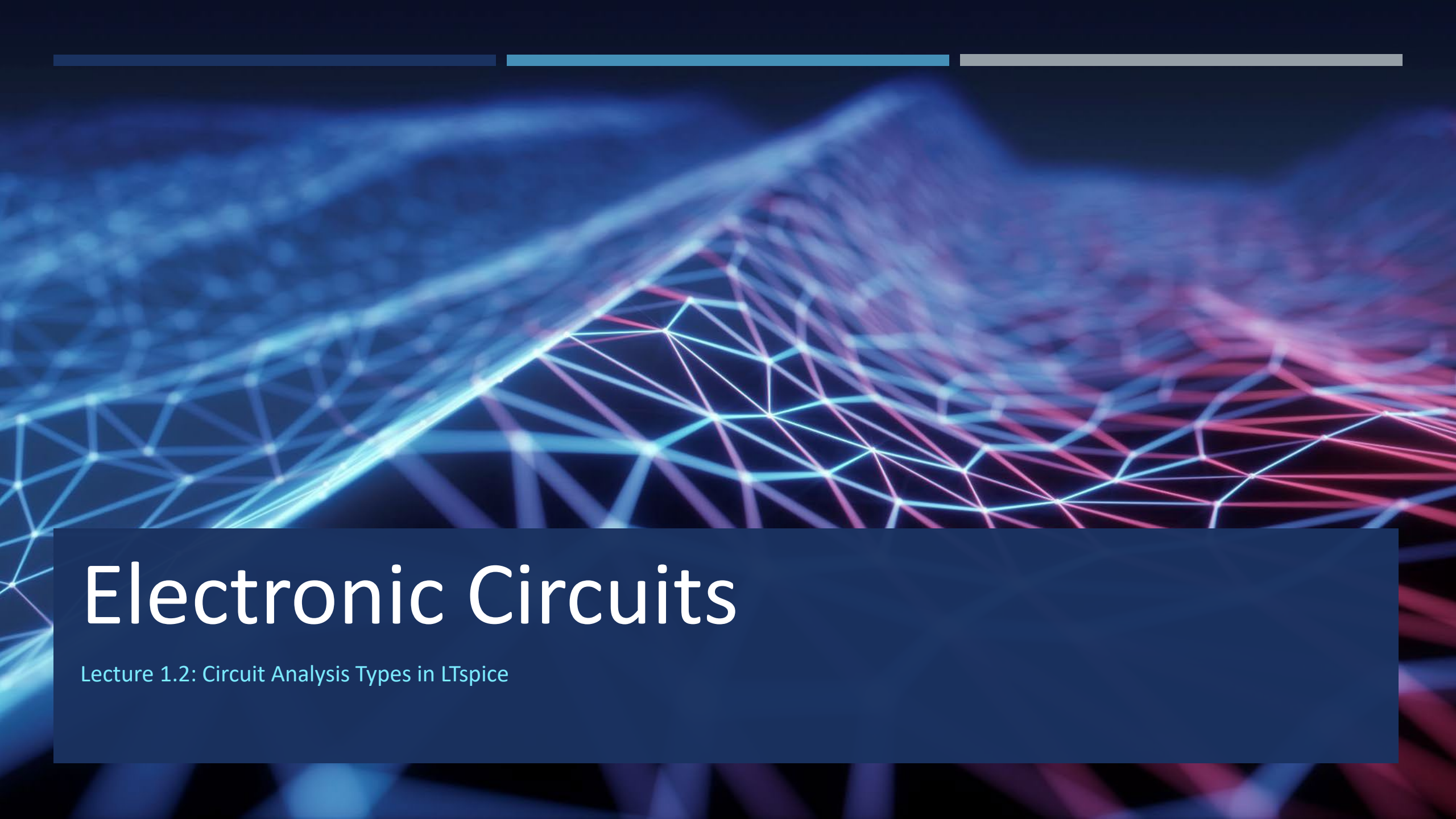


Electronic Circuits

Lecture 1.2: Circuit Analysis Types in LTspice

Circuit Analysis Types

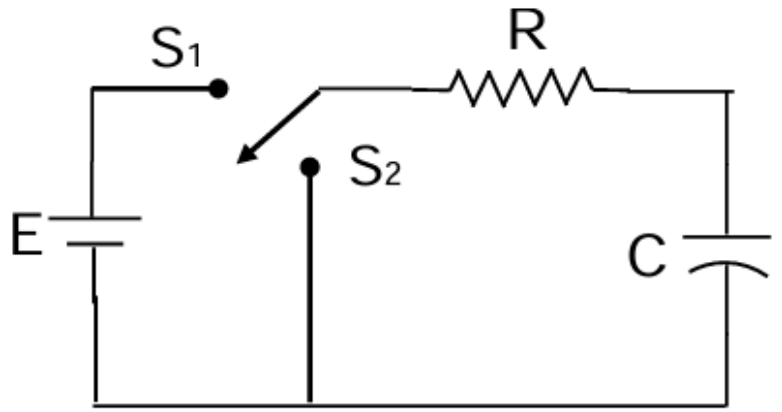
- By hand, we conduct
  - DC Analysis (Inputs and Outputs are isolated thanks to coupling capacitors)
    - Transient analysis (C → 1/sC and L → sL)
    - Steady-state analysis (C: open circuit, L: short circuit)
  - AC Analysis (DC sources: V → short circuit, I → open circuit)
    - Mid-frequency response (C: short circuit, L: open circuit)
    - Low-frequency response (C → 1/sC and L → sL)
    - High-frequency response (Extra C and L → 1/sC and sL)

- Using LTspice, we have options for
  - Transient Analysis (.tran)
  - AC Analysis (.ac)
  - DC Sweep Analysis (.dc)
  - Noise Analysis (.noise)
  - DC Transfer Function Analysis (.tf)
  - DC Operating Point Analysis (.op)
  - Parametric Analysis (.step)
  - Temperature Analysis (.temp)
  - Monte Carlo Analysis (.mc)

https://spiceman.net/ltspice-analysis-type/

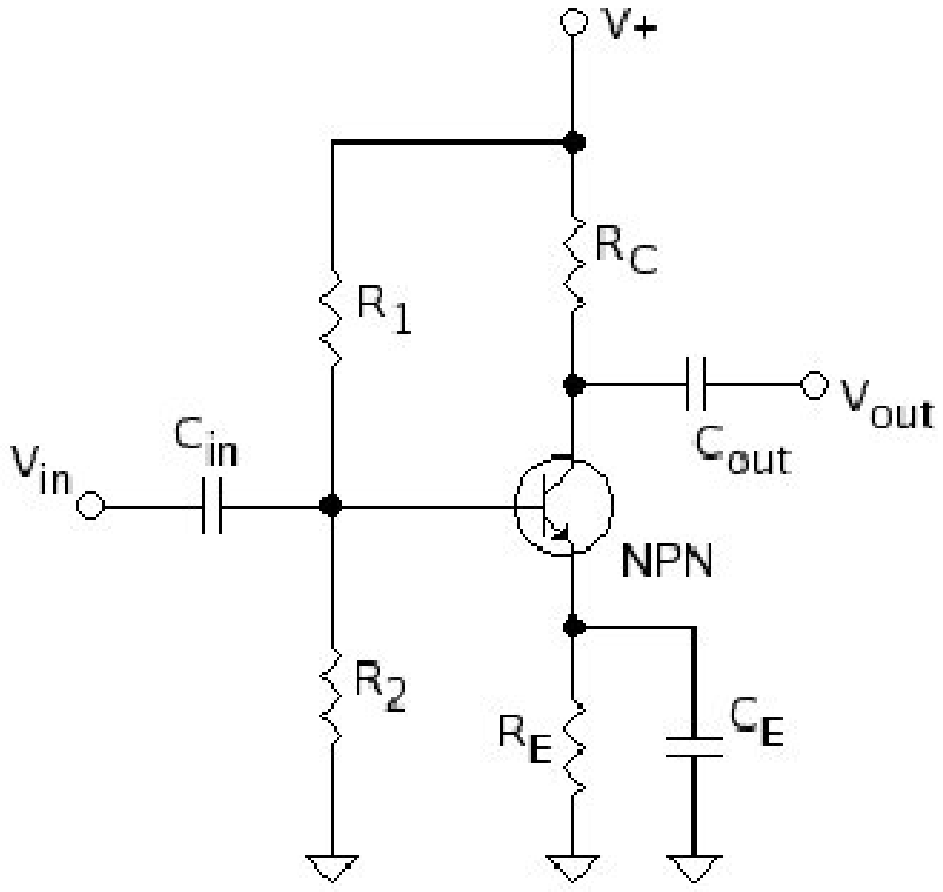
Circuit Analysis by Hand (1)

- DC Analysis
  - To find the steady-state values before the change
    - All C are open circuit and L are short circuit
    - Calculate initial voltage for Cs and initial currents for Ls
  - To find the transient behaviors just after the change
    - All C and L are available (maybe, replacing C → 1/sC and L → sL)
    - Construct voltage equations for C
    - Construct current equations for L
  - To solve the constructed equations, use steady-state values of long time after the change
    - All C are open circuit and L are short circuit
    - Determine final voltage for Cs and final currents for Ls (t→ inf)



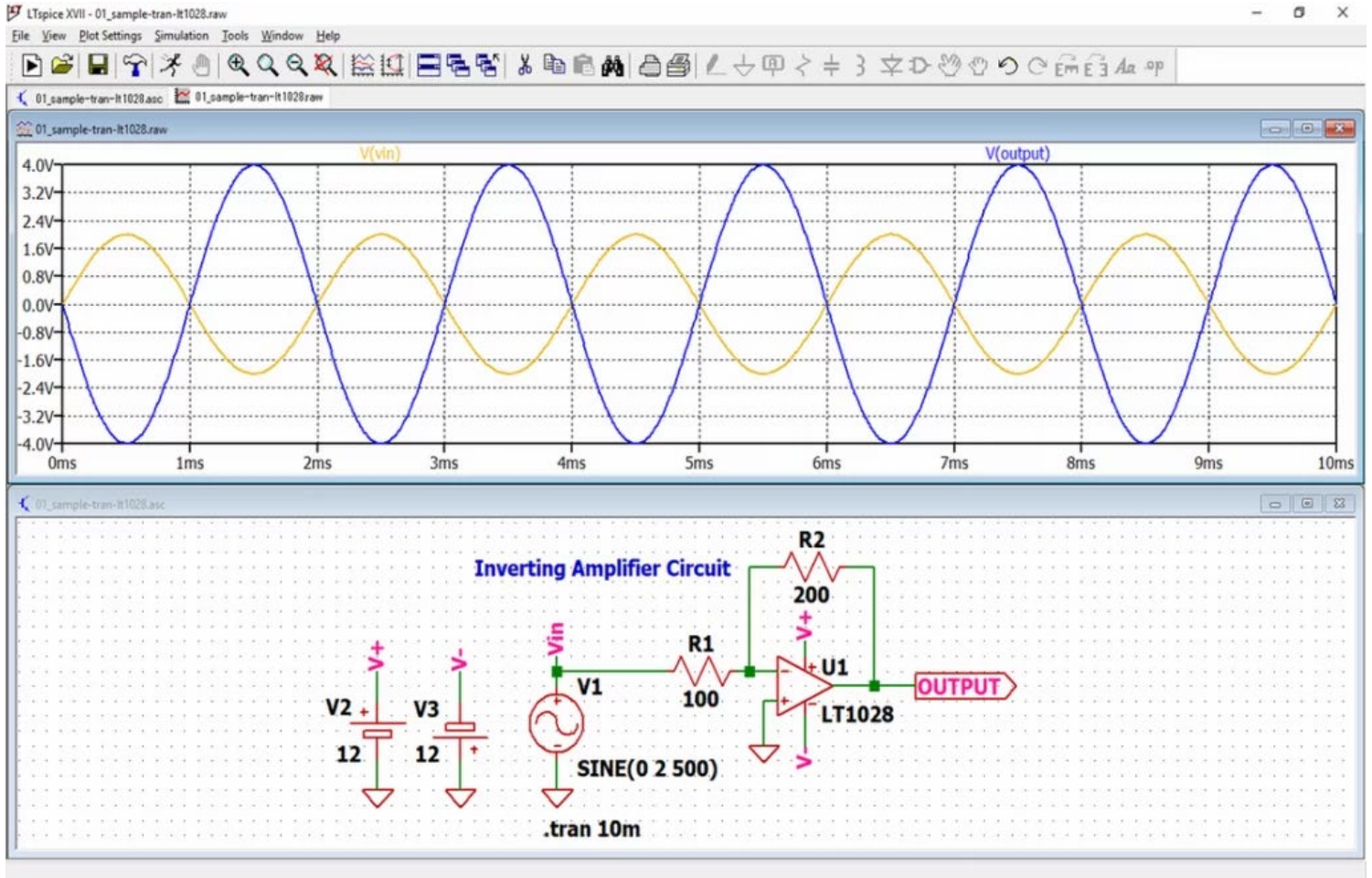
Circuit Analysis by Hand (2)

- DC Analysis
  - To find the operating (quiscent) point
  - All C are open circuit and L are short circuit
- AC Analysis (DC sources: V → short circuit, I → open circuit)
  - Eliminate DC effects first:
    - DC voltage sources are short circuit (V+ is connected to the ground)
    - DC current sources are open circuit
    - Replace the equivalent circuits for known-devices (for example NPN transistor)
  - Mid-frequency response:
    - Replace all C: short circuit, L: open circuit
    - Analyze the circuit
  - Low-frequency response:
    - Replace C → 1/sC and L → sL
    - Analyze the circuit
  - High-frequency response:
    - Replace all C: short circuit, L: open circuit
    - Replace extra C → 1/sC and L → sL
    - Analyze the circuit



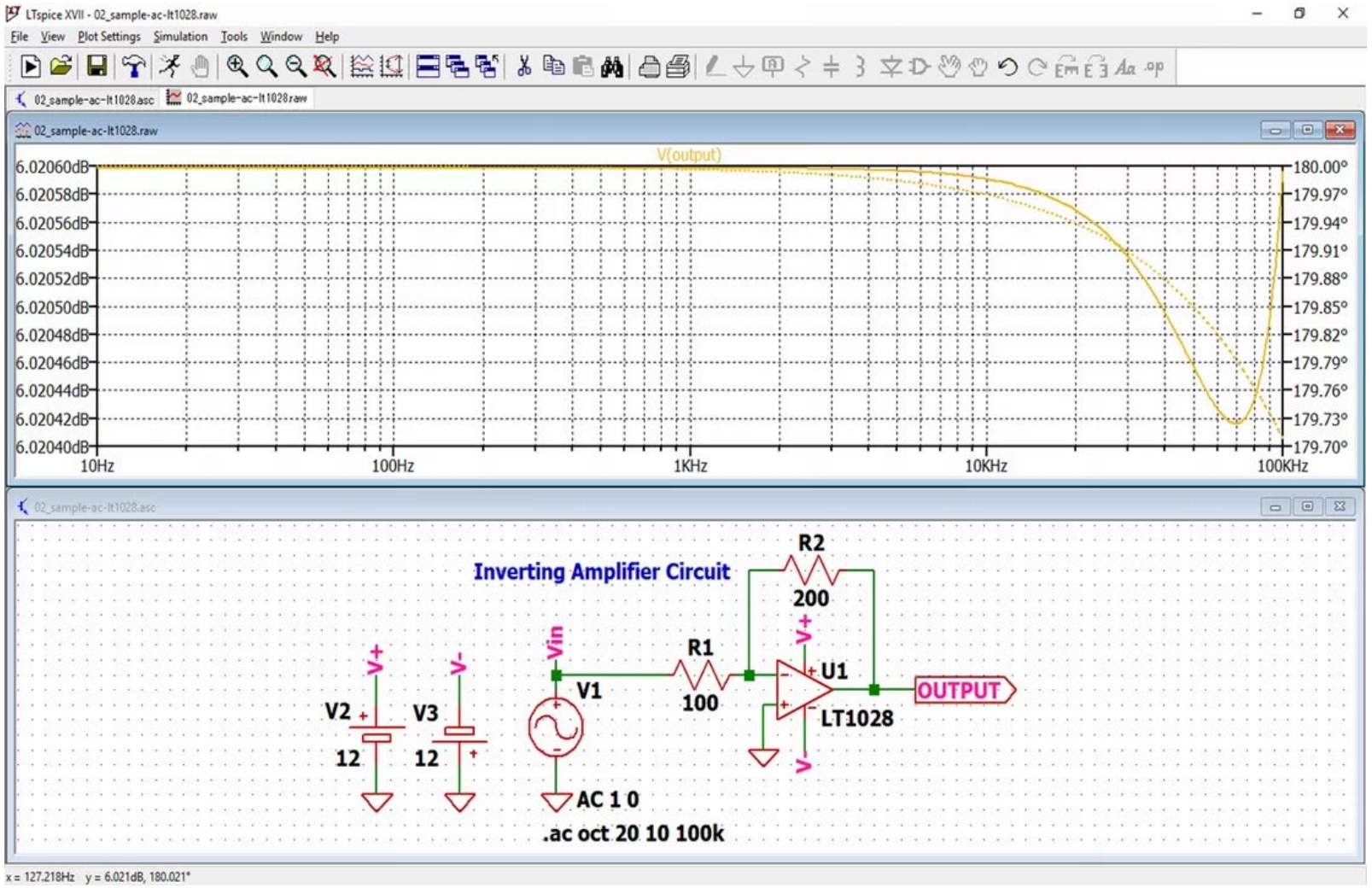
Circuit Analysis using LTspice (1): Transient Analysis

- Transient analysis(.tran) analyzes the temporal change of each voltage and current when the input signal is input to the electronic circuit.
- This is useful when we want to see transient changes just after turn off or on a switch before all the voıltages and currents become stable.
- SINE(0 2 500): V1=0+2*sin(2*pi*500*t) volts
- .tran 10m: simulation stops when time reaches 10 ms



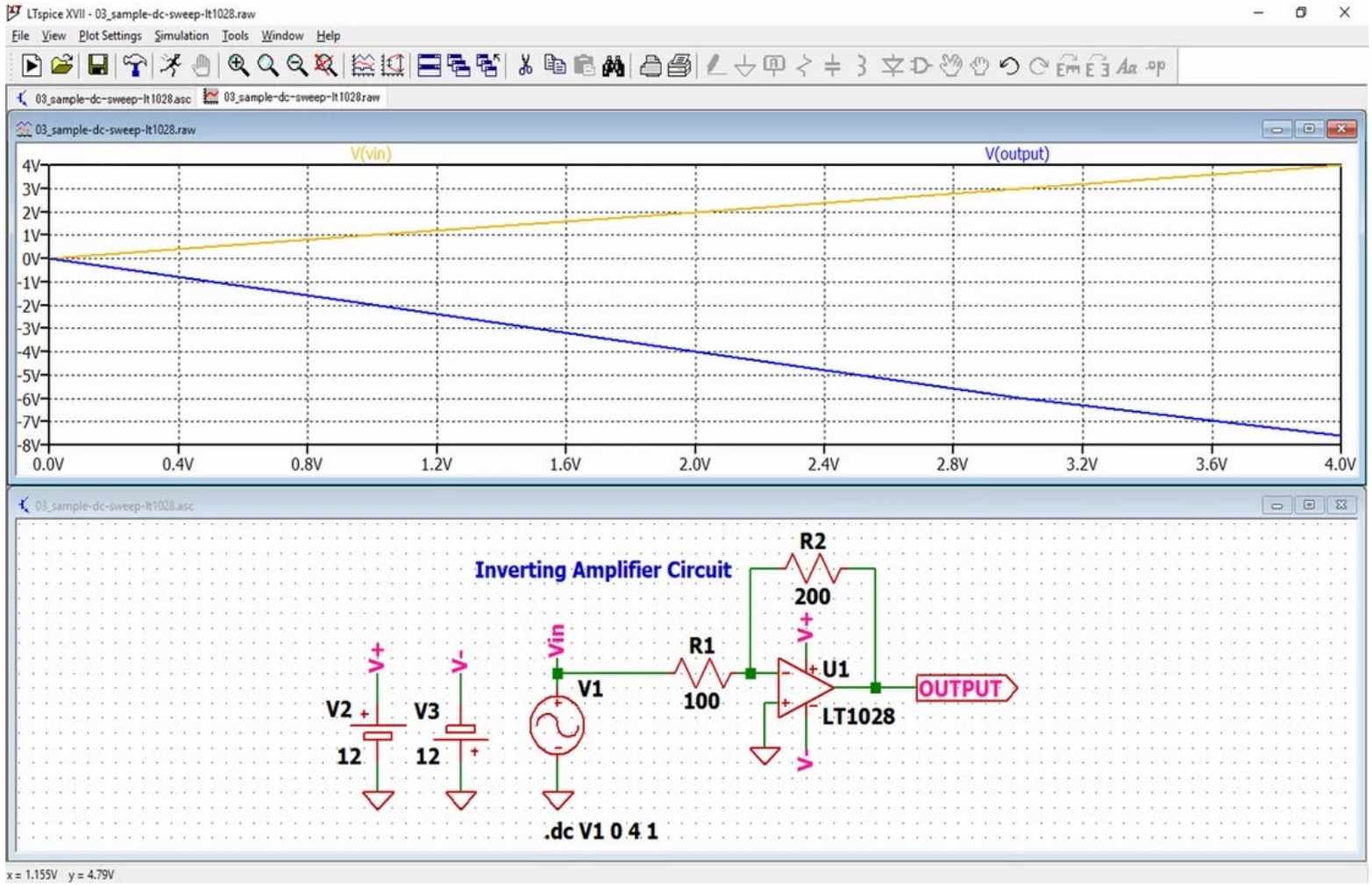
Circuit Analysis using LTspice (2): AC Analysis

- AC analysis(.ac) analyzes the frequency characteristics of electronic circuits.
- AC 1 0: V1 has the amplitude of 1V and the phase of 0 radians
- .ac oct 20 10 100k: start frequency is 10 Hz, stop frequency is 100 kHz, with 20 frequency points per octave sweep



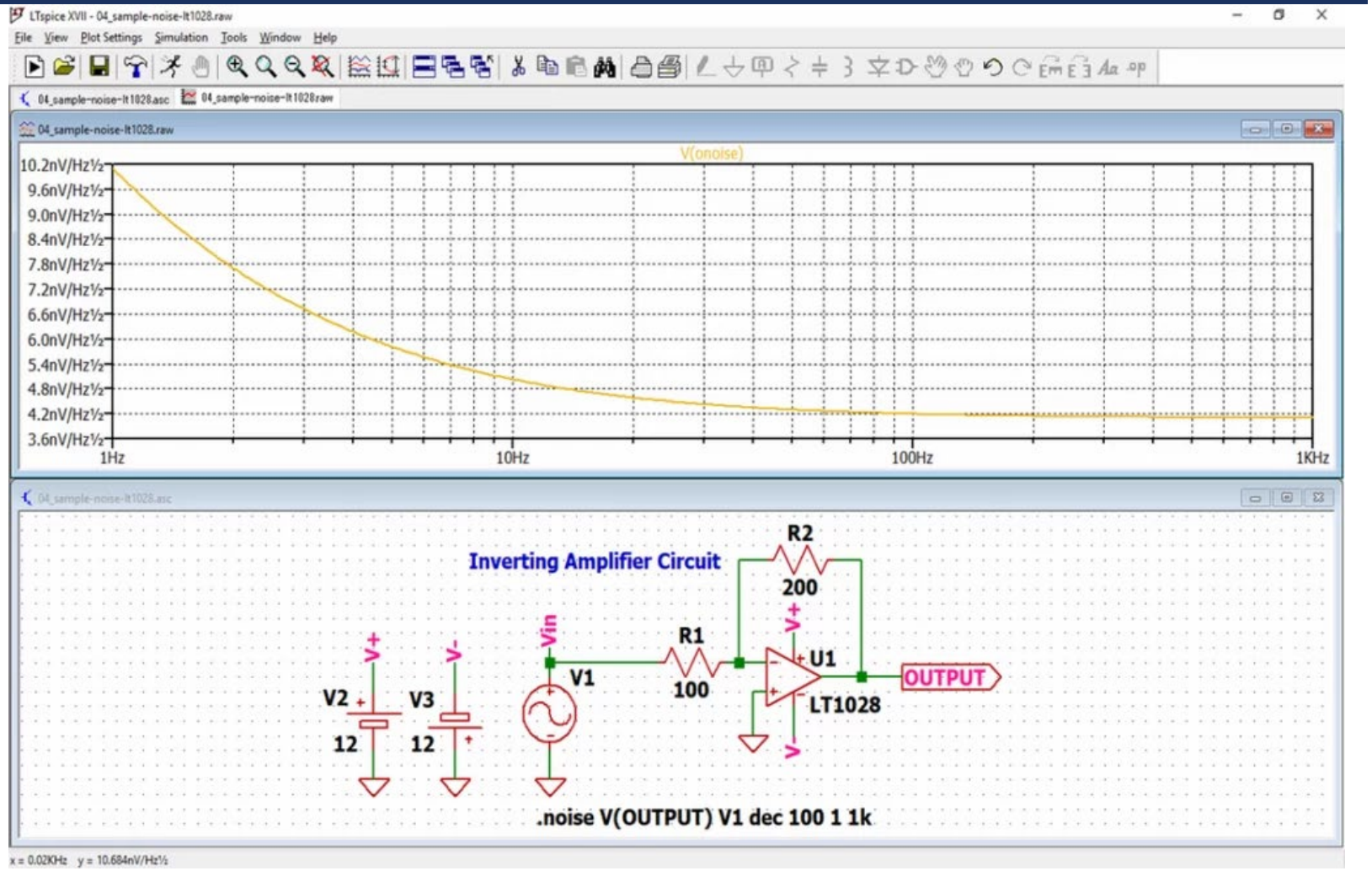
Circuit Analysis using LTspice (3): DC Sweep Analysis

- DC sweep analysis(.dc) sweeps and analyzes the DC voltage of the input signal of the electronic circuit.
- It is used to analyze DC characteristics of diodes, transistors, and op amps.
- .dc V1 0 4 1: repeat simulation from 0 V to 4 V with 1 V linear increments for values of V1



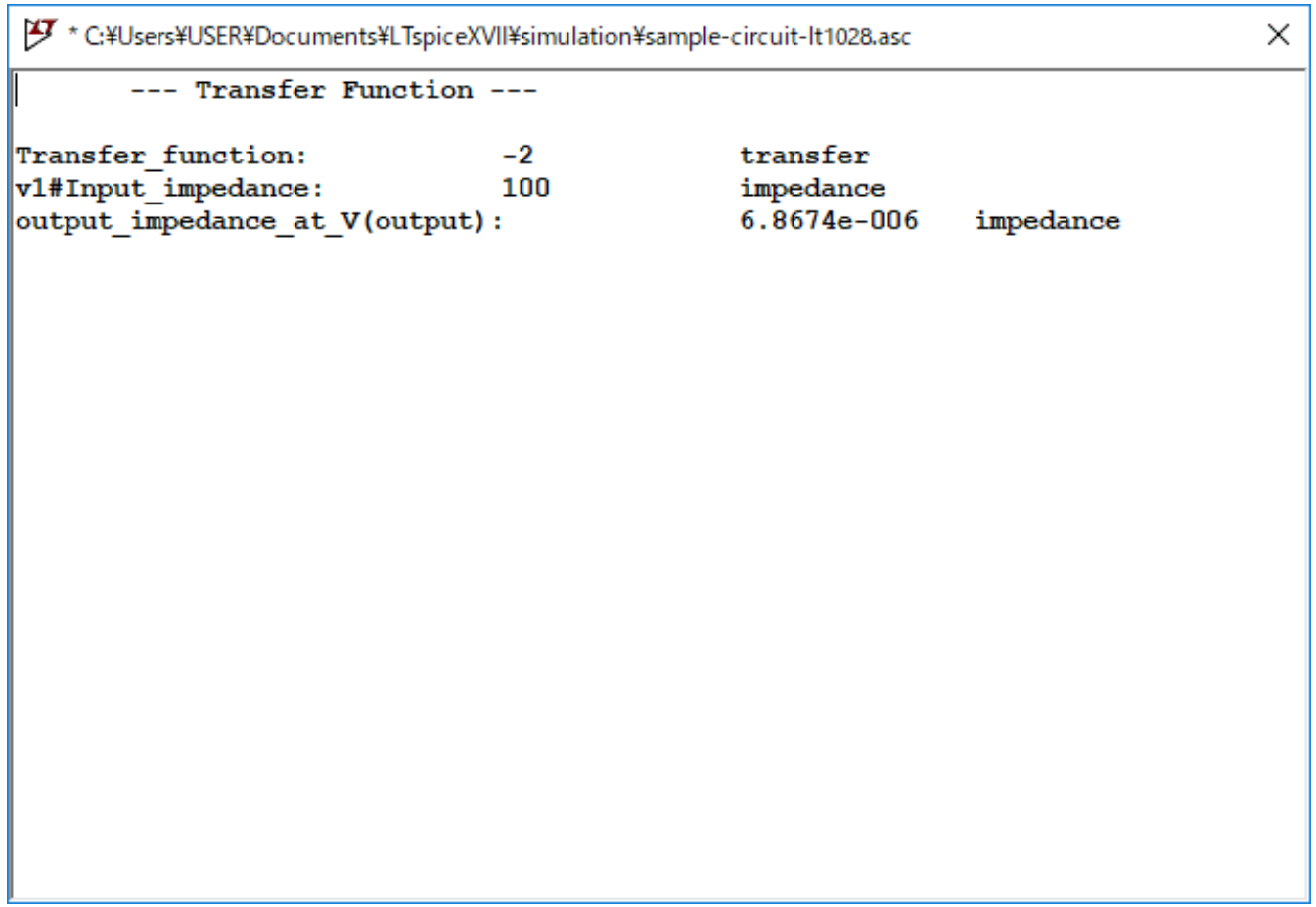
Circuit Analysis using LTspice (4): Noise Analysis

- Noise analysis(.noise) analyzes the frequency characteristics of electronic circuit noise.
- .noise V(OUTPUT) V1 dec 100 1 1k: repeat simulation from 1 Hz to 1 kHz with 100 points per decade where the input is V1 and the output V(OUTPUT)
- You can see the frequency characteristics of the noise on the output of the op-amp.



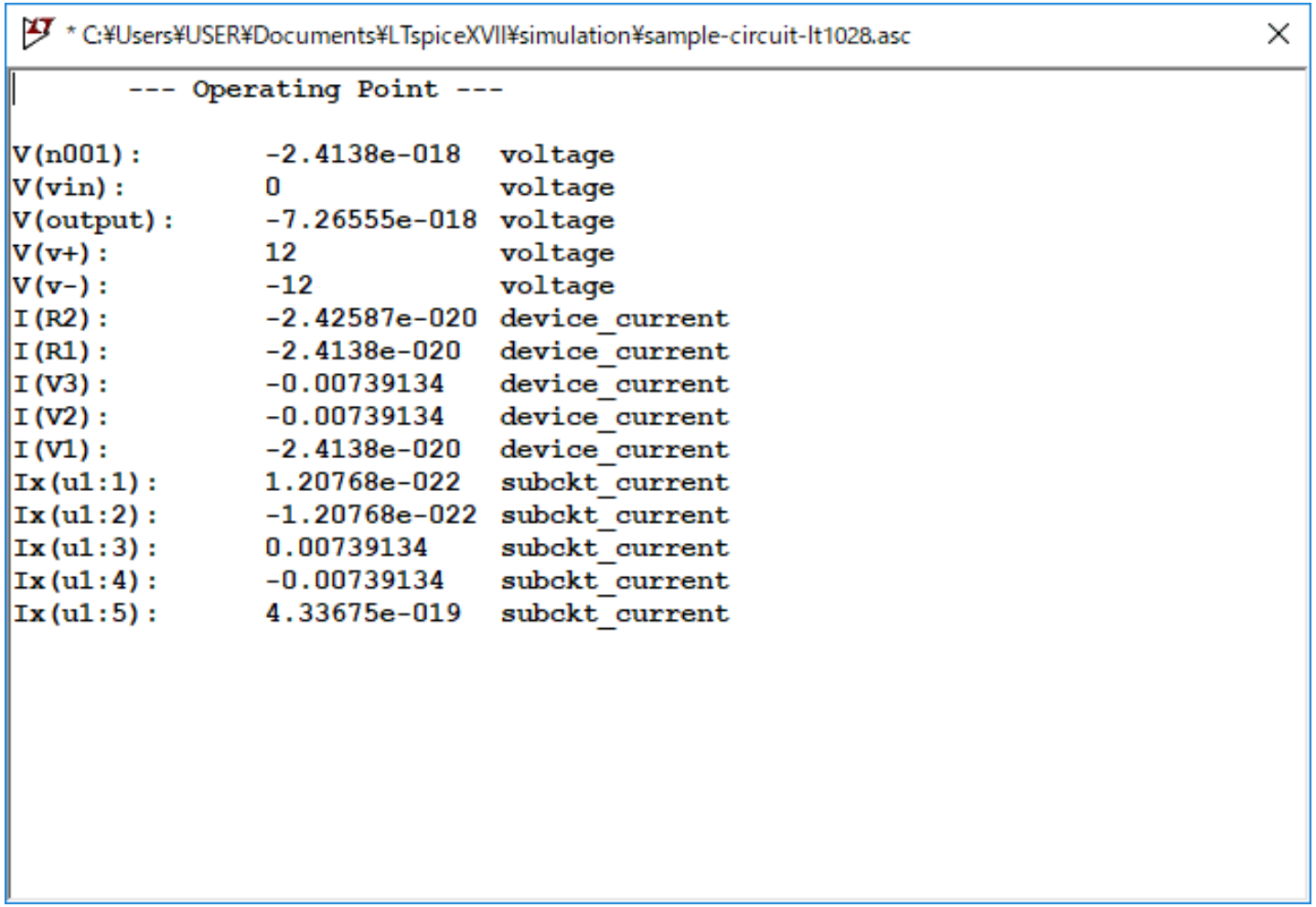
Circuit Analysis using LTspice (5): DC Transfer Function Analysis

- DC transfer function analysis (.tf) defines the input and output in the electronic circuit to calculate the transfer function of a small DC signal, and calculates the output/input conversion ratio, input impedance, and output impedance.
- .tf V(OUTPUT) V1: Determine the transfer function from V1 to V(OUTPUT).
- From the text output:
  - V(OUTPUT) / V1 = -2
  - Zin = 100 ohms
  - Zout = 6.8674 micro-ohms



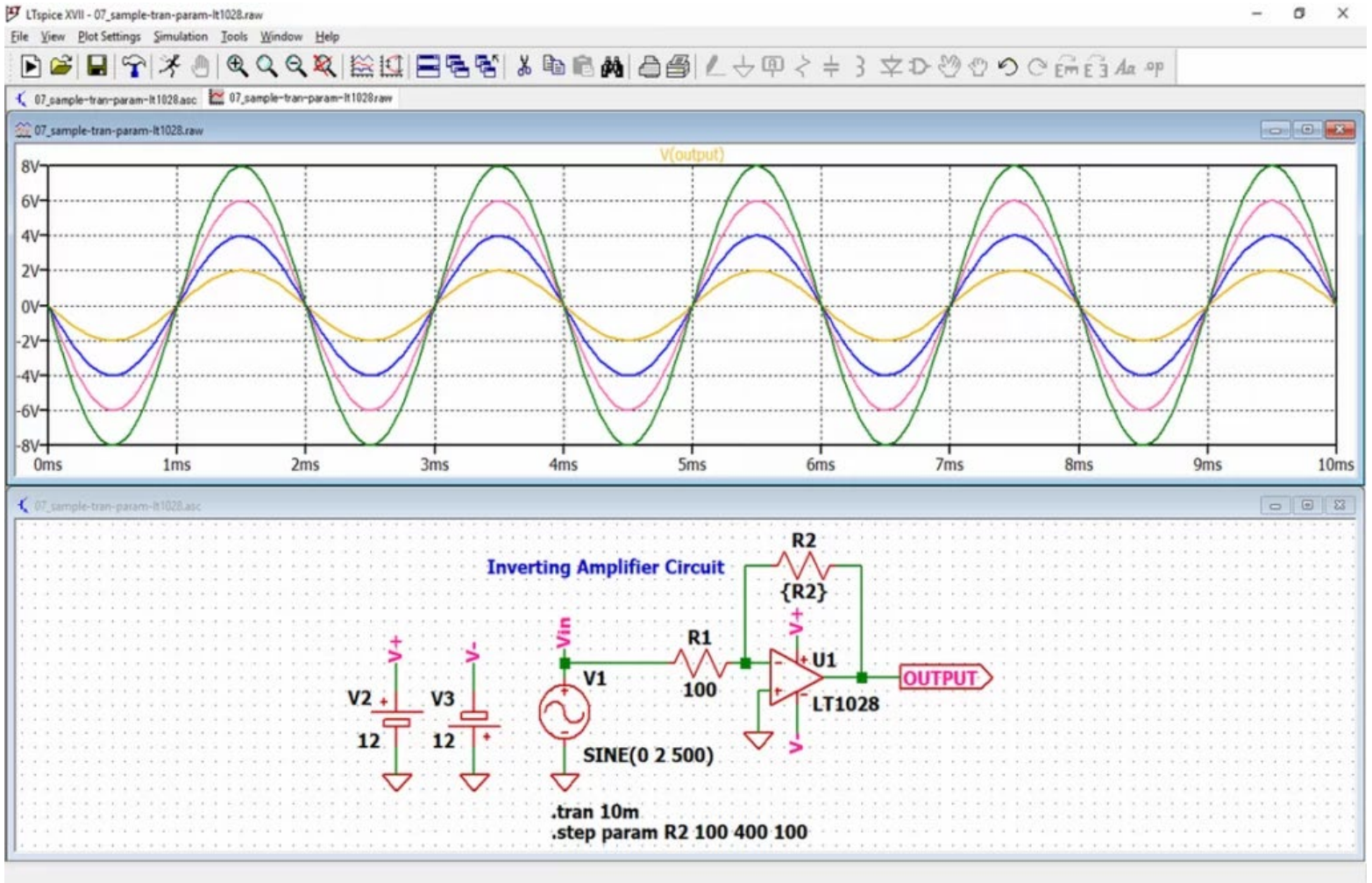
Circuit Analysis using LTspice (6): DC Operating Point Analysis

- DC operating point analysis(.op) calculates the DC voltage and current of each node in the steady state of the electronic circuit.
- .op: NO PARAMETER ☺
- The number to be calculated varies depending on the circuit size. In this circuit, 5 voltage values and 10 current values are calculated.



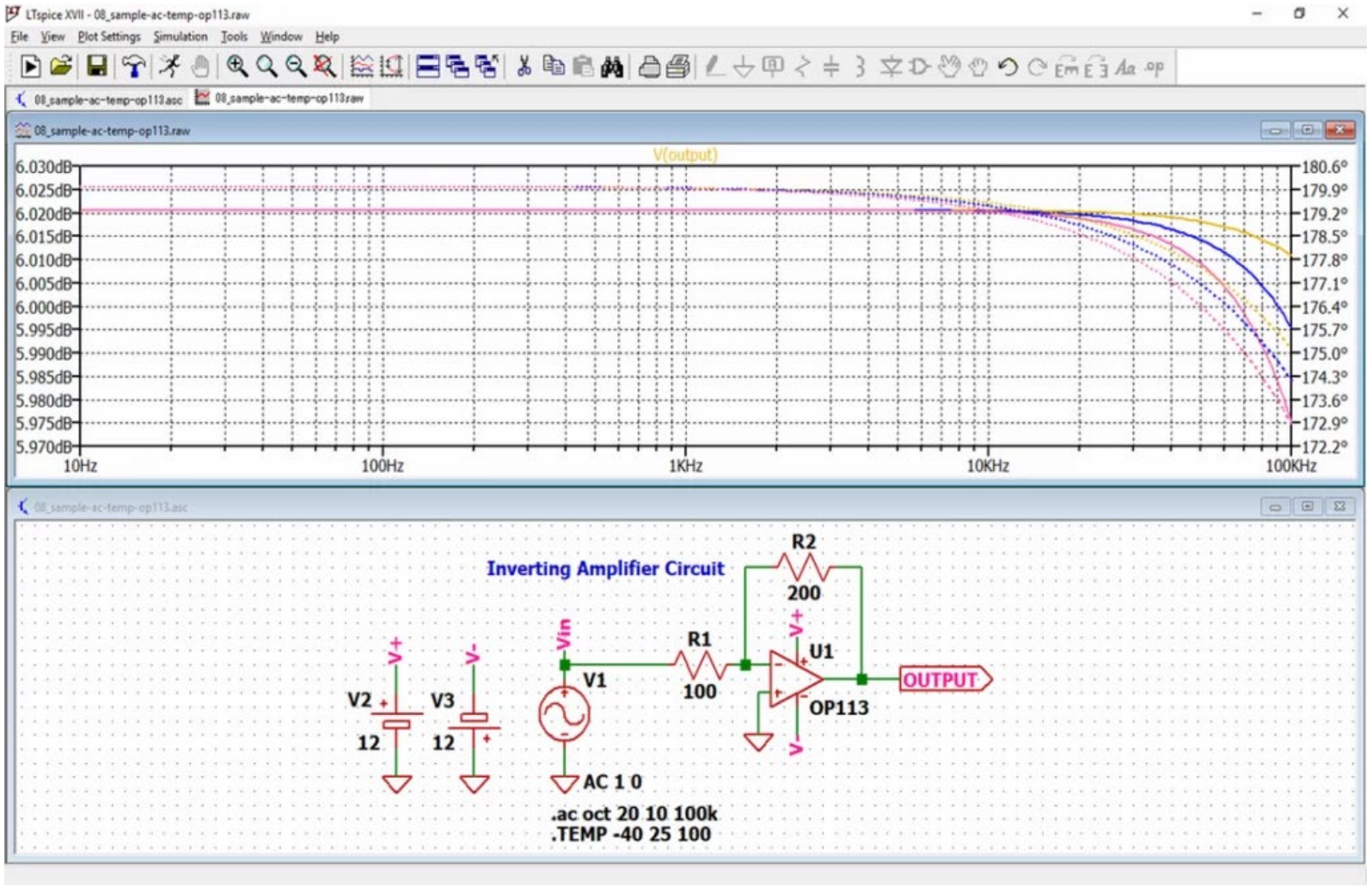
Circuit Analysis using LTspice (7): Parametric Analysis

- Parametric analysis(.step) analyzes while changing parameters such as resistance, capacitor, inductor, and power supply of electronic circuit.
- It is performed simultaneously with transient analysis, AC analysis, DC sweep analysis, etc.
- .step param R2 100 400 100: Repeats simulation for R2 values from 100 ohms to 400 ohms with 100 ohms increments.
- Make sure that the parameter of R2 is {R2}.



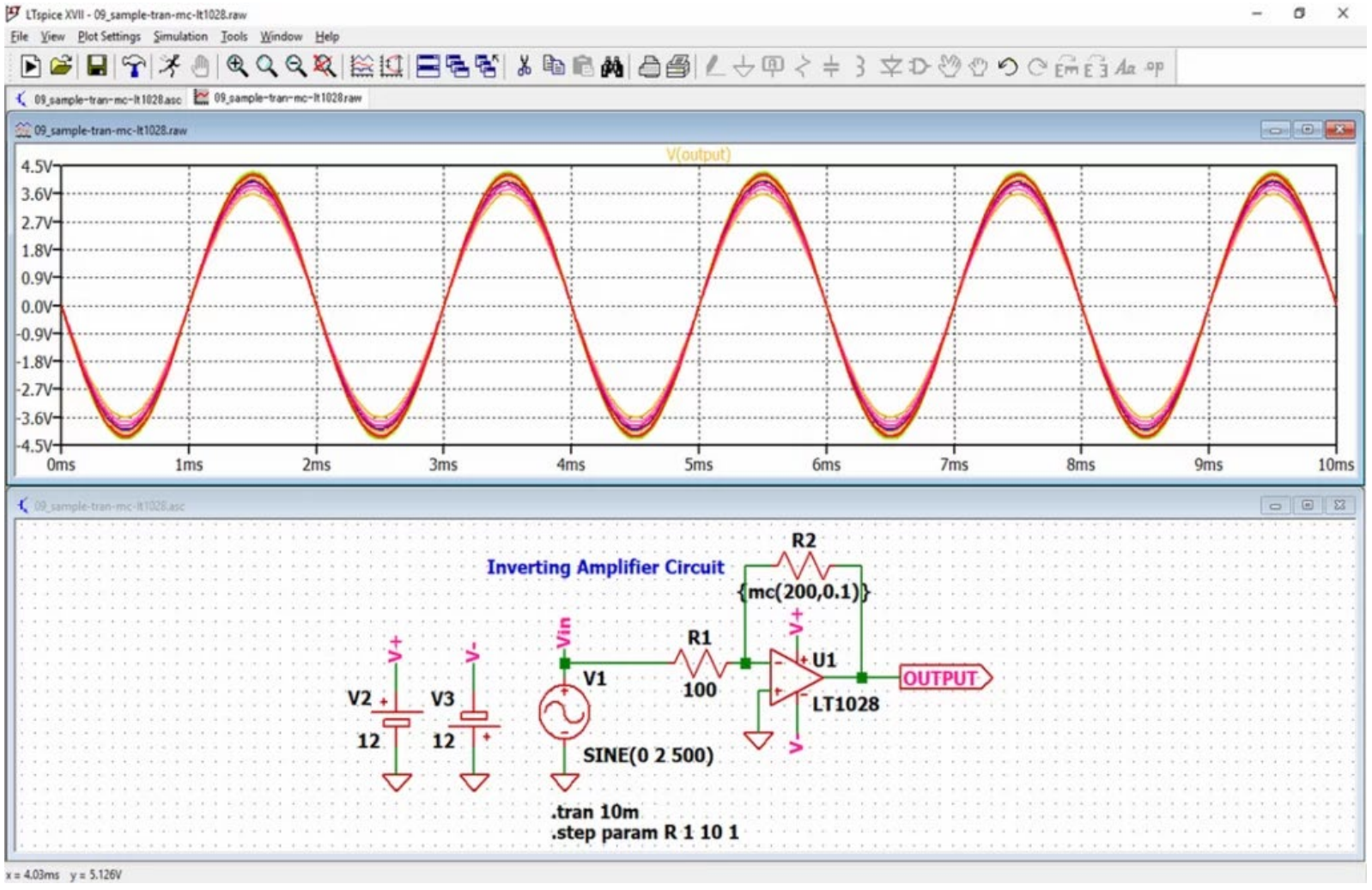
Circuit Analysis using LTspice (8): Temperature Analysis

- Temperature analysis(.temp) analyzes the effects of temperature characteristics of semiconductors such as op-amps and transistors in electronic circuits.
- It is performed simultaneously with transient analysis, AC analysis, DC sweep analysis, etc.
- .TEMP -40 25 100: Repeats simulation for the temperatures of -40, +25, +100 C degrees.



Circuit Analysis using LTspice (9): Monte Carlo Analysis

- Monte Carlo analysis (mc) analyzes the effects of errors due to electronic circuit parts.
- It is performed simultaneously with transient analysis, AC analysis, DC sweep analysis, etc.
- .tran 10m: It is a transient analysis that sops at 10 ms.
- .step param R 1 10 1: Repeats simulation from 1 ohms to 10 ohms with 1 ohms increment of R values. Since there is no R in the circuit, it repeats the simulation 10 times with randomly selected R2 values thanks to mc command ☺
- Make sure that the parameter of R2 is «{mc (200, 0.1)},» which means the resistance value of R2 is set to perform Monte Carlo analysis with an error of 10%.



Thanks for listening ☺

YALÇIN İŞLER

Assoc. Prof.