

# Installation of LTspice and Practice to draw simple circuit schematic

“LTspice” is a circuit simulator.

And it's free software and a very good simulator.

Please refer to this document to install LTspice on your PC.

- For Windows (Slide No.2 – 11)
- For Mac (Slide No.12 – 22)
- Practice for drawing circuit schematics on LTspice (Slide No.23 - )

For Windows

# Download of LTspice

- Please access below.  
<https://www.analog.com/en/resources/design-tools-and-calculators/ltpice-simulator.html>

Amplifier & Linear

Clock & Timing

Data Converter

EE-Sim

**LTspice**

Power Management

RF & Synthesis

Cybersecurity

## LTspice

**Fast • Free • Unlimited**

LTspice® is a powerful, fast, and free SPICE simulator software, schematic capture and waveform viewer with enhancements and models for improving the simulation of analog circuits. Its graphical schematic capture interface allows you to probe schematics and produce simulation results, which can be explored further through the built-in waveform viewer.

Learn how to use LTspice with [our tutorials below](#) or dive deeper with our selection of helpful tips and articles. You can also browse our library of macromodels and demo circuits for select Analog Devices products.

LTspice's enhancements and models improve the simulation of analog circuits when compared to other SPICE solutions. Download LTspice below to see for yourself!

**Download LTspice**

Download our LTspice simulation software for the following operating systems:

Date models updated - Jun 17 2024

[Download for Windows 10 64-bit and forward](#) Version 24.0.12

[Download for MacOS 10.15 and forward](#) Version 17.2.4

[Download for Windows XP](#) (End of Support)

[Download for MacOS 10.9](#) (End of Support)

[Download LTspice XVII for Windows](#) (End of Support)

**Download the file that suits your PC environment.**

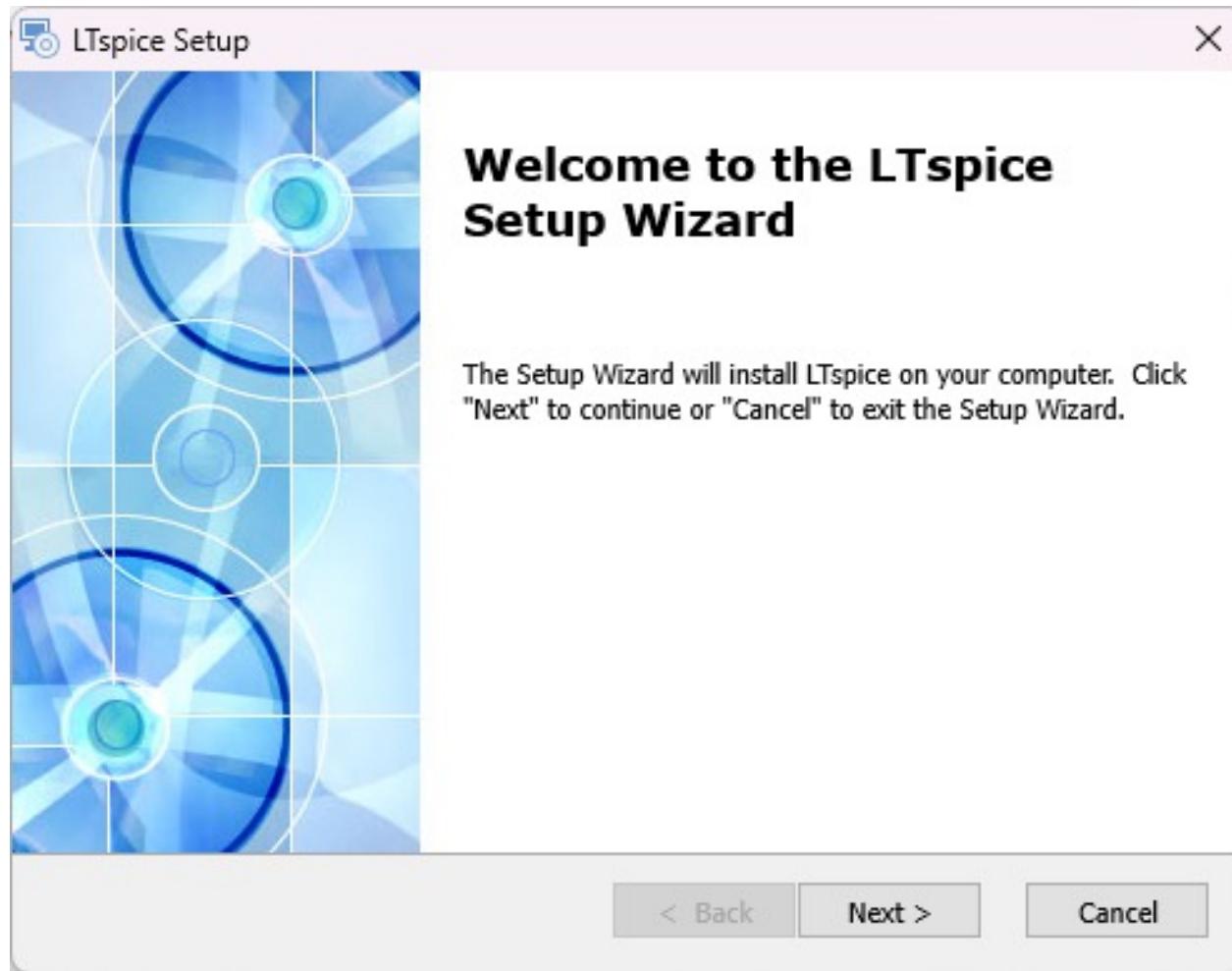
**In the case of Windows,**

**File name: LTspice64.msi**

**After downloading the installation file, double click the file to start to install.**

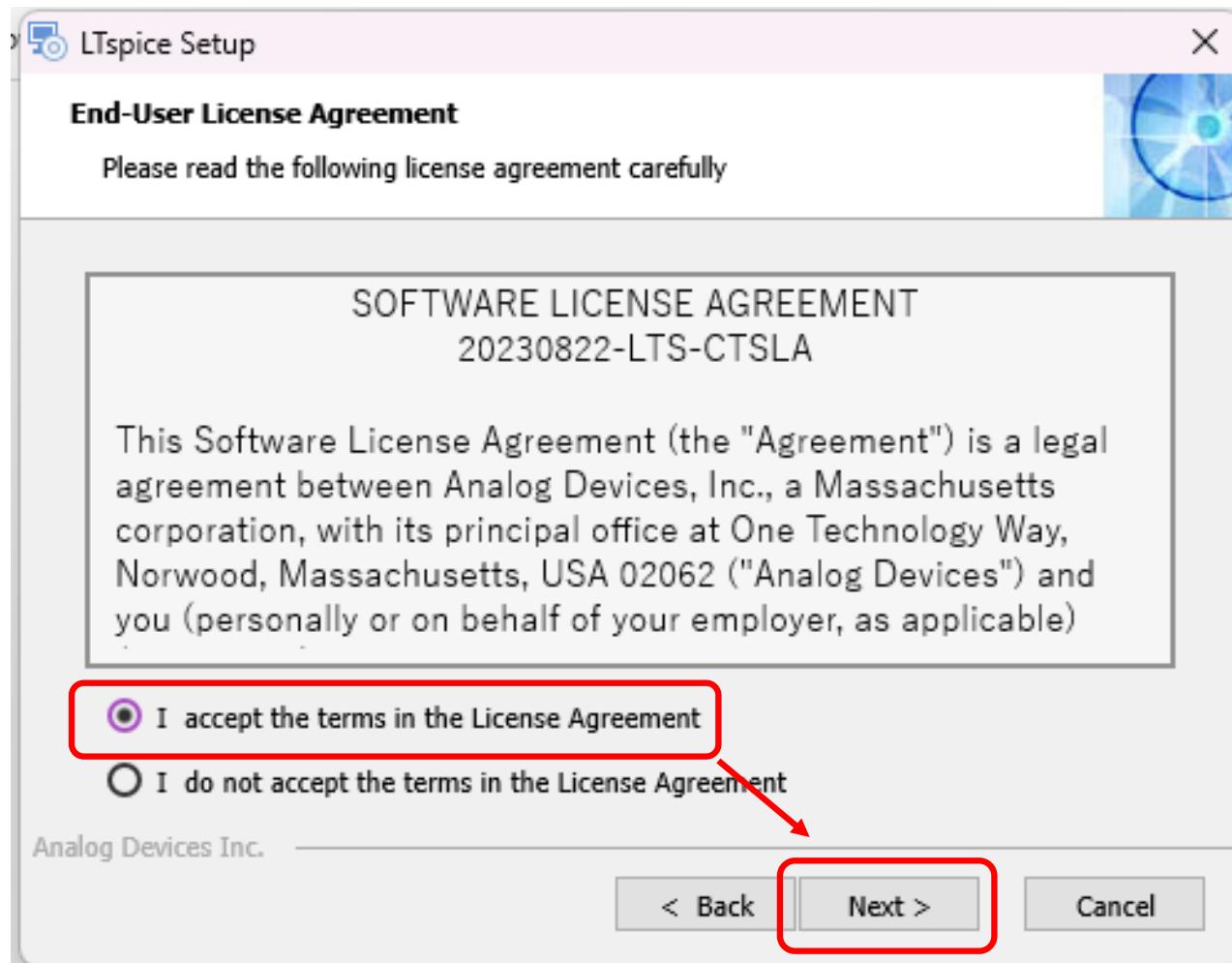
# Starting LTspice Setup Wizard

After launching installer, “LTspice Setup Wizard” window will be opened.  
Please select “Next”.



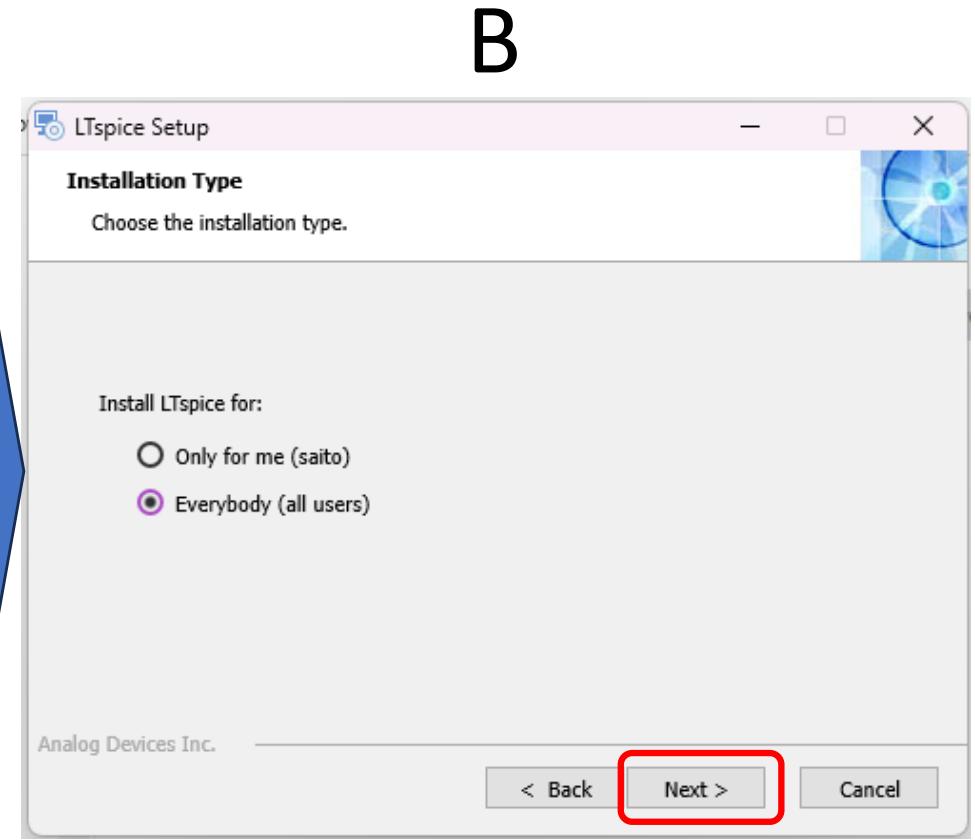
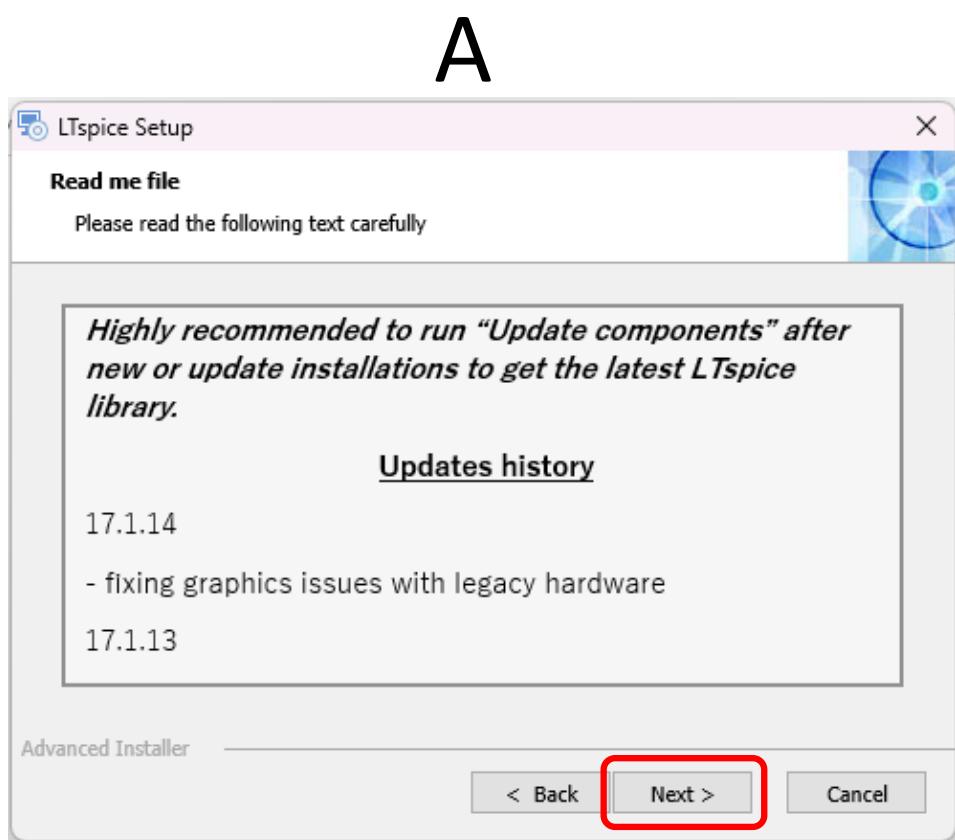
# License Agreement

After launching installer, open dialog about License Agreement.  
Please select “I accept...” and “Next” for progress.



# Continuation of Setup

- A: After “License Abreemnt”, “Read me file” dialog will open, please confirm it and click “Next” to proceed.
- B: You will be asked for the “Installation Type”, please choose the one you like and click “Next” to proceed.



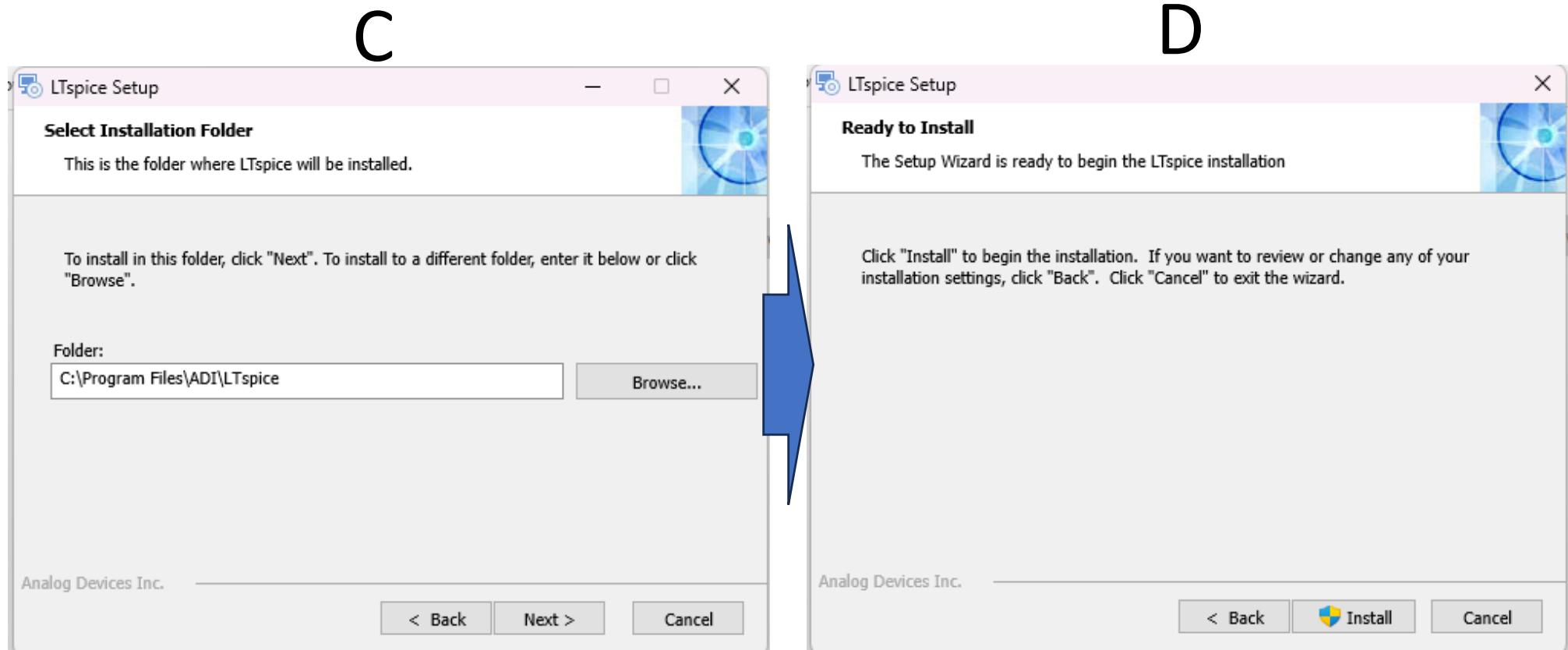
# Selection of folder for installation of LTspice program => Starting installation

C: You will be asked for the installation folder, so click “Next” to proceed.

- \* Basically, the default folder is sufficient.

- \* Please set the installation destination as necessary.

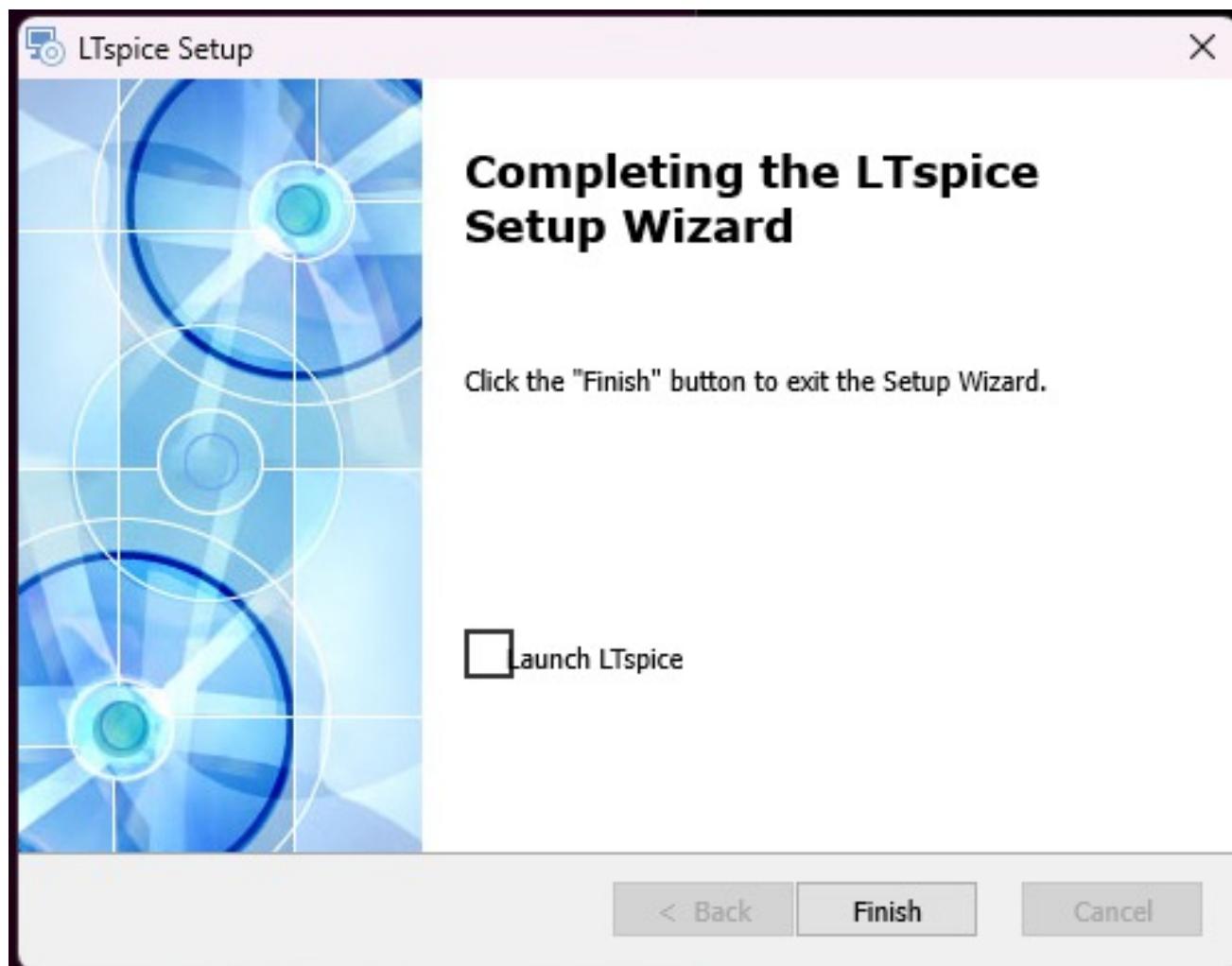
D: The installation is ready, so press “Install” to start the installation.



\* After D, you may be asked “Do you want to allow apps to make changes to your device?”. Then, select “Yes” to proceed with the installation.

# Finish the installation

The following screen will be appeared, and the installation will be completed.  
Press “Finish” to finish the installation.  
If you check “Launch LTspice” check box, LTspice software will be lunched.



# Installation complete!

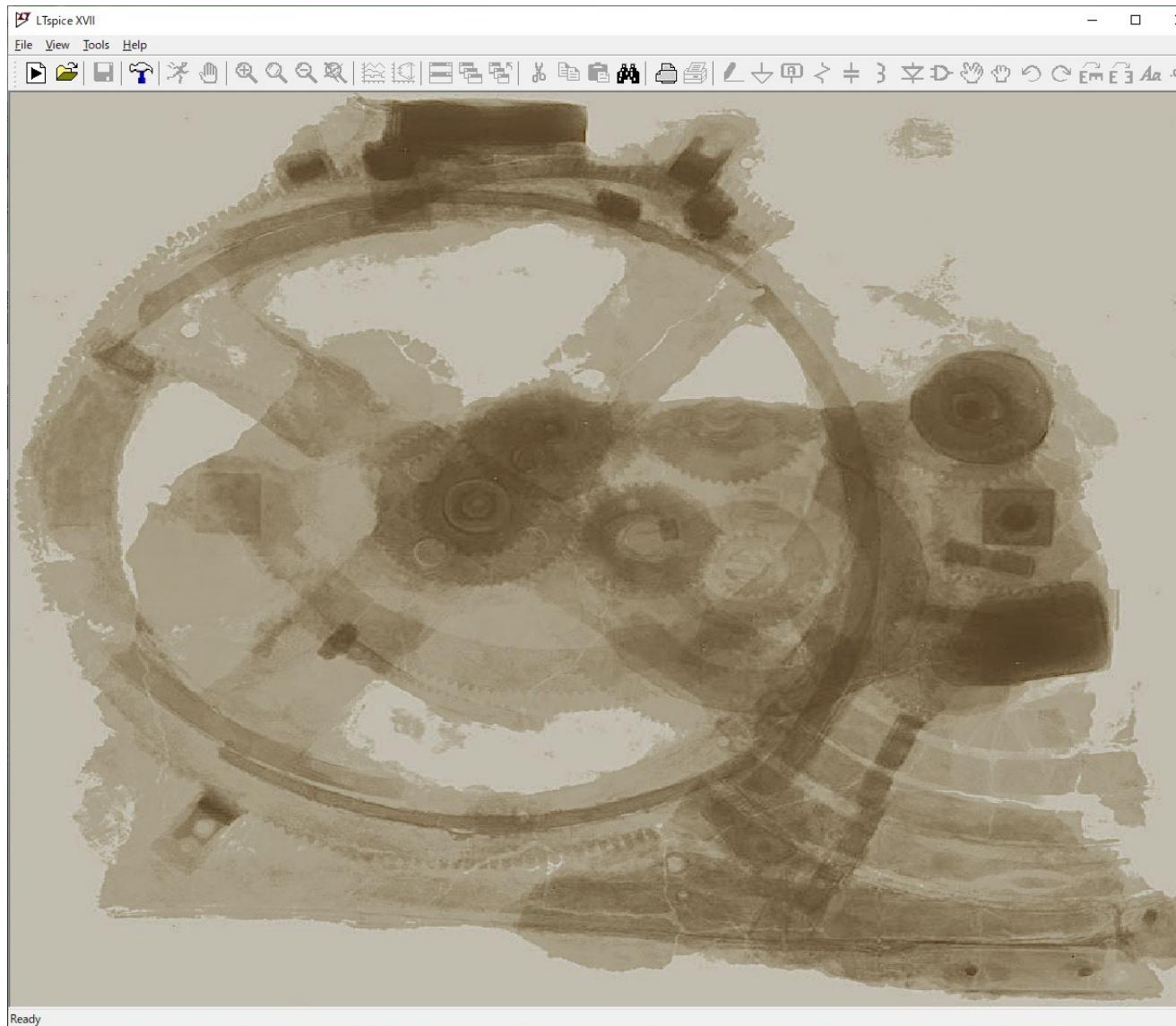
A library (LIBRARY directory) that summarizes information about parts such as Spice models  
Preparations will begin.  
After waiting for a while, LTspice will start up.

2020/09/07 9:38 アプリケーション 43,231 KB



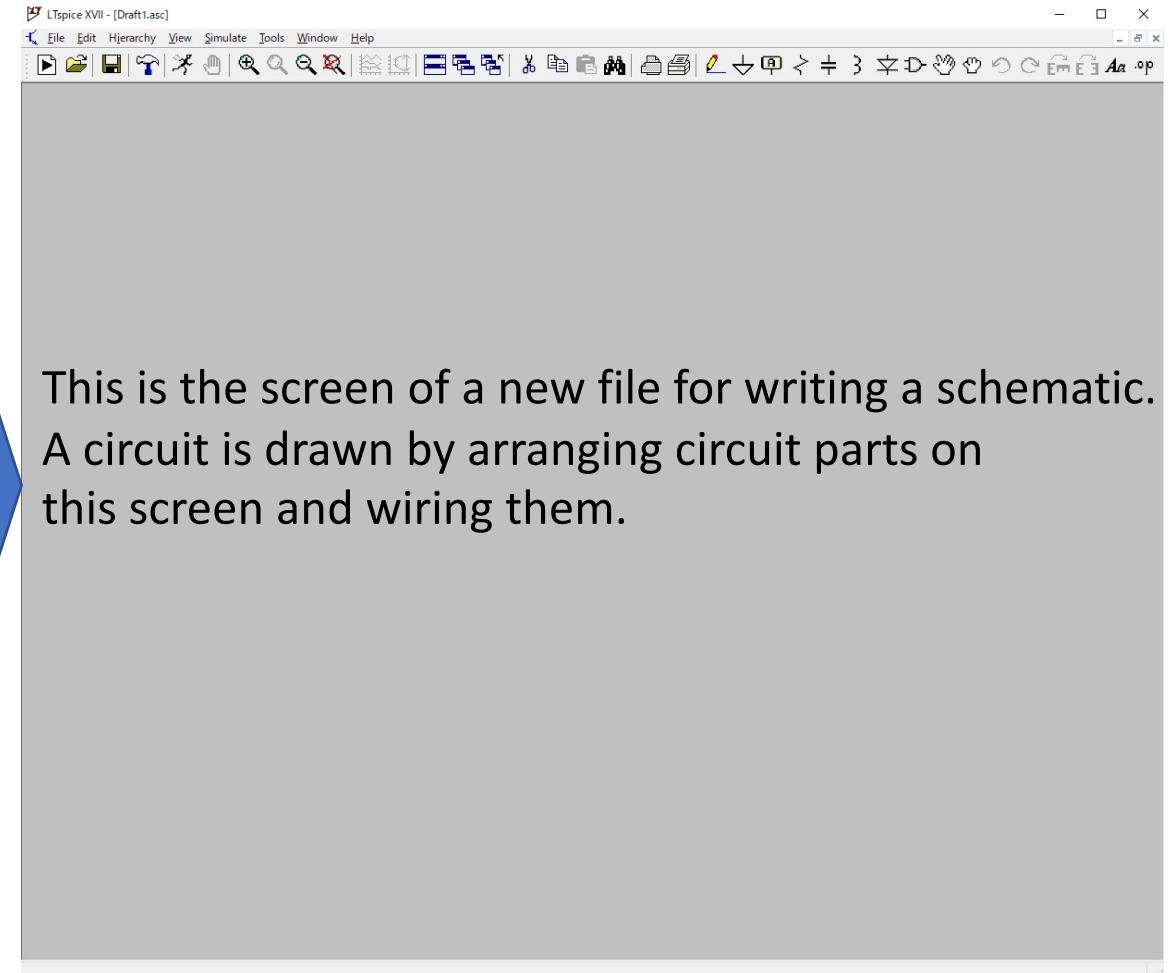
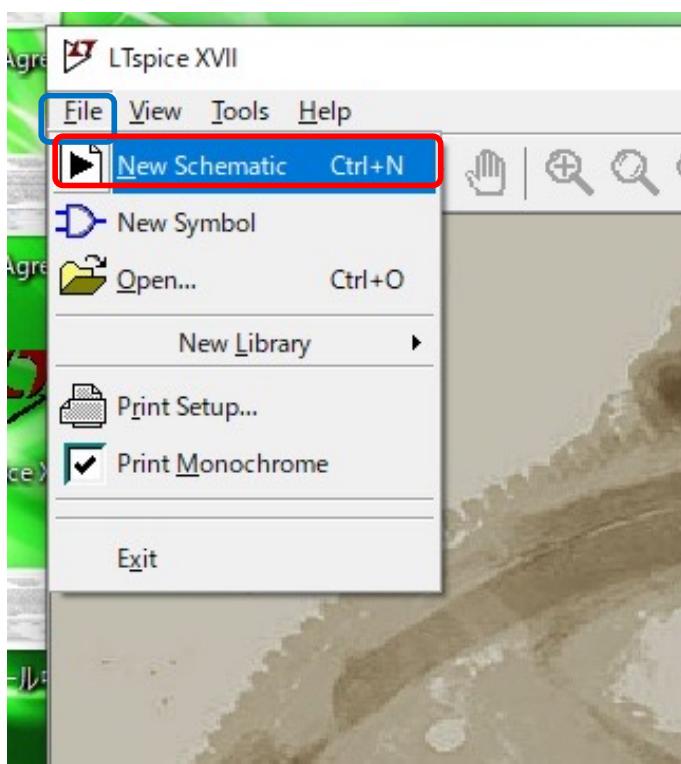
# Startup window of LTspice

When the installation is completed and the library is ready, LTspice will start up.



# Drawing circuit schematics

Select “**New Schematics**” from “File” menu, You can open a new file to draw the Circuit Schematic.



This is the screen of a new file for writing a schematic. A circuit is drawn by arranging circuit parts on this screen and wiring them.

For Mac

# Download of LTspice

- Please access below.  
<https://www.analog.com/en/resources/design-tools-and-calculators/ltpice-simulator.html>

Amplifier & Linear

Clock & Timing

Data Converter

EE-Sim

**LTspice**

Power Management

RF & Synthesis

Cybersecurity

## LTspice

**Fast • Free • Unlimited**

LTspice® is a powerful, fast, and free SPICE simulator software, schematic capture and waveform viewer with enhancements and models for improving the simulation of analog circuits. Its graphical schematic capture interface allows you to probe schematics and produce simulation results, which can be explored further through the built-in waveform viewer.

Learn how to use LTspice with [our tutorials below](#) or dive deeper with our selection of helpful tips and articles. You can also browse our library of macromodels and demo circuits for select Analog Devices products.

LTspice's enhancements and models improve the simulation of analog circuits when compared to other SPICE solutions. Download LTspice below to see for yourself!

**Download LTspice**

Download our LTspice simulation software for the following operating systems:

Date models updated - Jun 17 2024

[Download for Windows 10 64-bit and forward](#) Version 24.0.12

[Download for MacOS 10.15 and forward](#) Version 17.2.4

[Download for Windows XP](#) (End of Support)

[Download for MacOS 10.9](#) (End of Support)

[Download LTspice XVII for Windows](#) (End of Support)

**Download the file that suits your PC environment.**

**In the case of Mac,**

**File name: LTspice.pkg**

**After downloading the installation file, double click the file to start to install.**

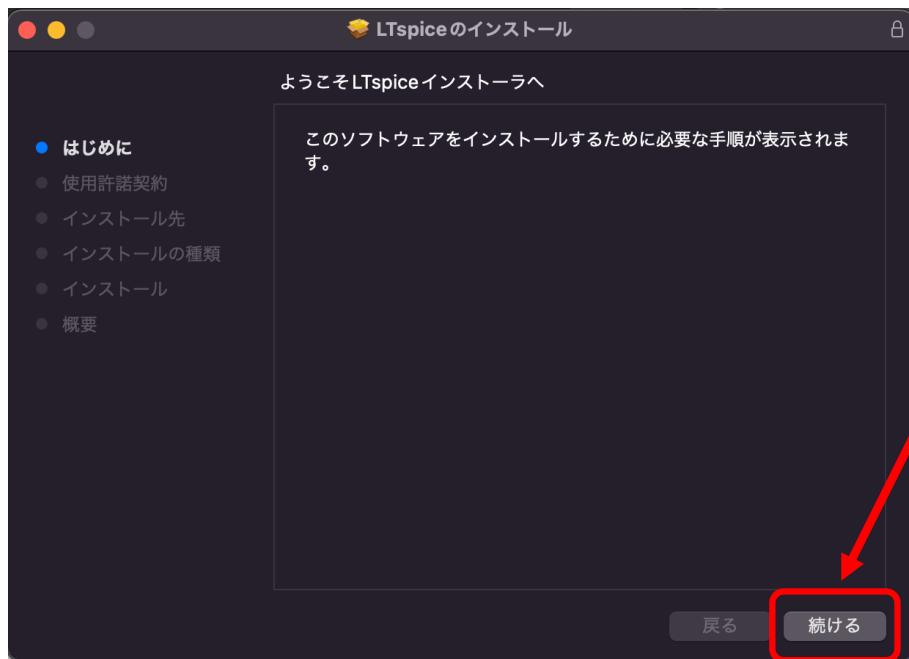
# Installation of LTspice for Mac



In the example on the left, Installer file is saved on the desktop.  
The file name is "LTspice.pkg".

Depending on the browser settings, it may be saved in the download folder.

**Double-click the file to start the installer.**



The installer will start.  
Please select “Next” to proceed.

Sorry!!

My Mac's locale is set to Japan. Therefore, the screenshot above is in Japanese.

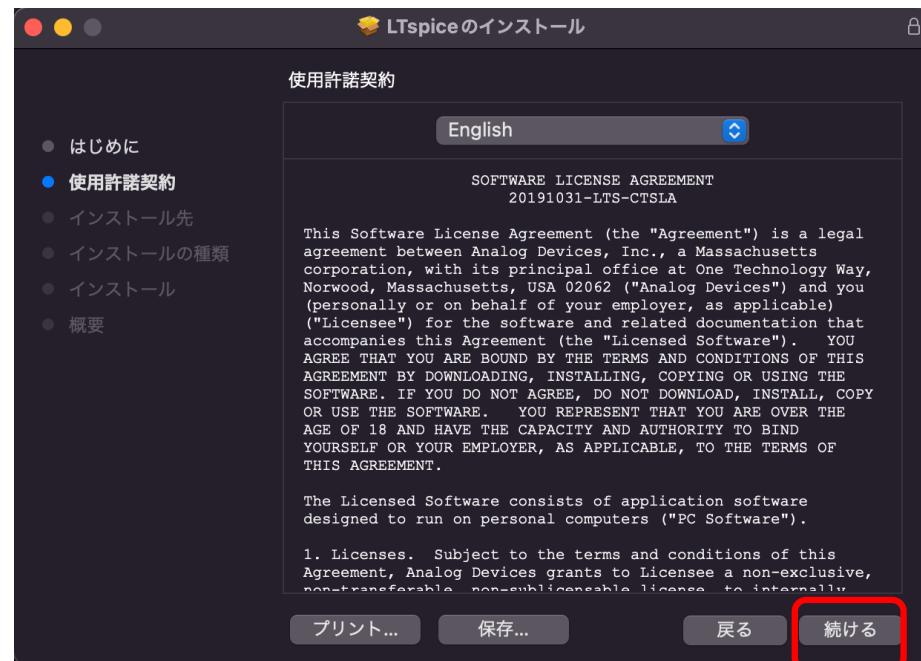
# License Agreement

After starting installation...

A: “License Agreement” window will open, so click “Continue”.

B: Message window will appear, so click “Agree” to accept.

A



Continue

B



# Starting installation

C: You will be asked where to install, so select “Install” and proceed.

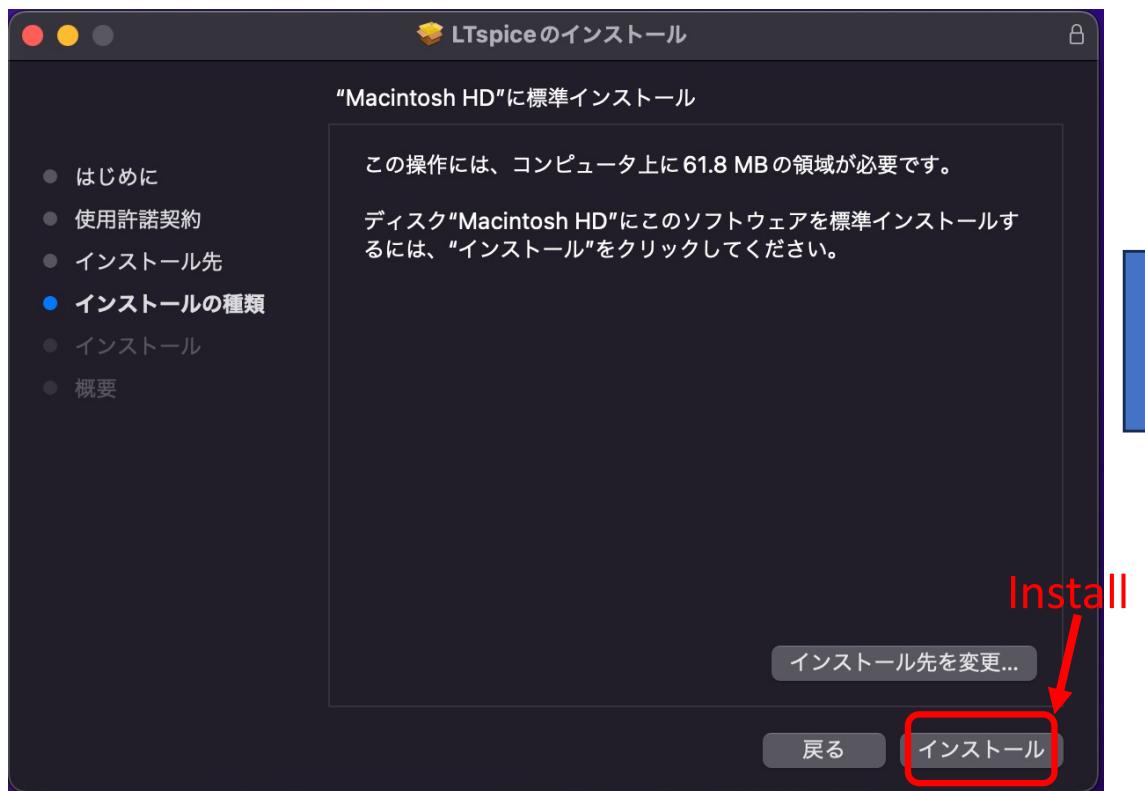
\* Usually, the default is fine.

\* You can change the installation location, if necessary.

D: Depending on your system configuration, you may be required to enter a password to allow the installation.

For that case, please enter your login password and proceed.

C



D

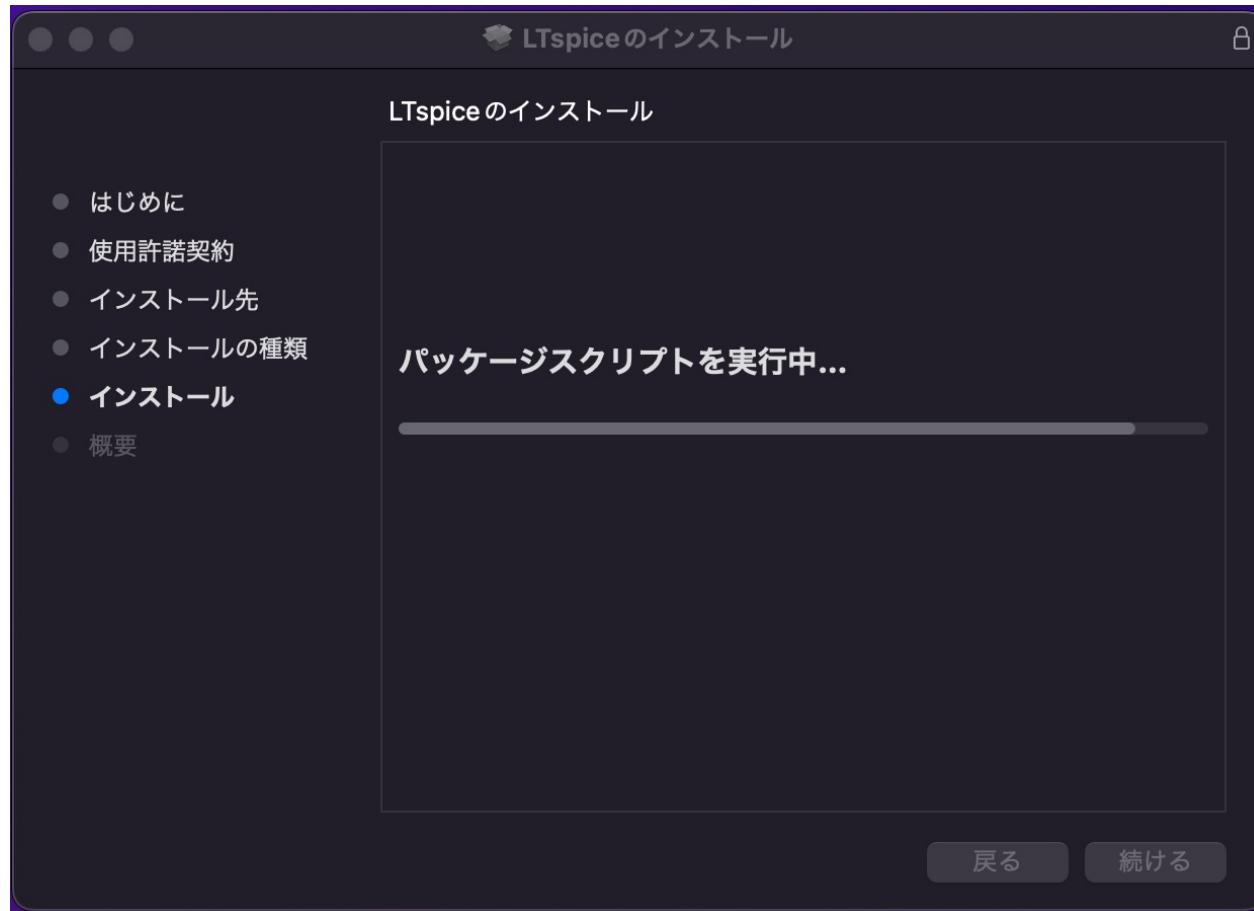


Sorry!! Screenshots above are in Japanese.

# In the middle of installation...

The window below is displayed during installation.

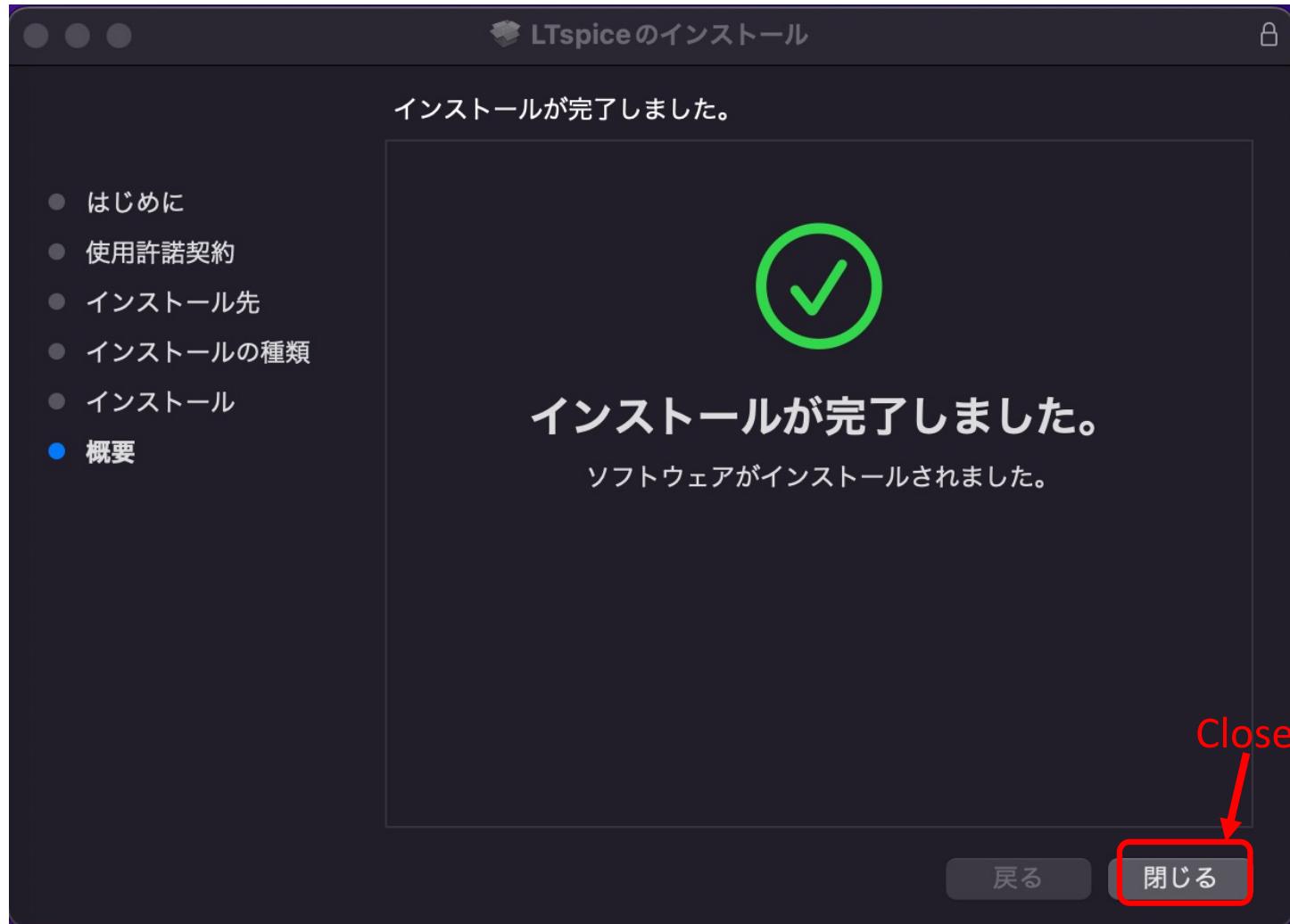
Sometimes, in the middle of the installation,  
a message like “LTspice is trying to access your document folder.” dialog may appear.  
In that case, press “Continue” to proceed.



Sorry!! Screenshot above is in Japanese..

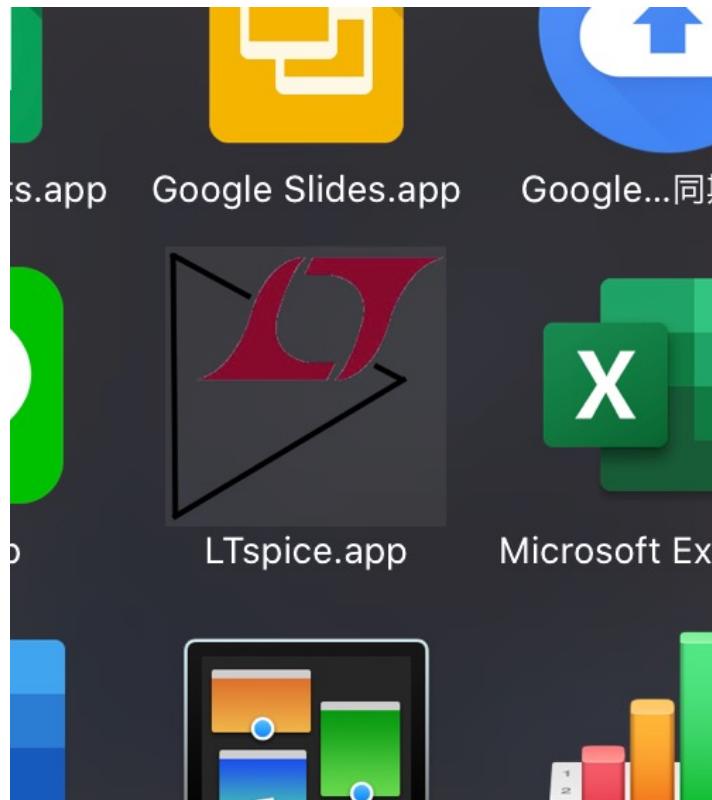
# Finish installation

When the following screen appears, press “Close” to finish the installation.  
Once the installation is complete, launch it from the application folder.



Sorry!! Screenshot above is in Japanese..

# Lunching LTspice



Double-click the LTspice icon to start it.

\* At first time to lunching LTspice, the following message may appear.

"Do you want to allow this app to make changes to your device?"

LTspice® XVII installation program

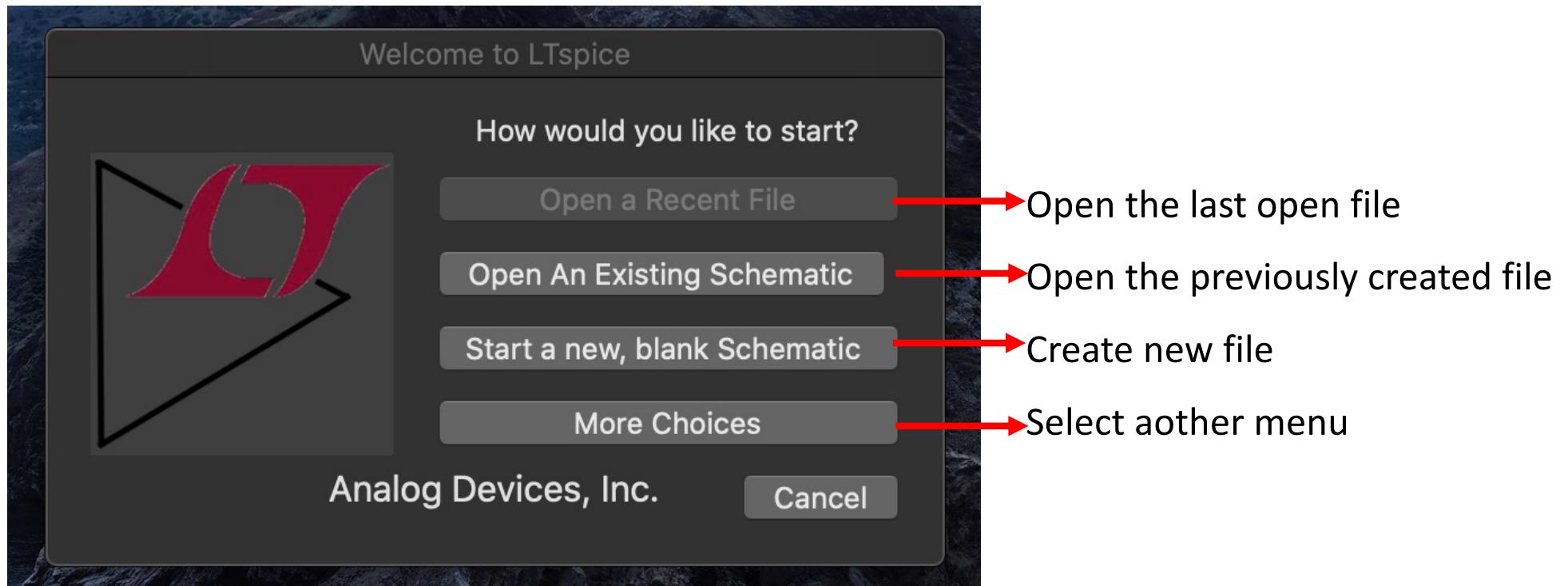
Select "Yes".

# “Welcome to LTspice” window after launching LTspice

When you start LTspice for Mac, the following small dialog will open.

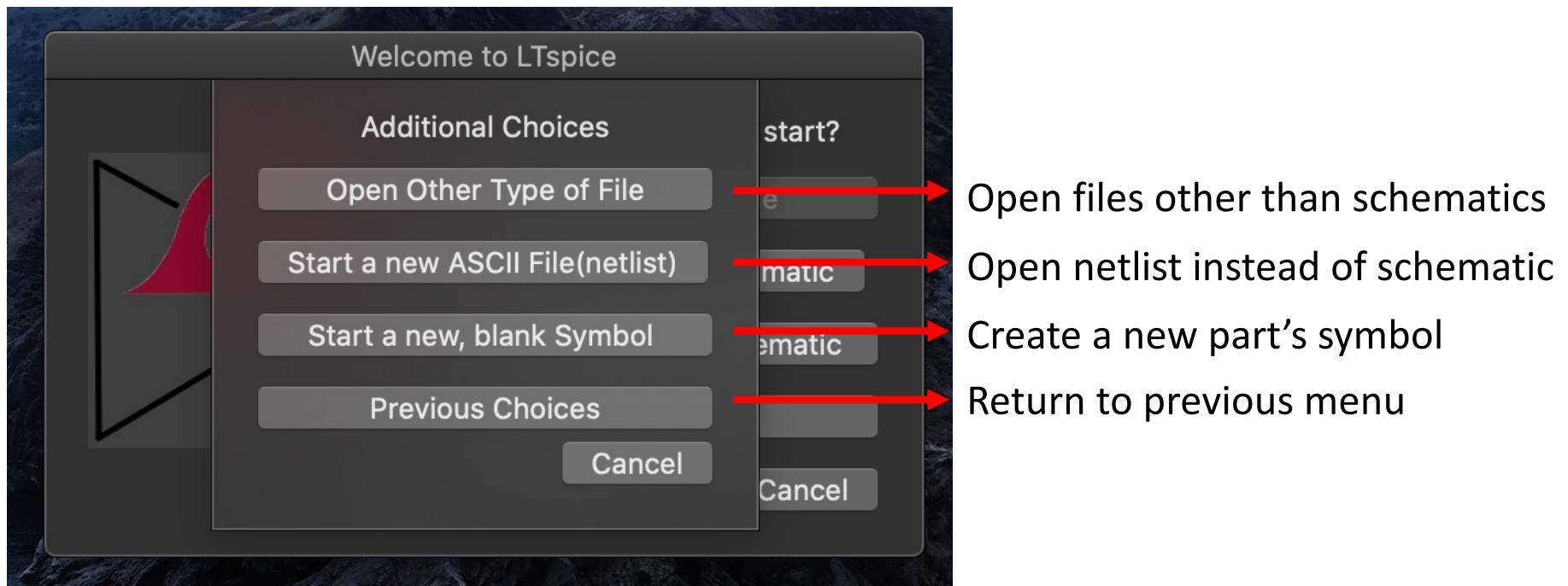
From here you can open existing files or create new ones.

It is also possible to cancel this screen and open or create a file from the LTspice menu.

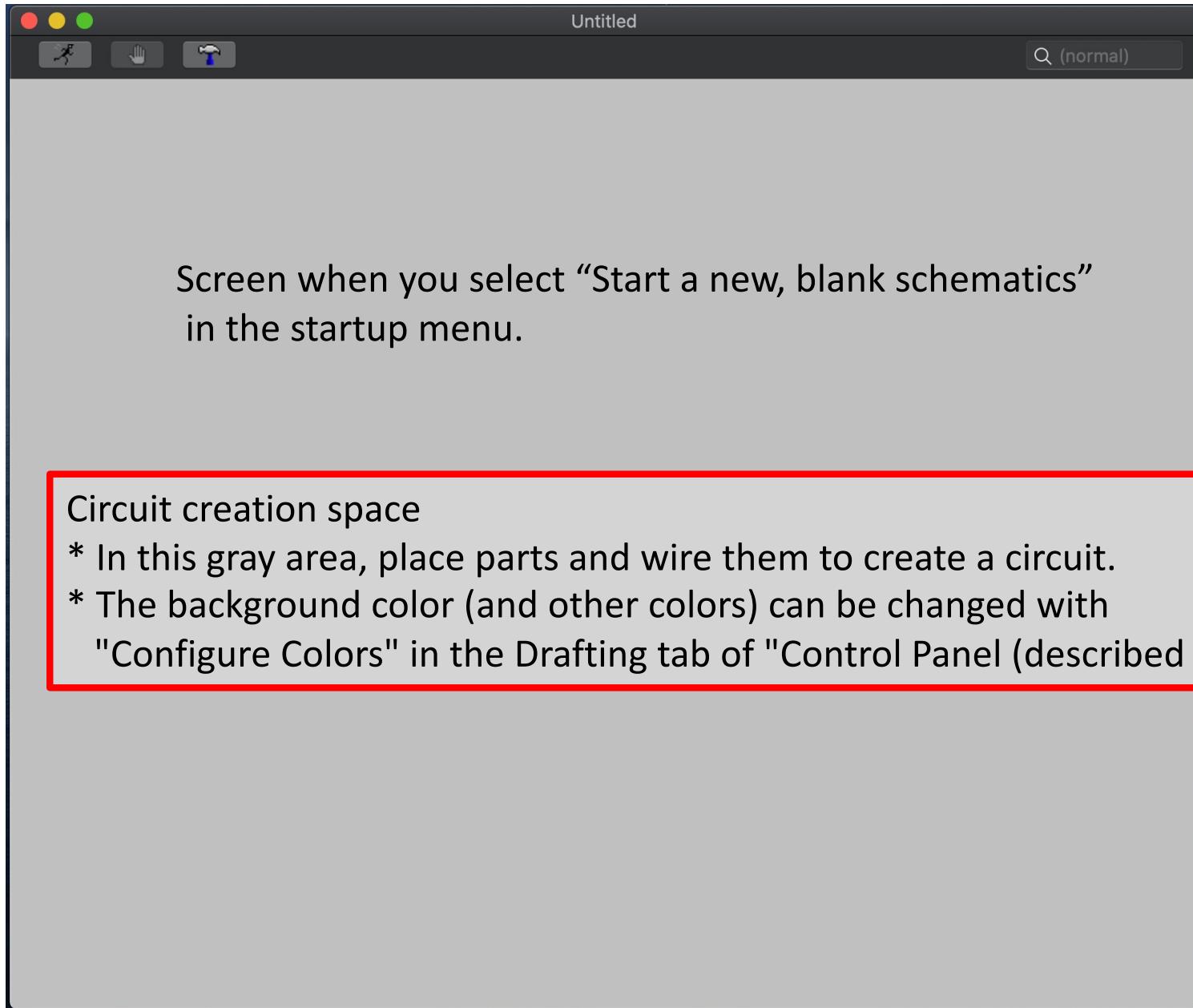


# Sub menu of “Welcome to LTspice” Window

If you select "More Choices" in the “Welcome to LTspice” window ,  
the following menu will appear.



# Lunched LTspice window

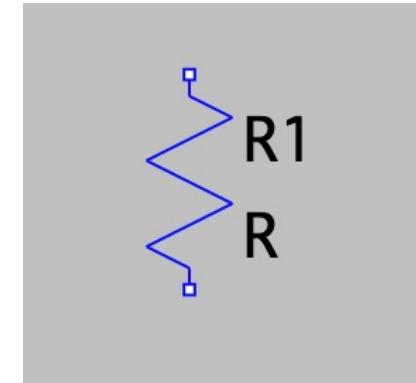
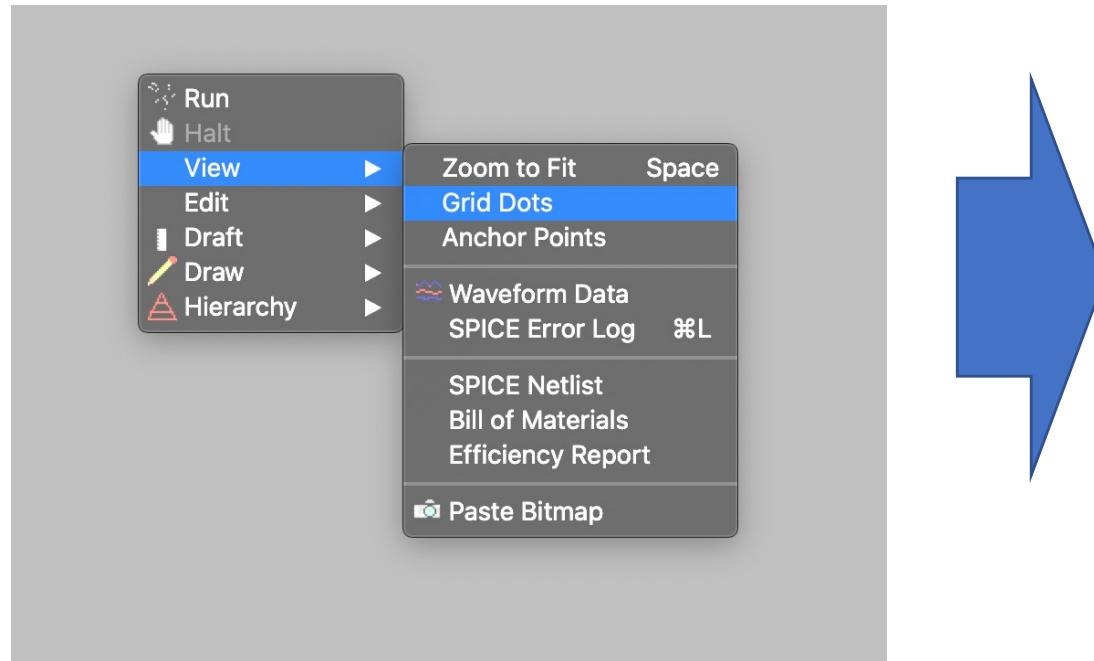


Practice to draw circuit schematics

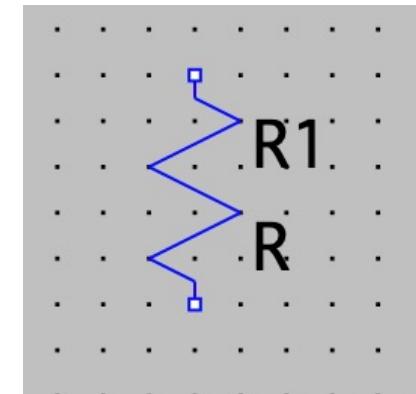
# Display grid dots

Right-click anywhere on the screen to bring up the menu.

You can turn the grid display on and off by selecting  
"Grid Dots" from "View menu".



Without grid dots

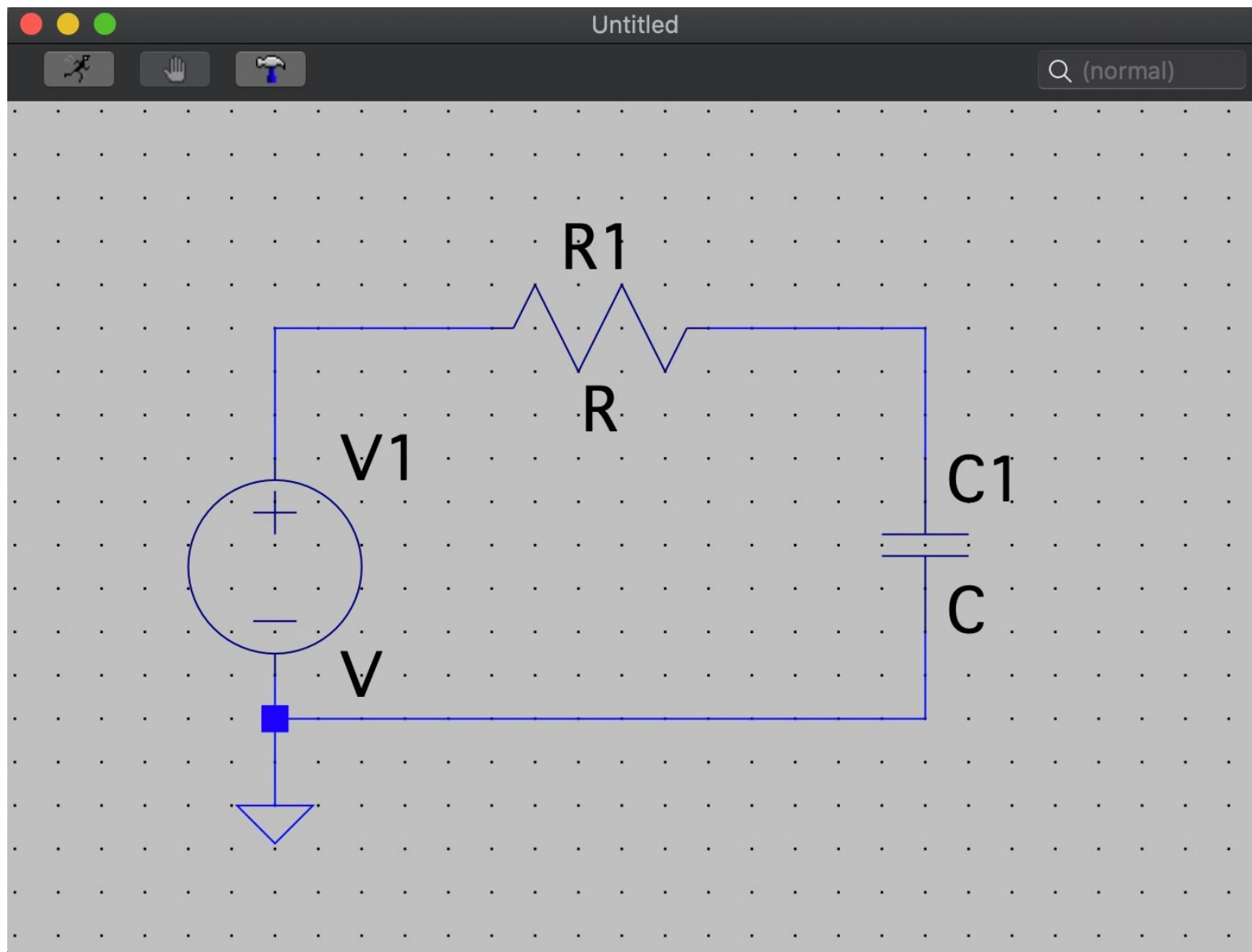


With grid dots

"With grid dots" makes it easier to decide position to put the parts.

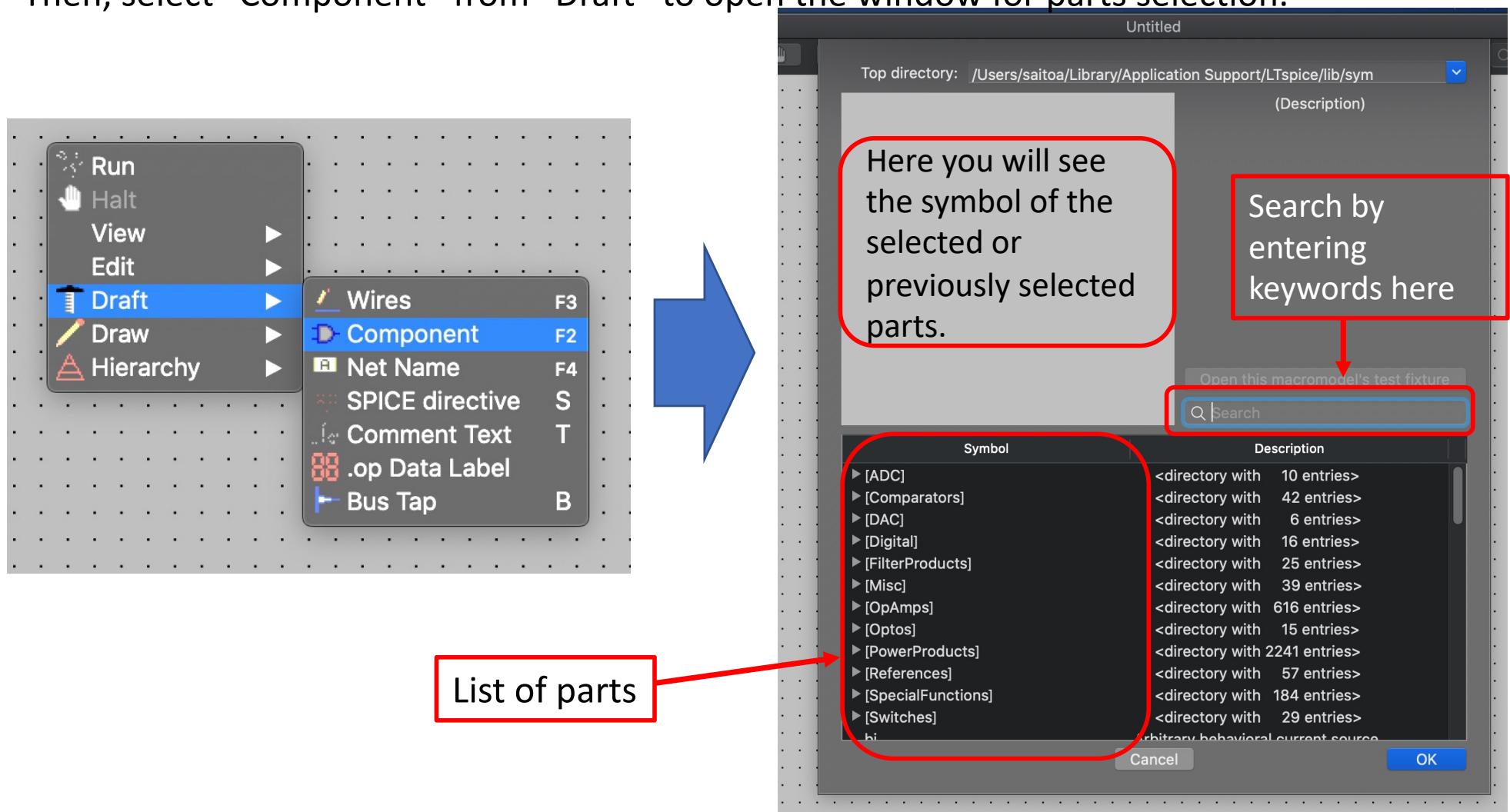
# Try to draw a simple circuit schematics

Let's draw the following circuit in practice.



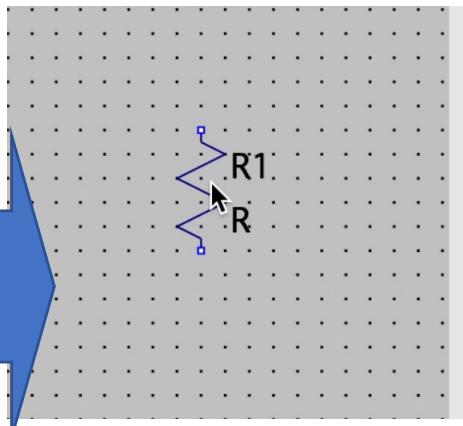
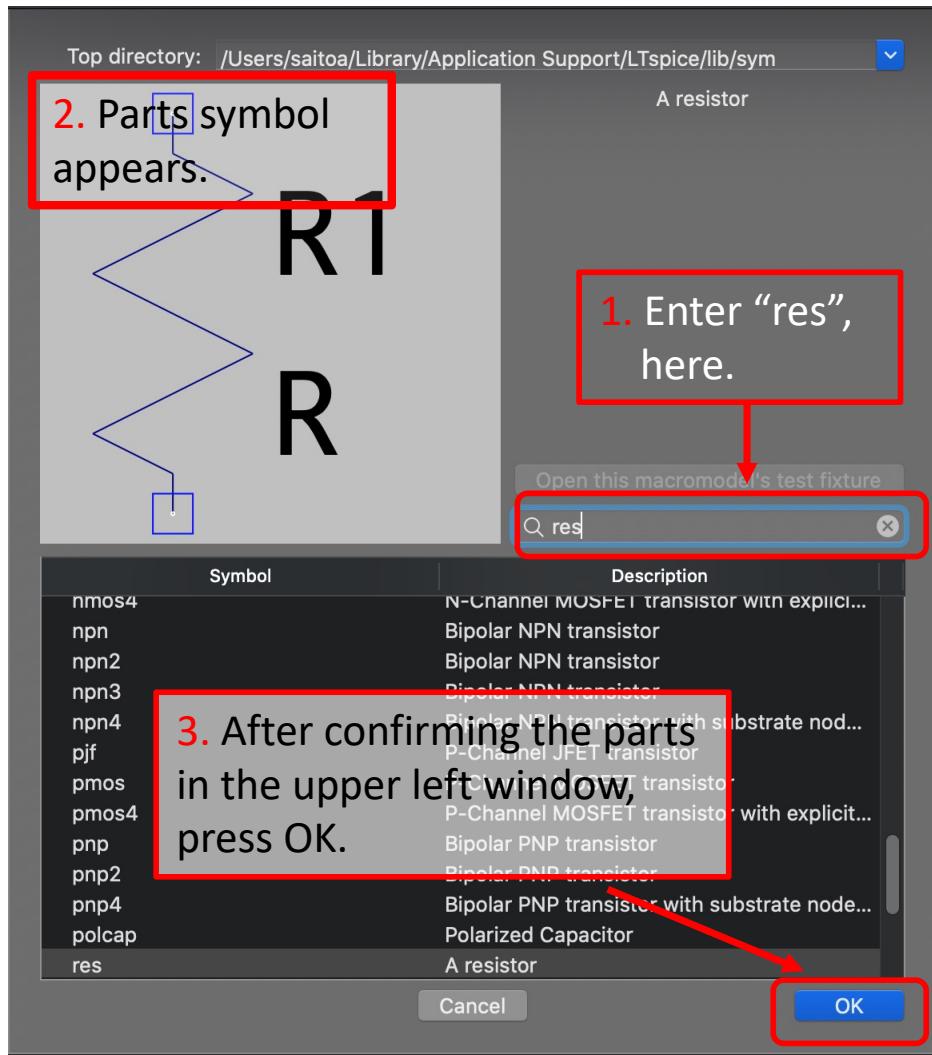
# Open the parts selection window

Right-click anywhere on the screen and select "Draft" from the menu that appears. Then, select "Component" from "Draft" to open the window for parts selection.

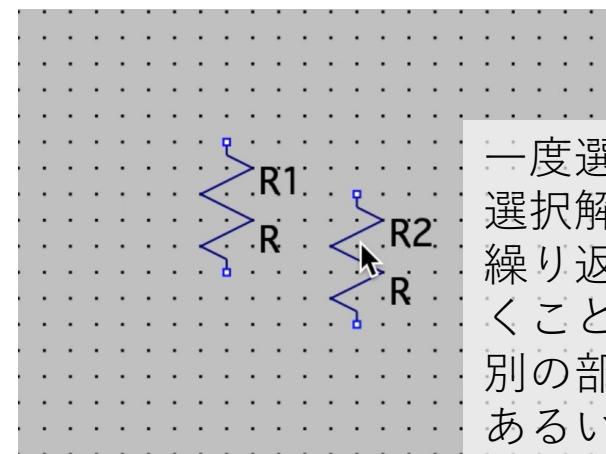


Window for parts selection

# Select a part (resistance) and place it



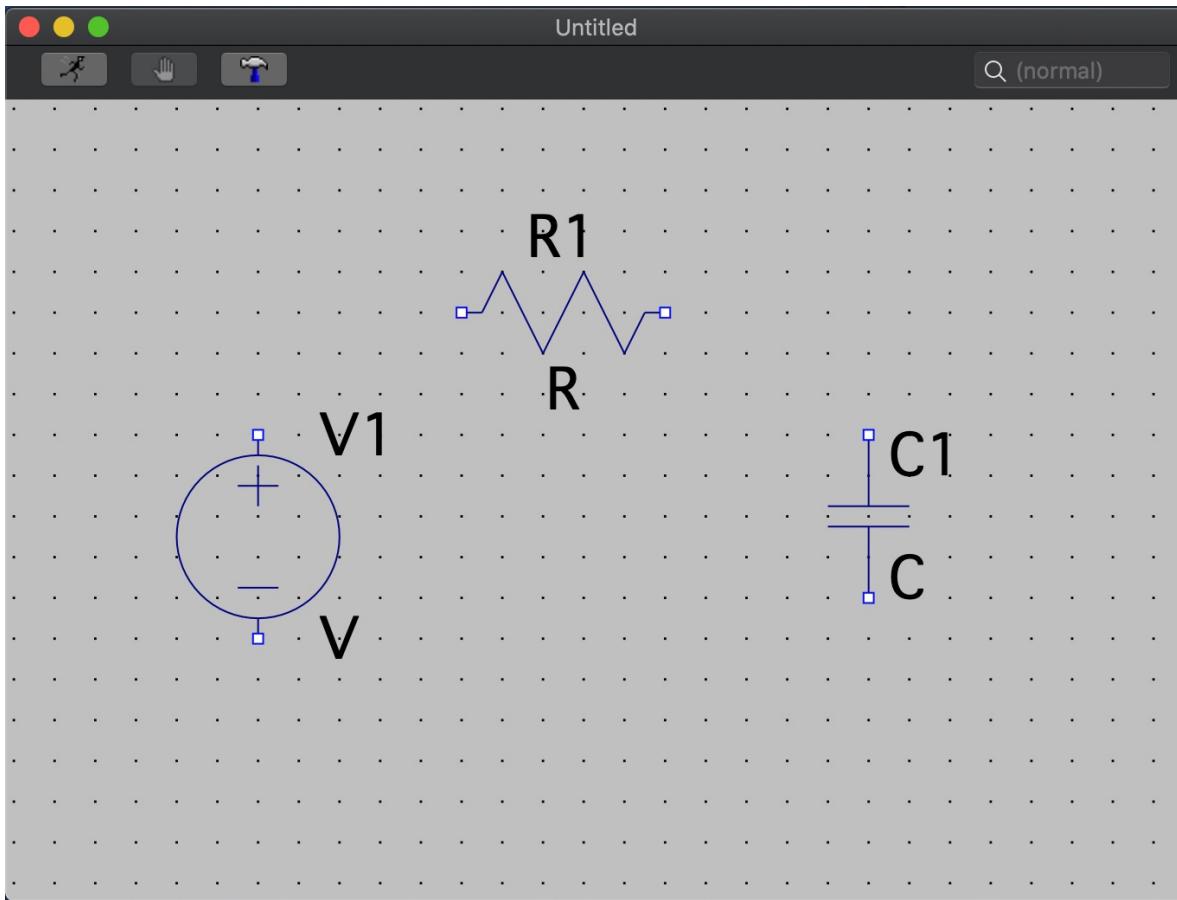
部品選択画面が消え  
選択したシンボルの  
配置モードになる。  
マウスカーソル付きの  
シンボルが現れるので、  
適切な位置でクリック  
すると部品が配置される。  
この時配置のクリック前に  
Control + Rを押すと部品が  
回転する。



一度選択された部品は  
選択解除しない限り、  
繰り返し同じ部品を置  
くことができる。  
別の部品に変えたい場合、  
あるいは部品の配置をや  
めたい場合はescボタン  
で選択解除する。

"Res" is one of the basic part names given to resistors in LTspice, which stands for "resistance". Except for the parts indicated by the model number, they are represented by such basic part names.

# Placing parts



Enter the following keywords in the search keyword input area for component selection, and place resistors, capacitors, and DC voltage source.

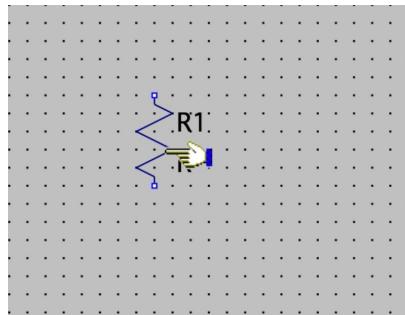
Resistance ⇒ "res"

Capacitor ⇒ "cap"

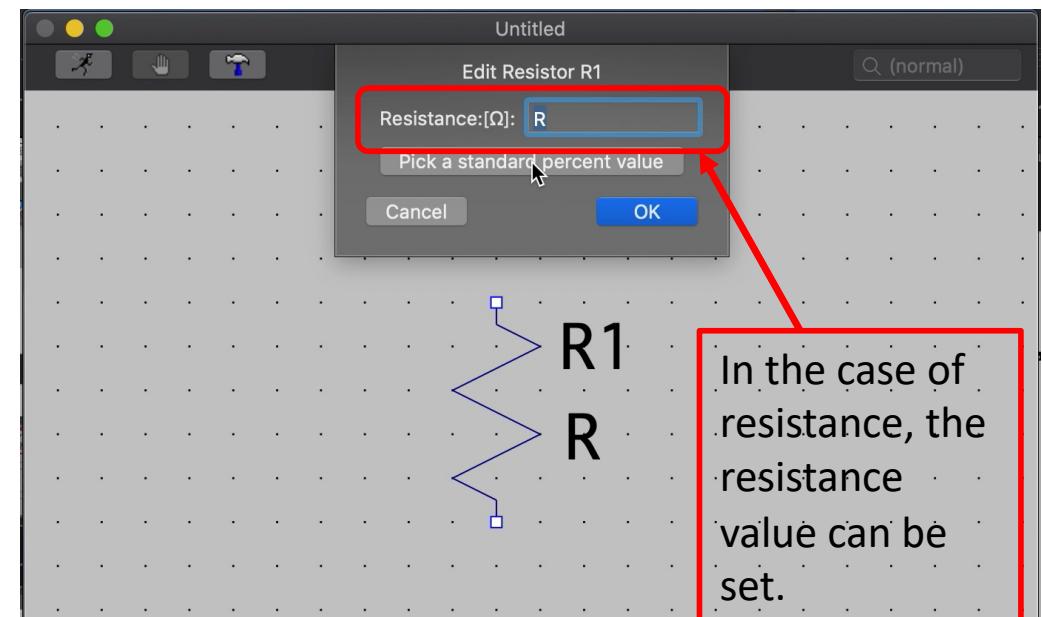
Voltage source ⇒ "voltage"

# Configuration of parts properties

When you move the mouse cursor to the parts placed on the screen, the shape of the cursor changes to the shape of the index finger.

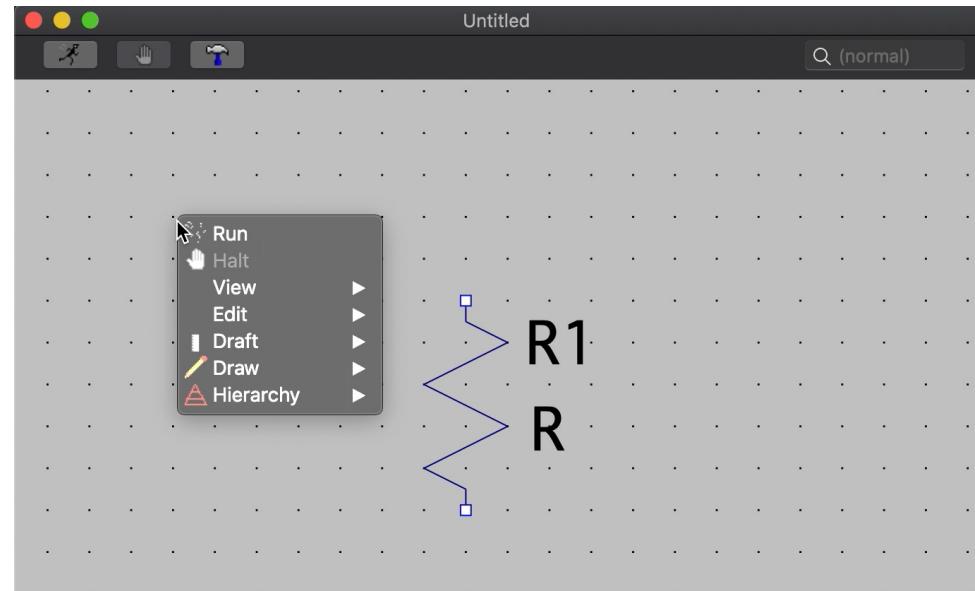
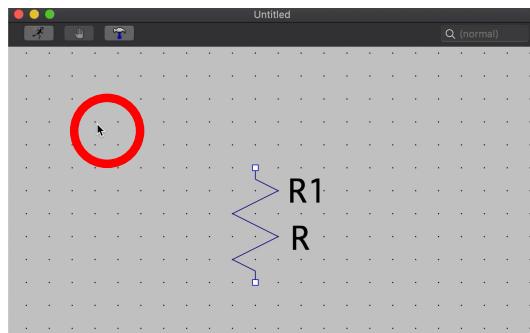
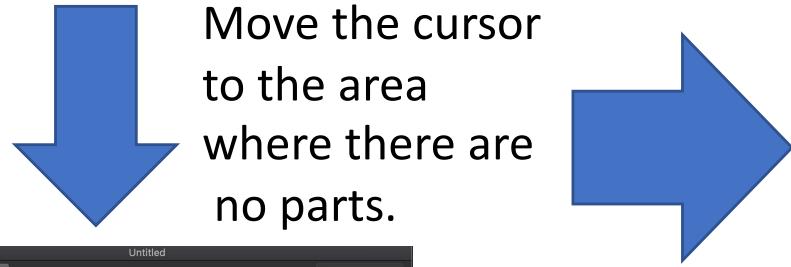
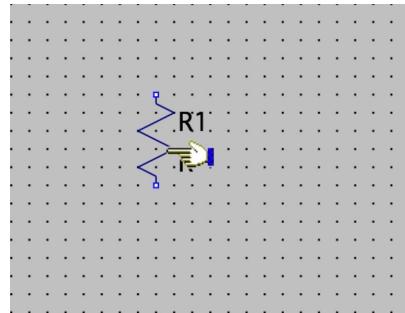


You can change the element value or the element model number by right-clicking in the state shown on the left.



# Operation for parts

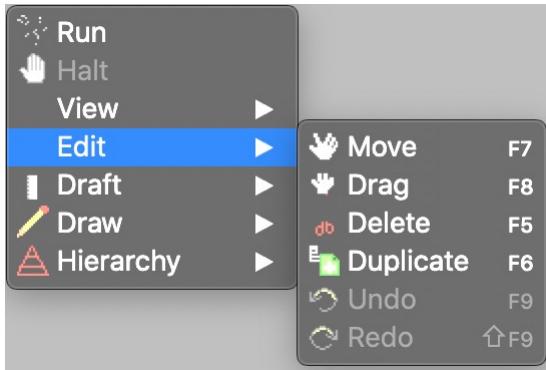
Select "Edit" from the menu that appears when you right-click on an area where there are no parts on the screen and perform operations such as moving or deleting parts.



An operation menu will appear.  
Select an operation for parts from  
this menu.

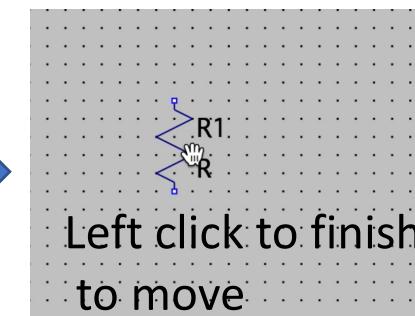
# Moving, rotating, and deleting parts

From “Edit” menu...



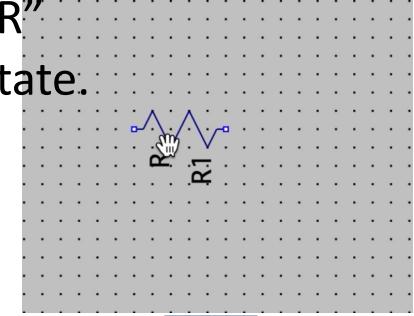
Select  
“Move”,  
or  
“Drag”

Moving parts



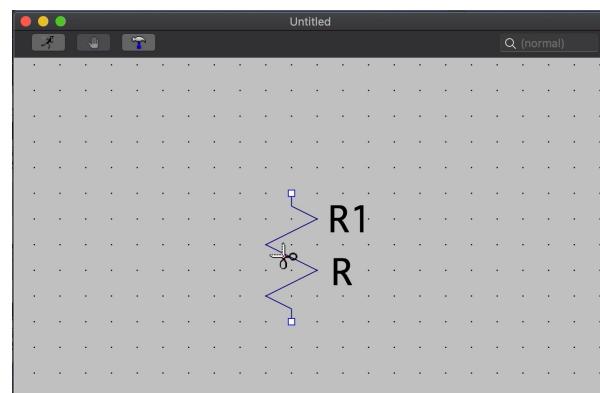
Rotate 90 °  
each time you  
press “control + R”  
in the selected state.

Rotating parts



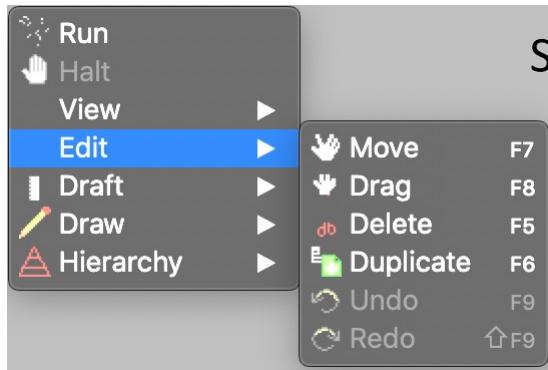
Select “Delete”

Deleting parts

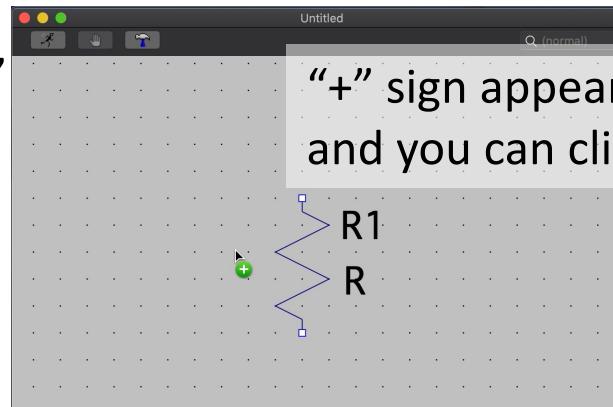


The cursor changes to a scissors shape.  
Click to select the part you want to delete.  
The part is deleted.

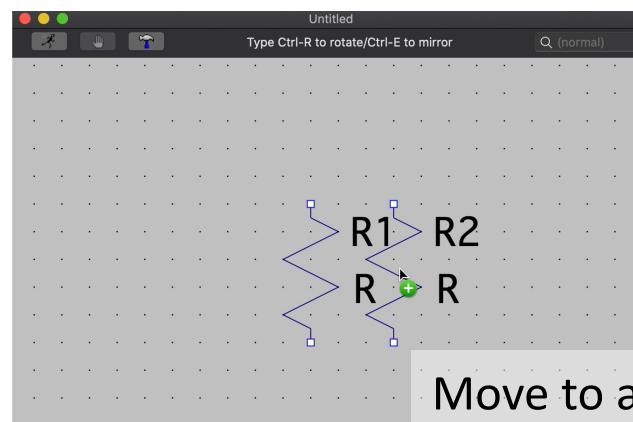
# Duplicating parts



Select “Duplicate”

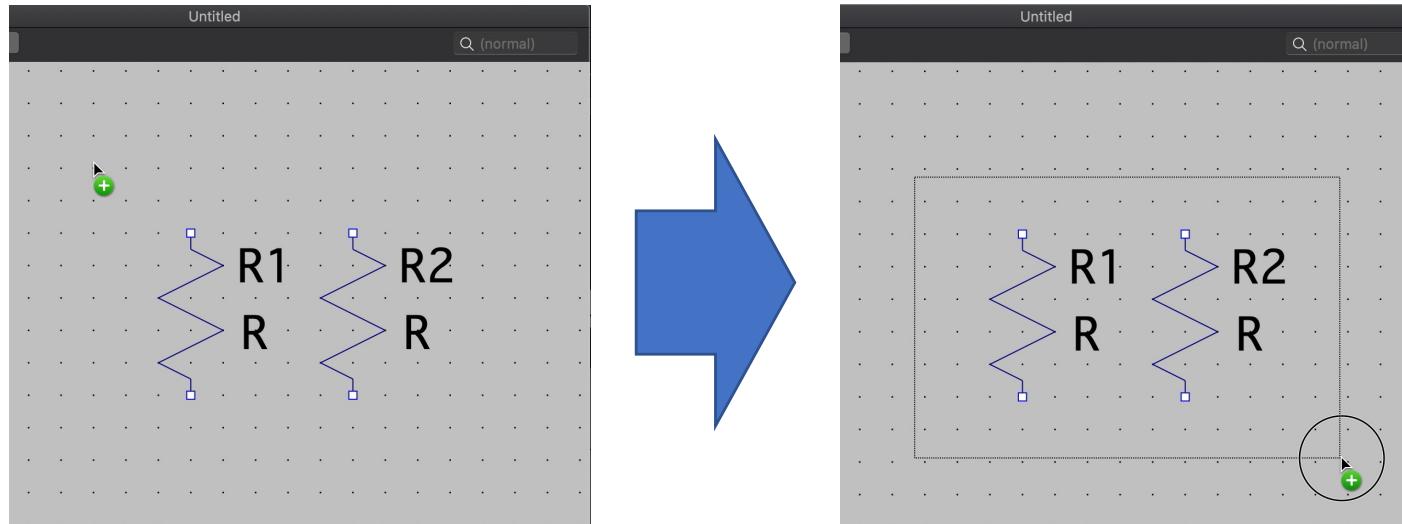


“+” sign appears near the cursor,  
and you can click to duplicate the part.

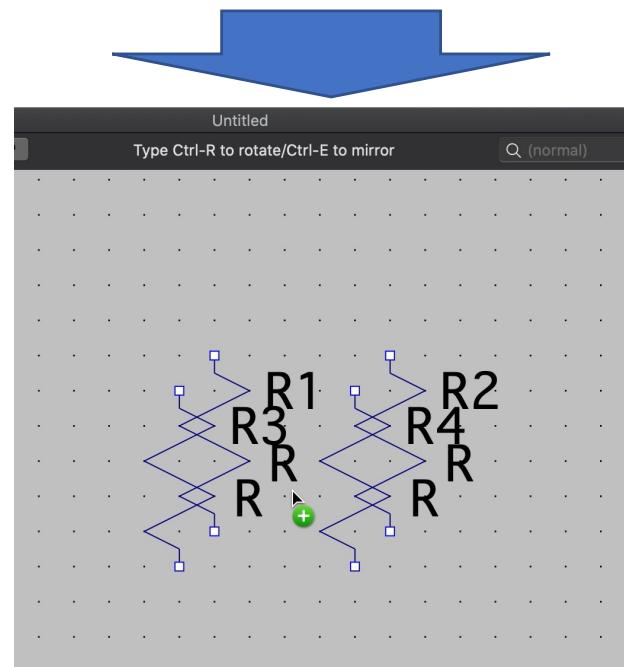


Move to a suitable position  
and click to place a duplicate.  
Cancel the operation with "esc".

# Operating multiple parts at once (Case of “Duplicate”)



When you can Duplicate, drag them around the objects you want to operate in a batch.  
Those are selected.



You can operate multiple parts in the same way with “Move” and “Drag”.

In the case of “Delete”, Selected parts will be deleted when selected.

In the case of “Duplicate”, when you move mouse, Multiple elements are duplicated together at once.

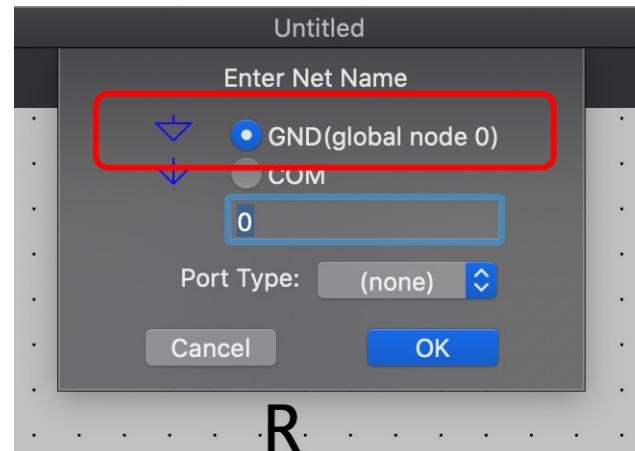
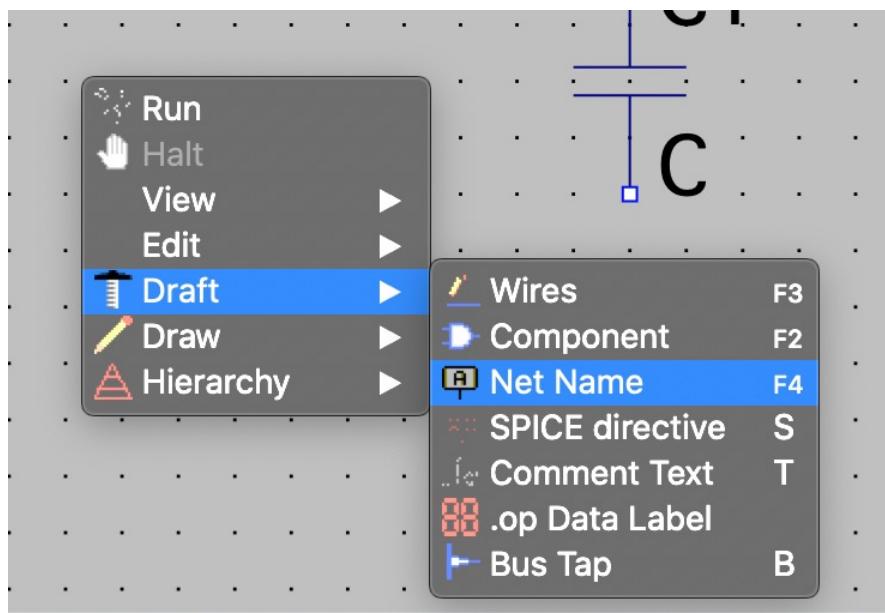
# Placing GND (Ground)

In the circuit simulator, the ground is intentionally placed. This is mandatory.

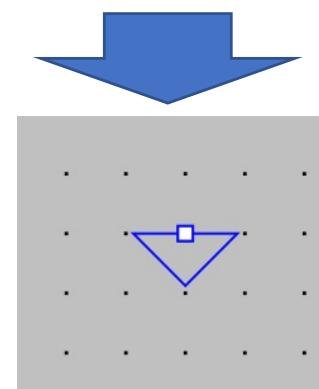
\* To define the voltage reference point for the simulator

Right-click at a position where the mouse cursor does not overlap the part, etc. to open the menu, and open the menu.

“Draft” ⇒ Select “Net Name”.

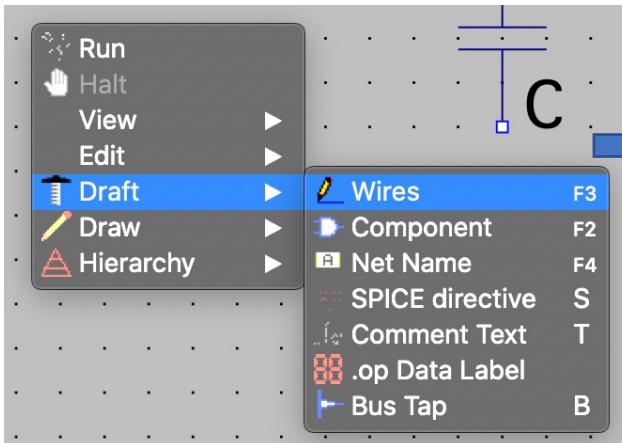


Select “GND (global node 0)” in the window that appears.

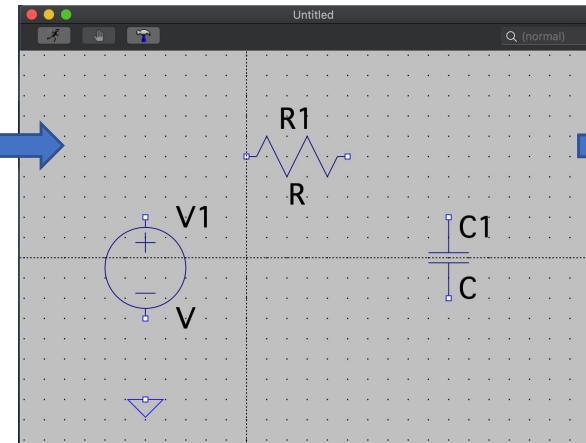


Since the GND symbol in LTspice appears, move it to an appropriate position and click to place it.

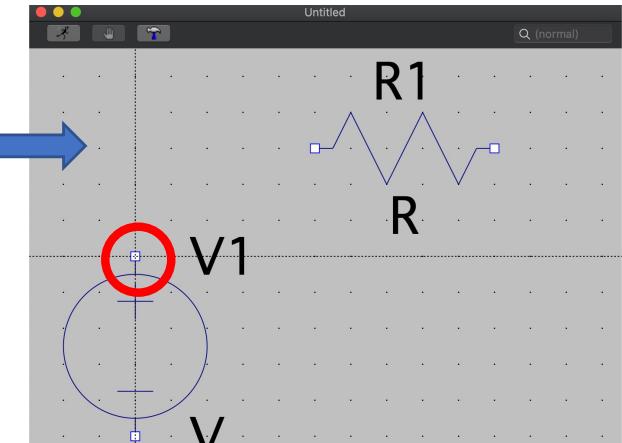
# Wiring



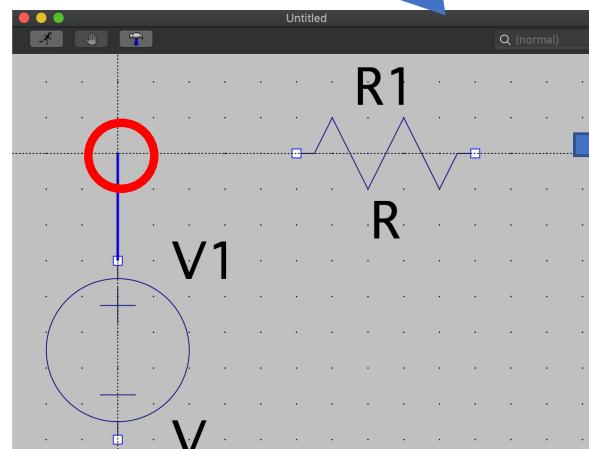
Right-click to open the menu and select "Draft"  $\Rightarrow$  "Wires" to switch to wiring mode.



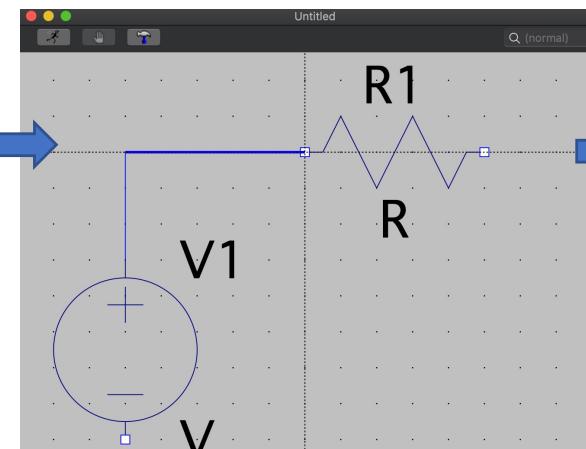
Vertical and horizontal guides appear in wiring mode .



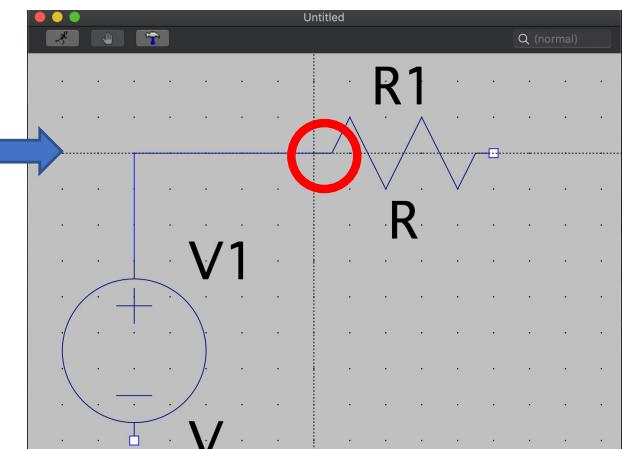
Move the guide to the starting point for wiring and click.



To wire by moving the mouse.  
\* Not a drug.

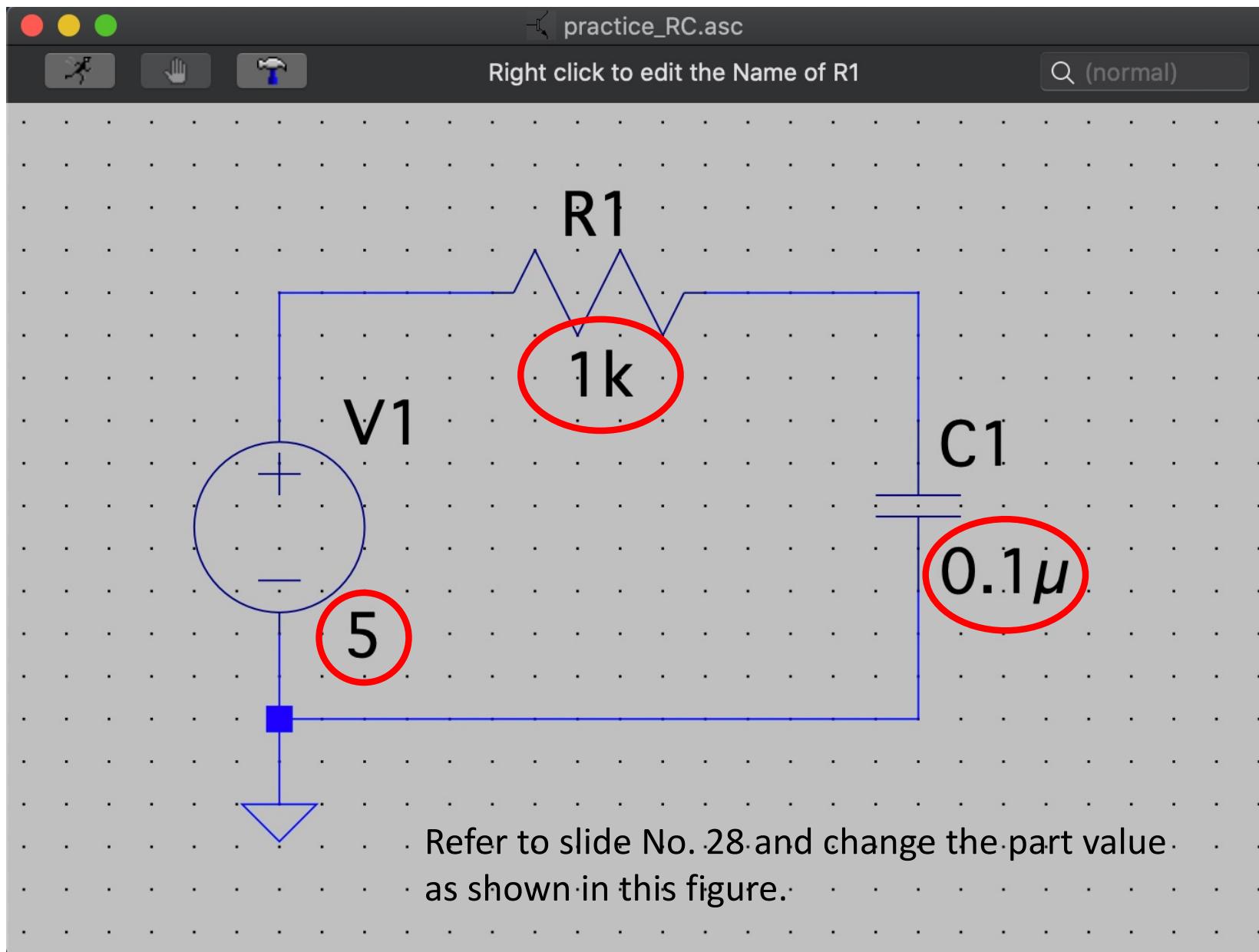


If you want to bend the wiring, click where you want to bend.



Click at the end of the wiring to finish.

# Completion of the practice circuit



# Saving circuit design

I think there is a "LTspice" folder in the "Document" folder on your PC.

Prepare a folder such as "For\_spice" for use in class and save future simulation files under this folder.

In the example below, saved as

"Documents" folder in your PC

⇒ "LTspice" folder

⇒ "For\_spice" folder

⇒ "Practice" folder

⇒ "practice\_RC.asc".

In the future, create a folder for each simulation so that you can easily understand it later.

