

To use OP.amp “LM358N” in LTspice  
and  
AC analysis preparation and simulation

# To use “LM358N” in your LTspice

## Windows Ver.

- Open your user folder.
- Open “AppData” folder in your user folder.
- Open “Local” folder in the “AppData” folder -> Find “LTspice” folder.
- Open “lib” folder in the “LTspice” folder.
- Open “sub” folder in the “lib” folder and make new folder “STMicro”.
- Copy the file “lm358n.sub” in the “STMicro” folder.
- Open “sym” folder in the “lib” folder
- Open “OpAmps” in the “sym” folder and make new folder “STMicro”.
- Copy the file “lm356n.asy” in the “STMicro” folder.

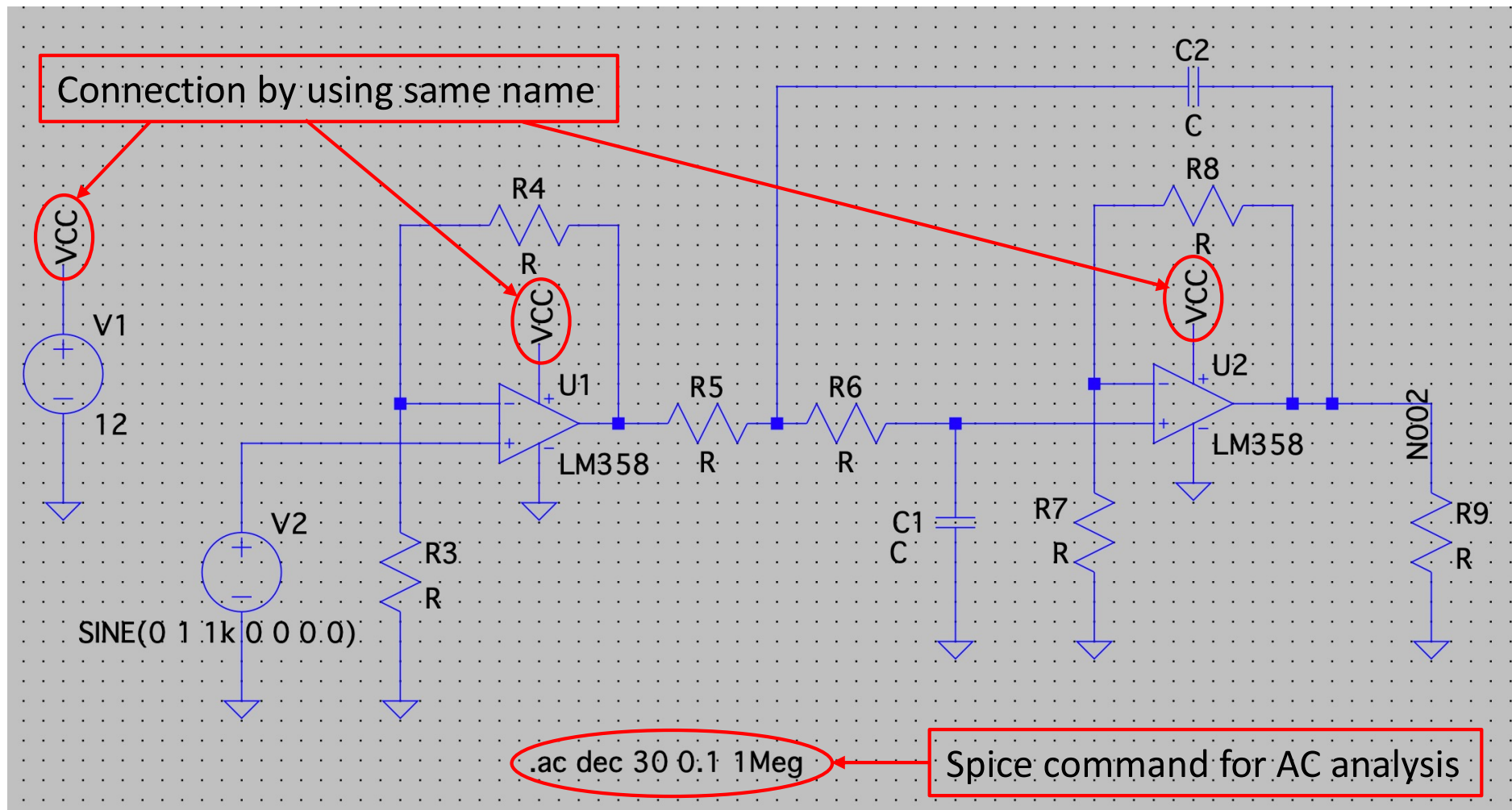
## Mac Ver.

- Open “library” folder in your user folder.
- Open “Application support” folder in the “library” folder.
- Open “LTspice” folder in the Application support” folder.
- Open “lib” folder in the “LTspice” folder.
- Open “sub” folder in the “lib” folder and make new folder “STMicro”.
- Copy the file “lm358n.sub” in the “STMicro” folder.
- Open “sym” folder in the “lib” folder
- Open “OpAmps” in the “sym” folder and make new folder “STMicro”.
- Copy the file “lm356n.asy” in the “STMicro” folder.

\* “STMicro” is the name of the provider of the spice model of LM358N.

# Simulation circuit for AC analysis

- LM358N can operate with single power supply.  
In this case, 12V DC supply is used for driving voltage of circuit.
- In AC analysis, input signal to circuit should be sine wave.  
Sine signal voltage source is connected to circuit instead of sensor output from sensor circuit.



Please set the value of each R and C depending on your circuit design.