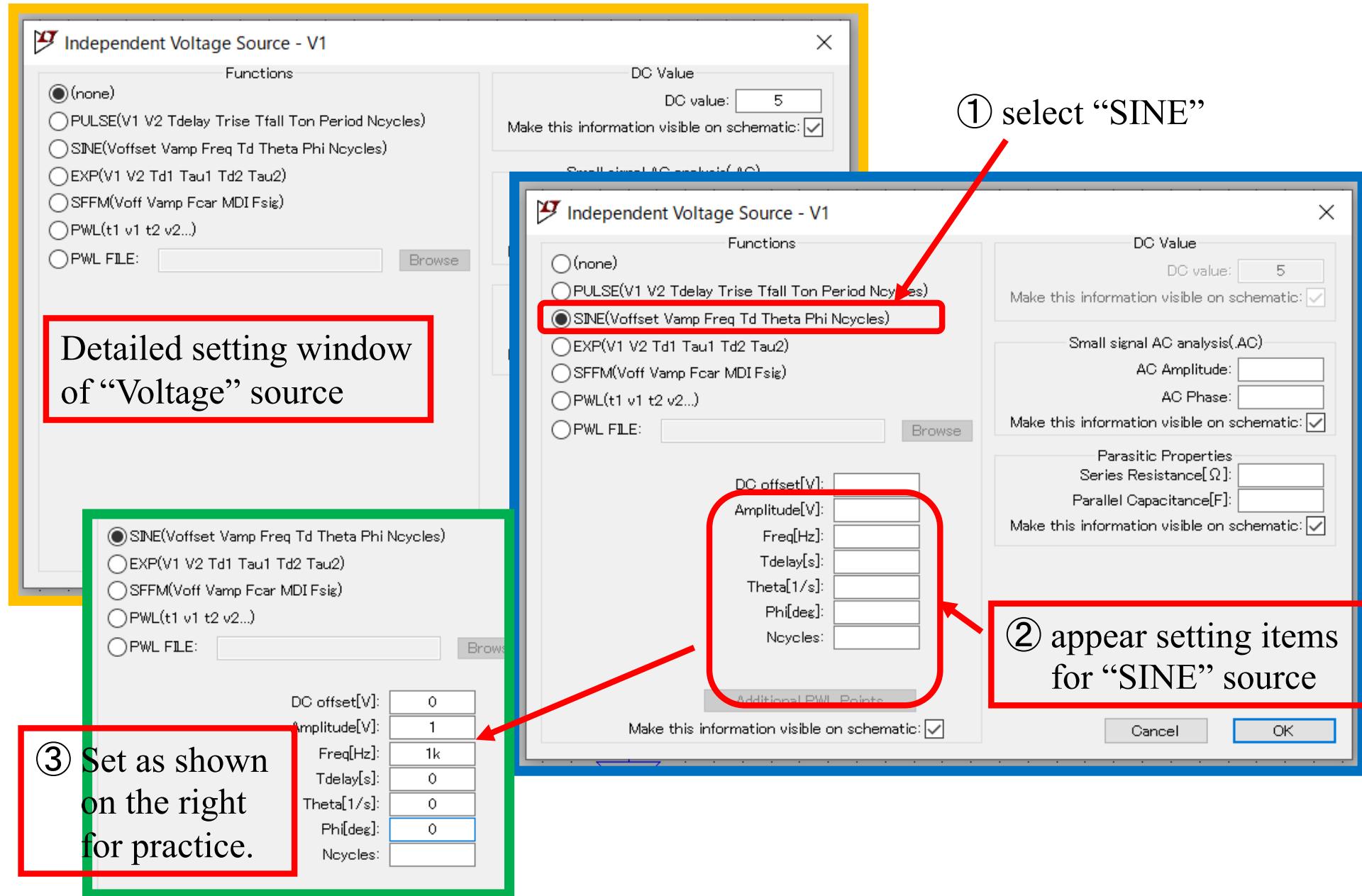
Windows Ver.



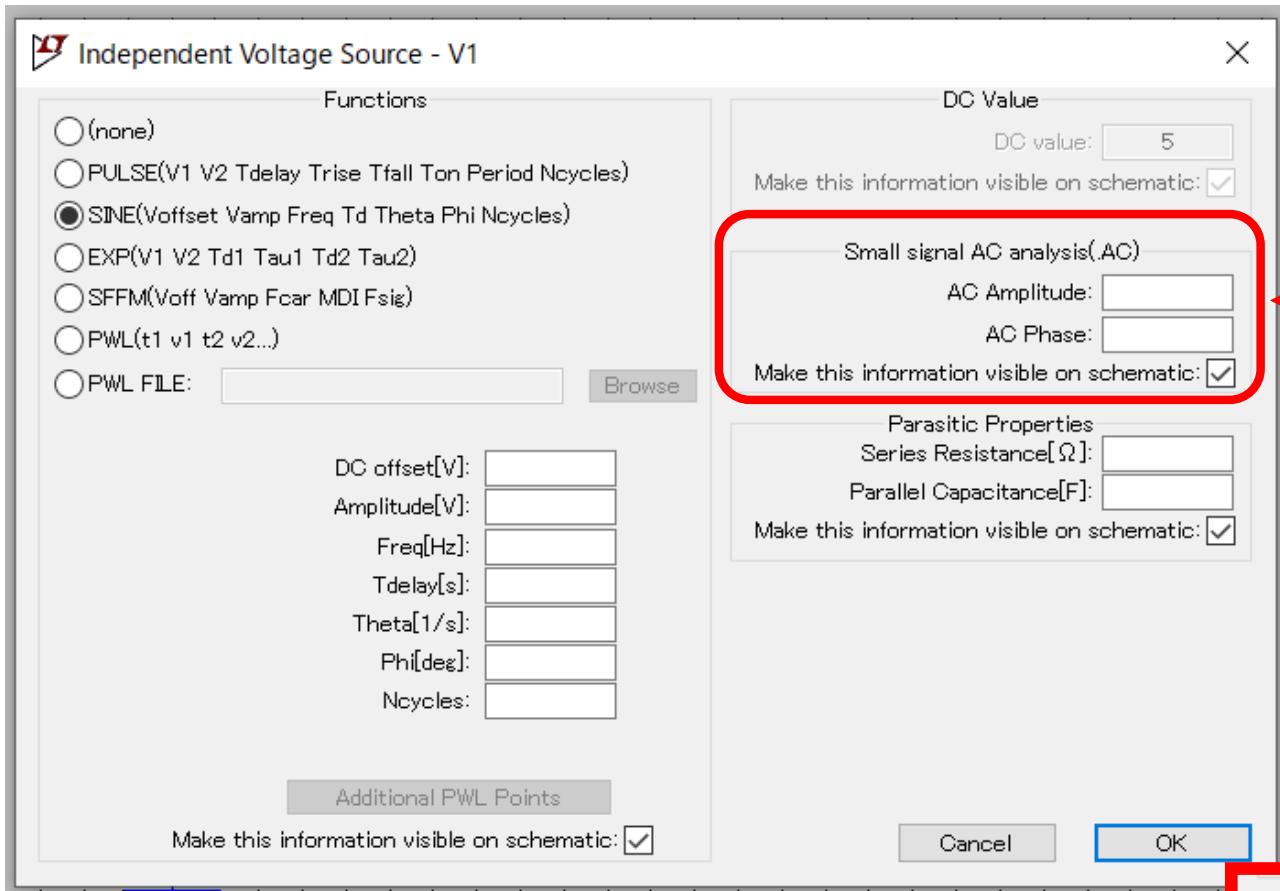
Mac Ver.

Press “Advanced” button and go to detailed setting window

Detailed setting of “Voltage” source (Windows)



Signal source setting for AC analysis (Windows)

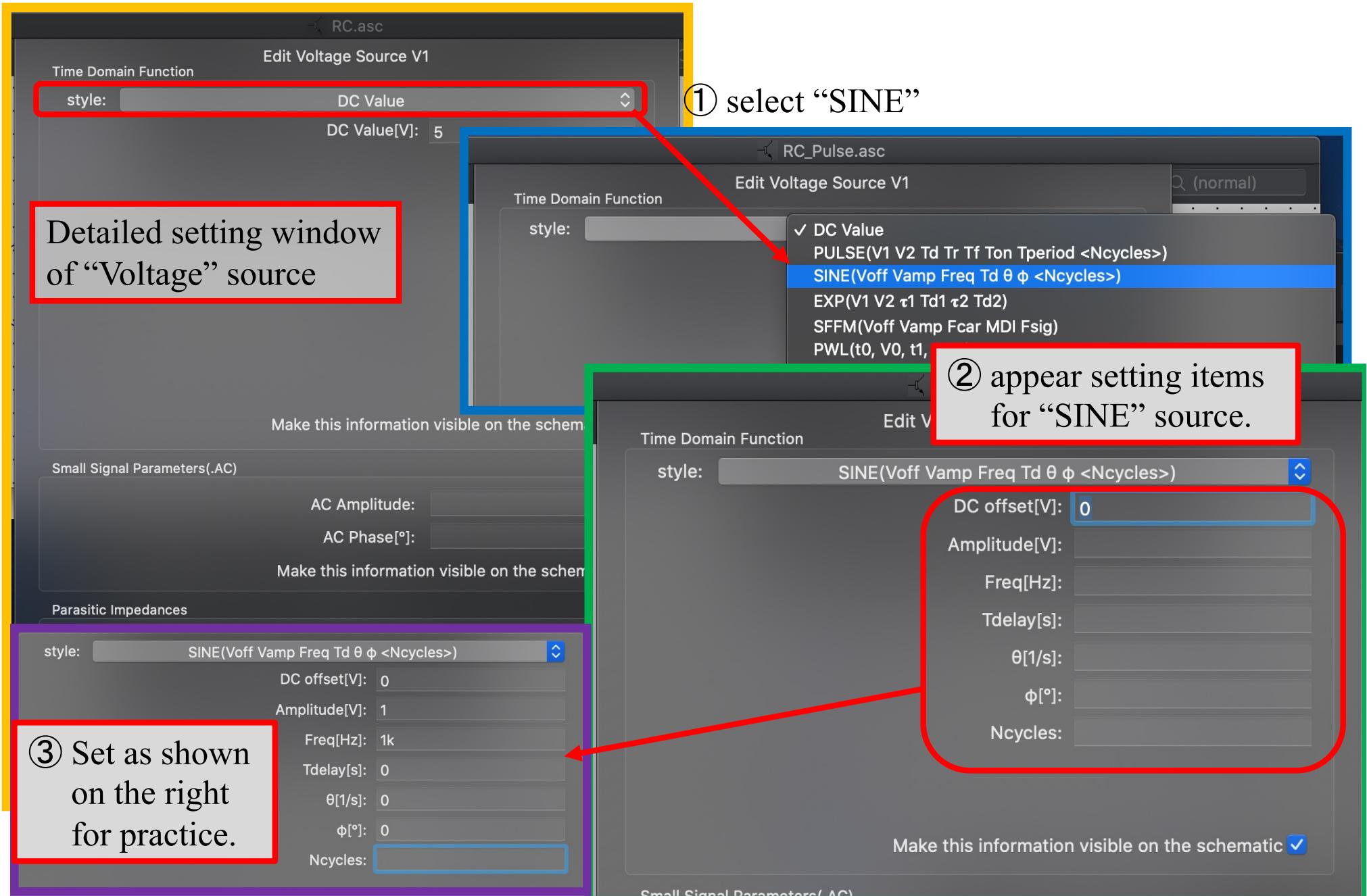


Items that need to be set for the sinusoidal voltage source when performing AC analysis. With this setting, the following conditions for the input signal during AC analysis are set.

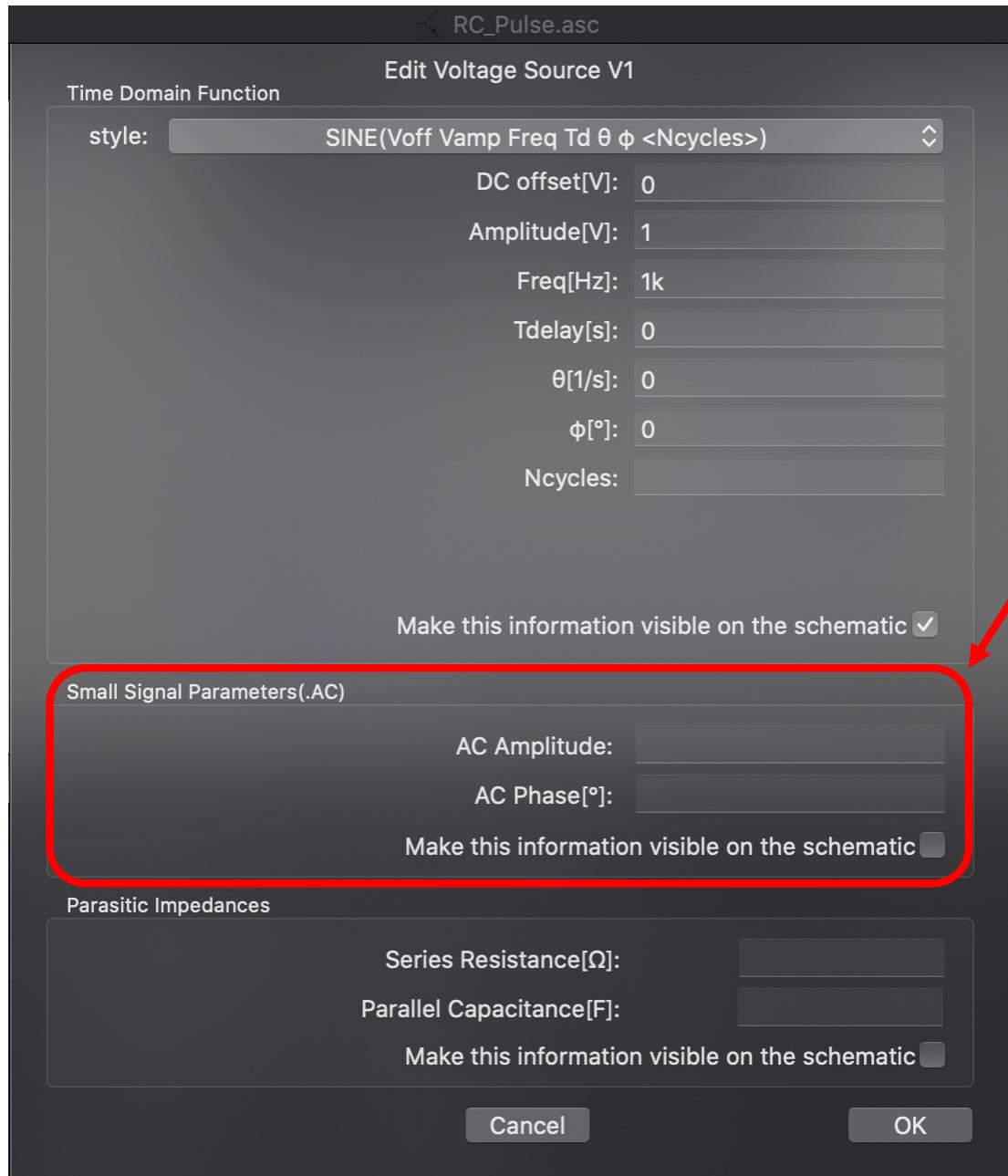
- Signal amplitude
- Signal phase

In this practice, set as follows.

Detailed setting of “Voltage” source (Mac)



Signal source setting for AC analysis (Mac)



Items that need to be set for the sinusoidal voltage source when performing AC analysis. With this setting, the following conditions for the input signal during AC analysis are set.

- Signal amplitude
- Signal phase

In this practice, set as follows.

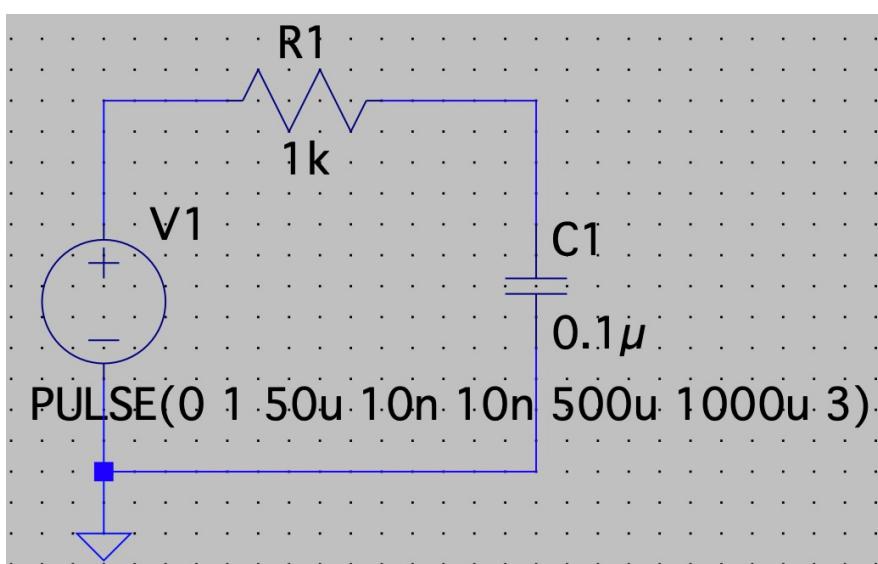
AC Amplitude: 1

AC Phase[°]: 0

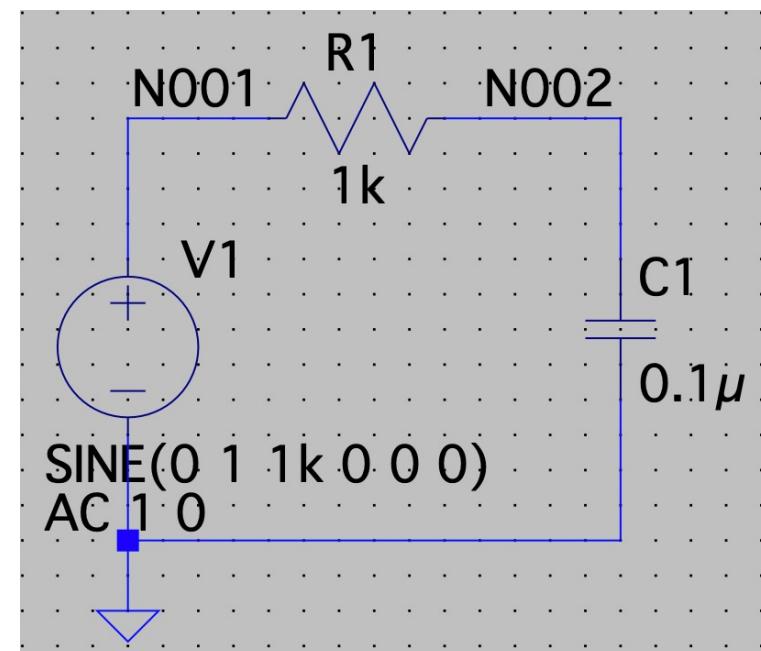
Make this information visible on the schematic

OK

Comparison between before and after setting of "Voltage" source



Before setting



After setting

Description format of “SINE” source

SINE(Voff Vamp Freq Td θ Φ <Ncycles>)

V_{off} (DC offset[V]): Off-set voltage * DC voltage which is included a signal.

V_{amp} (Amplitude[V]): Amplitude of sinusoidal wave

$Freq$ (Freq[Hz]): Frequency of signal

Td (Tdelay[s]): Delay time to input signal to circuit from the start of simulation

θ (θ[1/s]): Coefficient for attenuation over time (Damping coefficient)

Φ (Φ[°]): Initial phase

Ncycles: The number of repetitions of cycle

SPICE command for AC analysis

- As with Transient analysis, enter the appropriate SPICE command on the schematic screen for AC analysis.
- The command for AC analysis is ".ac".
- Set the ".ac" command and the parameters required for analysis.
- The format of the ".ac" command is as follows.

```
.ac Frequency sweep interval setting Number of analysis points Start Freq End Freq
```

Frequency sweep interval setting: Select one of “oct” for Octave, “dec” for Decade, “lin” for Liner

Number of analysis points: The number of analysis points per interval

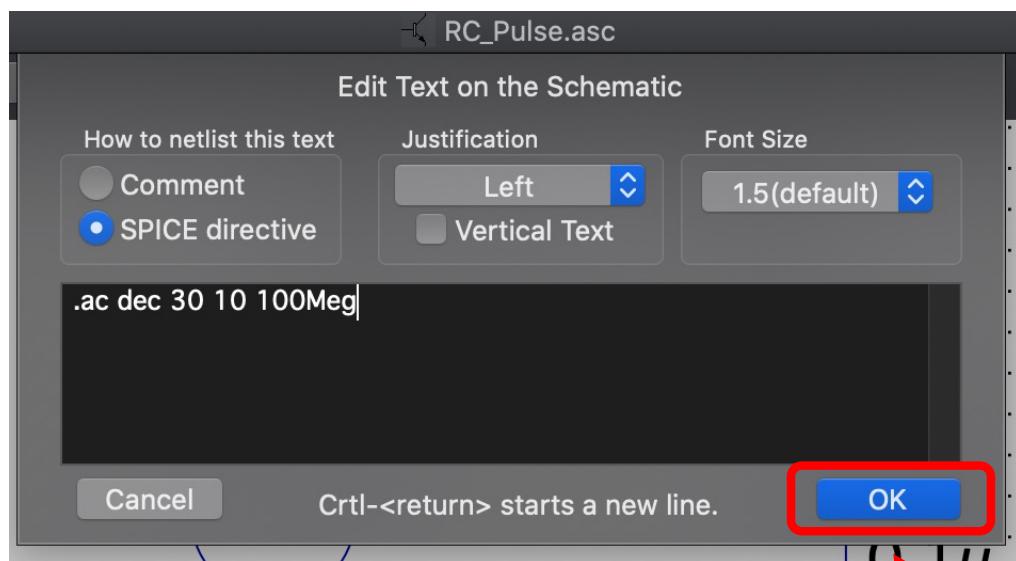
Start Freq: The lower side of the frequency range to be analyzed

End Freq: The higher side of the frequency range to be analyzed

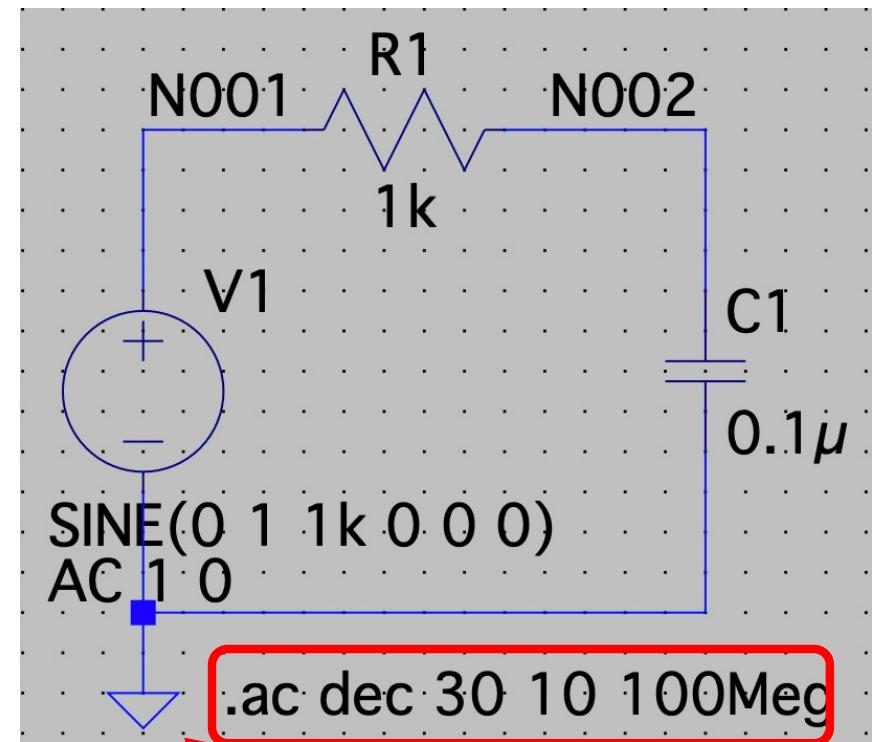
Entering “.ac” command

- In this practice, enter as follows,

.ac dec 30 10 100Meg



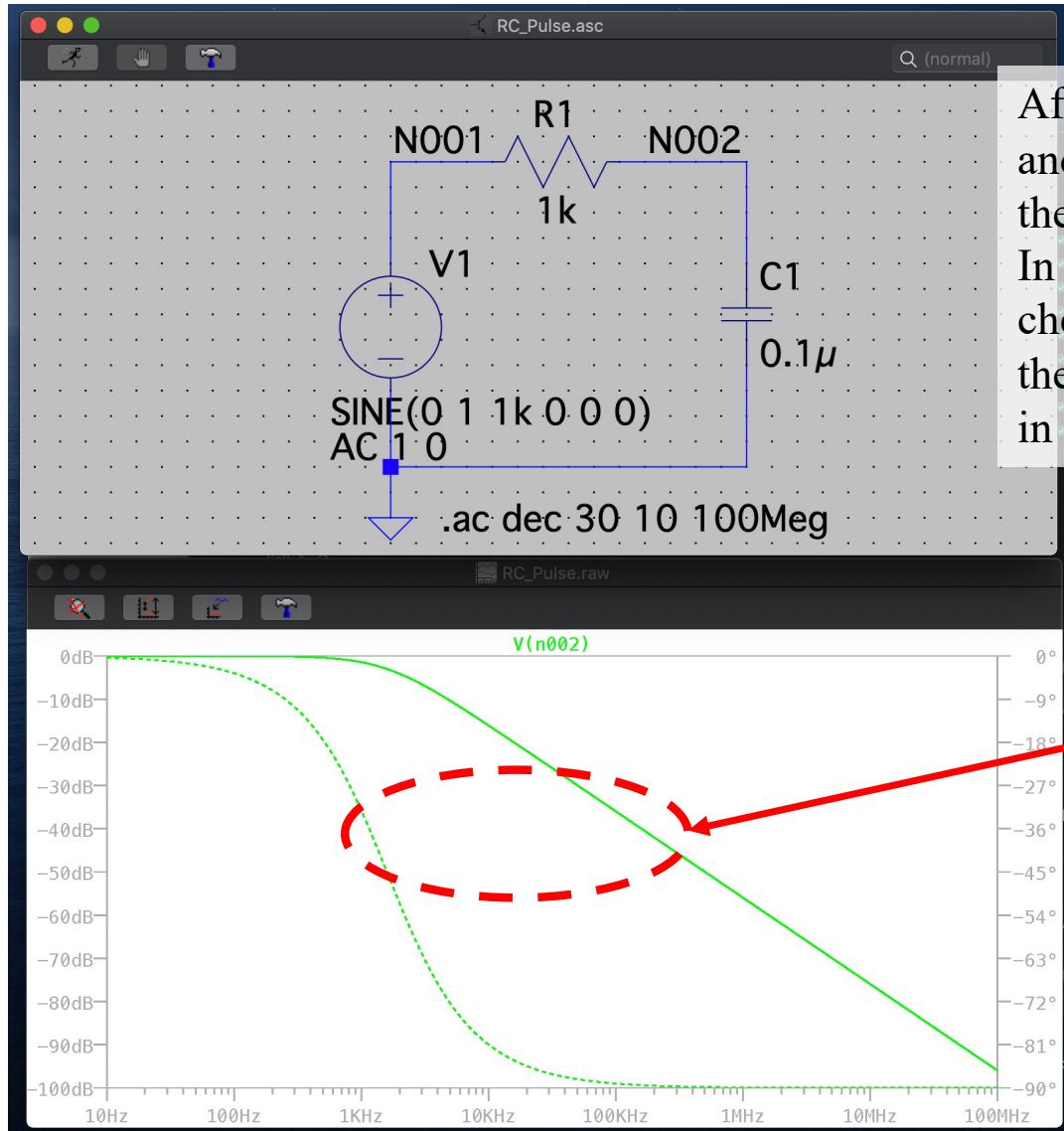
Mac Ver.



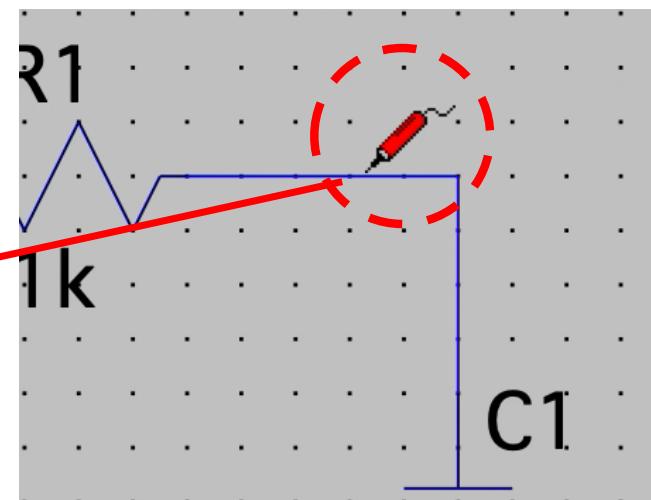
- After confirming the input and pressing "OK", the menu disappears, and the entered text character can be moved with the mouse cursor.
- Click it at an appropriate position to place it.
- After placing the command, press Run to execute. When the simulation is finished, the graph window opens.

Display simulated result

When AC analysis is performed, the frequency characteristics of the voltage at each node of the circuit are calculated as the result of comparison with the input signal.

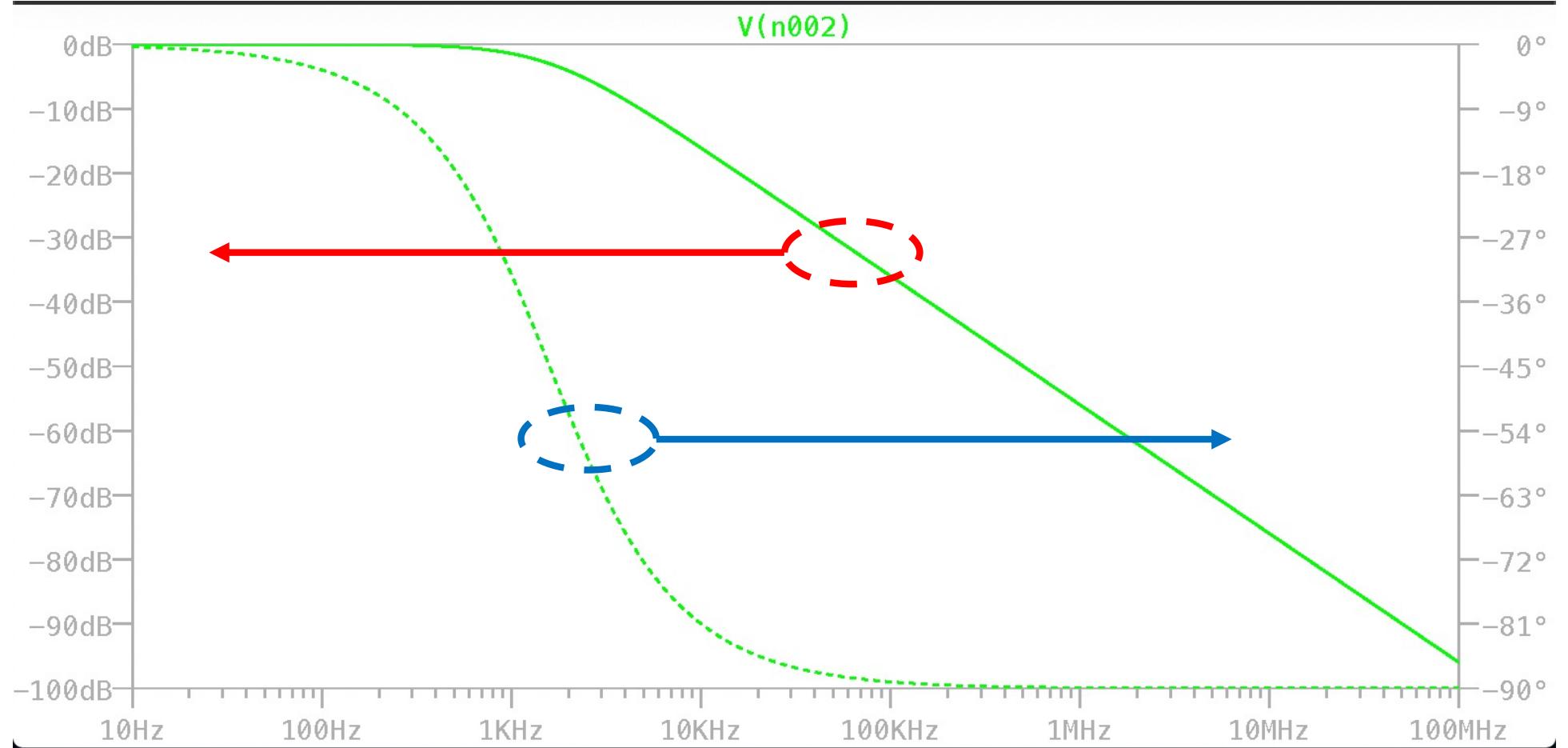


After the simulation, select the schematic screen and move the mouse cursor to any point to change the icon as shown in the figure below. In this state, click the node for which you want to check the analysis result, and the time transition of the voltage at that node will be displayed in the graph window.



Simulated result for test circuit

Frequency characteristics of the node “N002” obtained by AC analysis



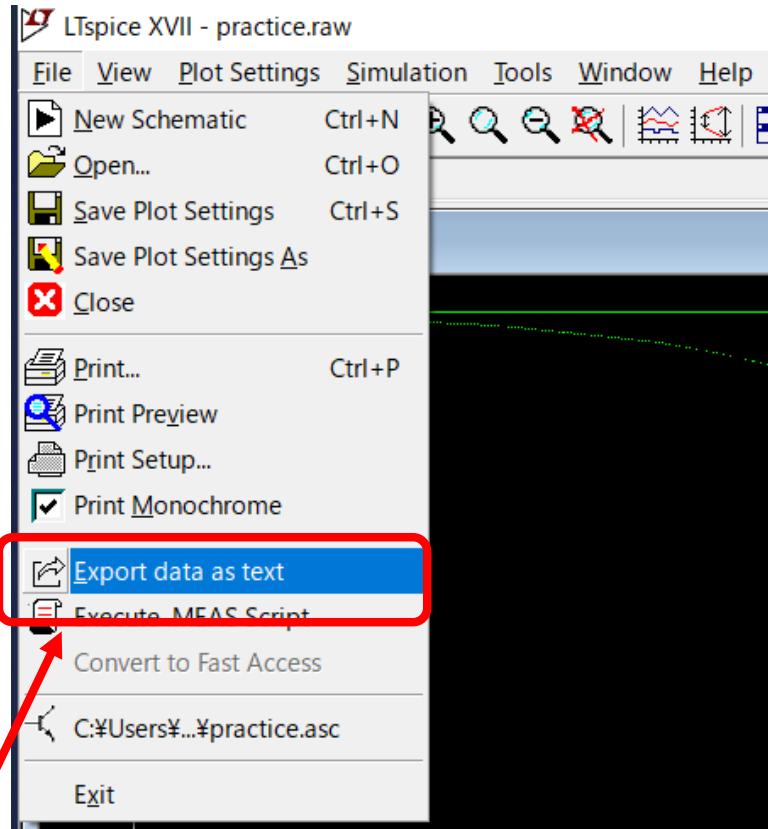
Horizontal axis: Frequency

Vertical axis: Left -> Amplitude ratio of input and output signal in dB

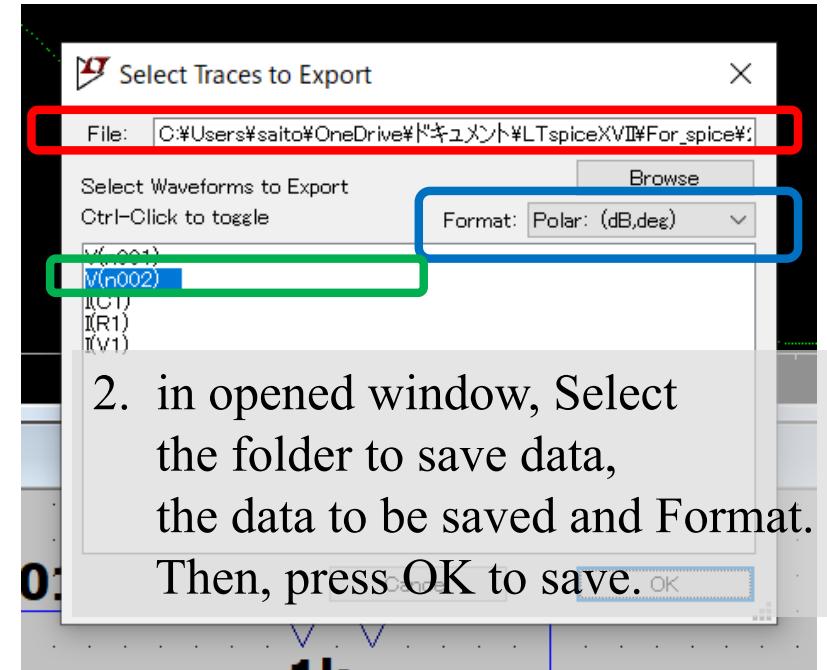
Right -> Phase difference of output signal against input signal

Saving simulated data

Saving simulated data as text file (Win)



1. select “Export data as text” from “file” menu.



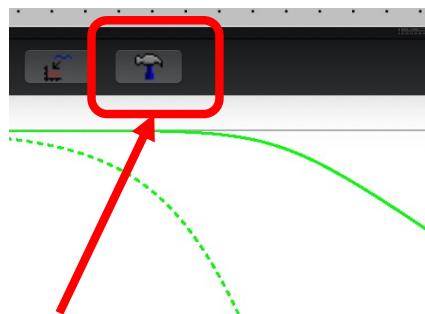
2. in opened window, Select the folder to save data, the data to be saved and Format. Then, press OK to save.

The graph data file to be saved is a text file with the extension ".txt". It can be read by Excel etc.

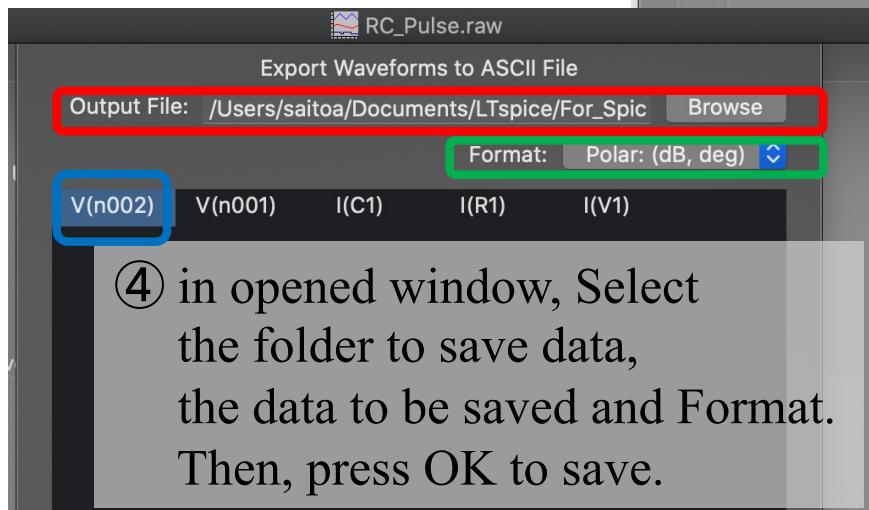
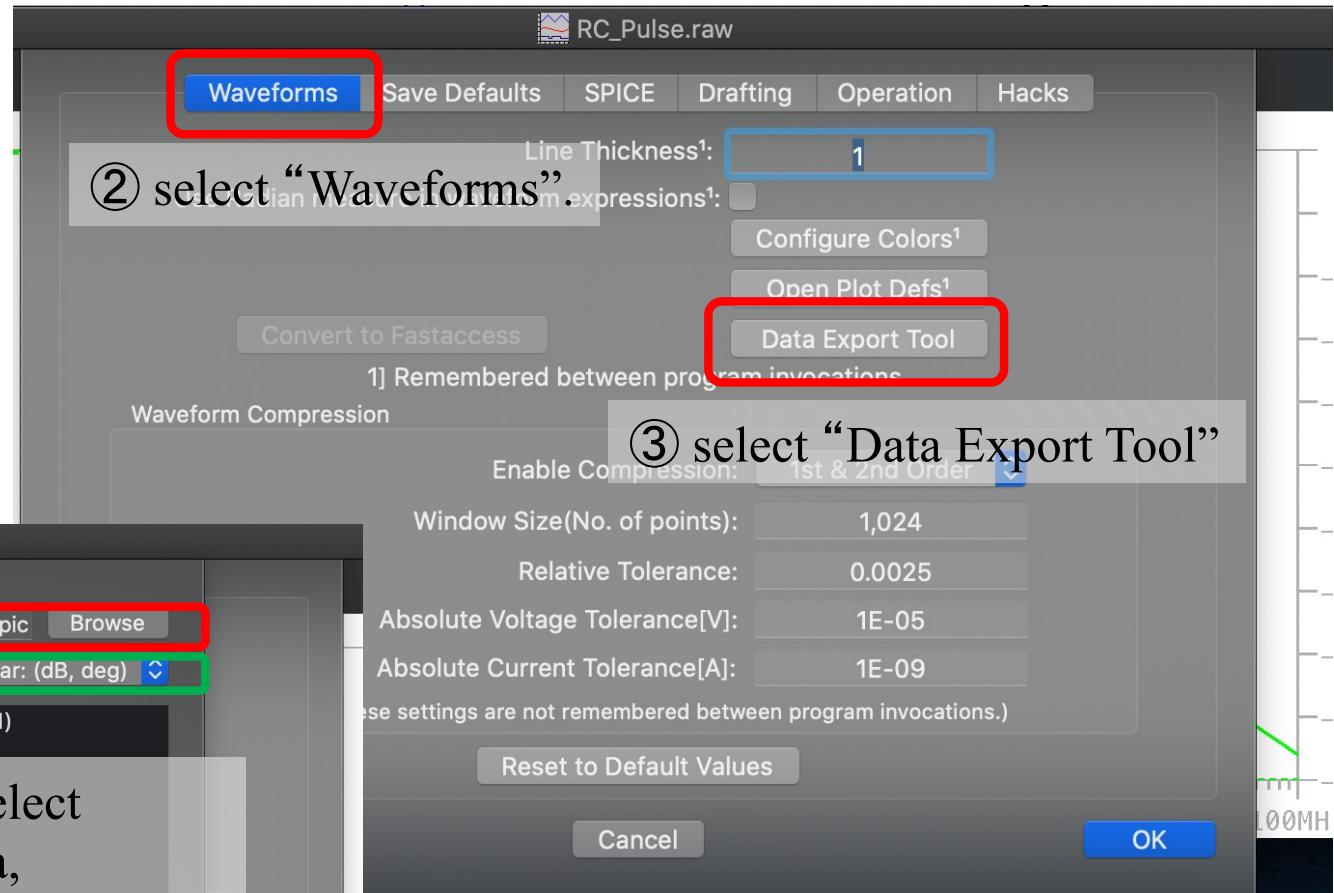
【Important!!】

In the case of AC analysis data, select “Cartesian: re, im” as data format. Not “Polar: (dB, deg)”

Saving simulated data as text file (Mac)



- ① Press the hammer-shaped icon in the graph window.



- ④ in opened window, Select the folder to save data, the data to be saved and Format. Then, press OK to save.



The graph data file to be saved is a text file with the extension ".txt". It can be read by Excel etc.