ECE 3020
Introduction to Electronics

Section 9: Circuit Simulation Software (*TopSpice*)

Spring 2024

Tawfiq Musah, Assistant Professor

Dept. of Electrical & Computer Engineering

The Ohio State University

Acknowledgement

◆ **The instructor would like to acknowledge and thank Prof. Nima Ghalichechian and Prof. Asimina Kiourti for kindly providing these TopSpice notes.**

Virtual Machine/Lab Access

◆ **For students who don't have access to a Windows device, access to TopSpice demo has been arranged (To be confirmed):**

➢ **1. Virtual: ECE-HNC217492D, ECE-HNC217493D (from https://labstatus.coeit.osu.edu)**

➢ **2. Physical: ECE-HNC217487D, ECE-HNC217488, and ECE-HNC217499D from the front section of CL 260.**

What is SPICE?

◆ **SPICE:**
Simulation **P**rogram with **I**ntegrated **C**ircuit **E**mphasis

◆ **It is a general-purpose circuit simulator for analog electronics – used to check the integrity of circuit designs and to predict circuit behavior.**

➢ *In layman's terms: SPICE is a program that simulates electronic circuits on your PC.*

➢ *For example, circuits may contain resistors, capacitors, inductors, mutual inductors, independent or dependent voltage/current sources, transmission lines, switches, diodes, transistors, etc.*

➢ *You can view any voltage or current waveform in your circuit.*

◆ **Developed at the University of California, Berkeley during the mid-70s as a way to test and tweak circuit designs before the expensive fabrication process.**

Why use SPICE?

◆ **Acts like a "virtual" breadboard – much cheaper and faster to test than using an actual breadboard.**

◆ **Great tool for learning electronics.**
  ➢ **You can increase your understanding of circuits as you play and tinker with them, e.g., change a resistor value and see the effect on a circuit in seconds.**

Inverting Op Amp Breadboard
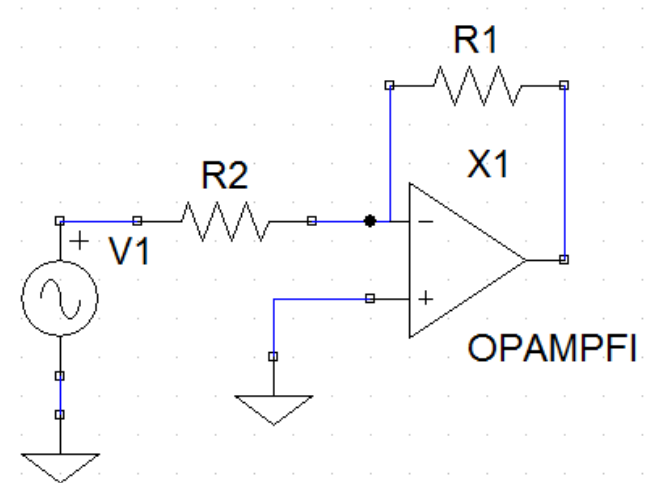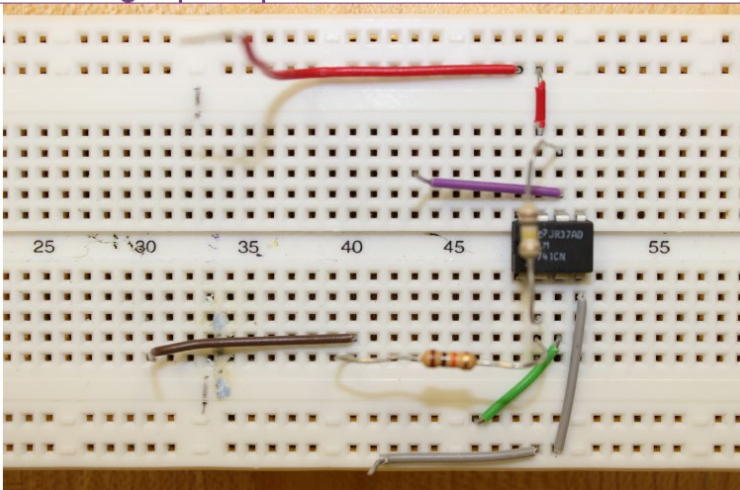
Software Packages that Implement SPICE

◆ **Many software packages available that implement SPICE**

  ➢ **Different graphical user interfaces**

  ➢ **Different model libraries**

  ➢ **Different interfaces to other tools**

  ➢ **etc.**

  ➢ **…and eventually --- different cost**

◆ **We will be doing demonstrations using <u>TopSpice</u> from Penzar Development (http://www.penzar.com/)**

  ➢ **They have a free demonstration version**

  ➢ **Fully functional but limits on circuit size and reduced model libraries**

  ➢ **Windows platform**

TopSpice by Penzar

THE OHIO STATE UNIVERSITY
ELECTRICAL & COMPUTER ENGINEERING
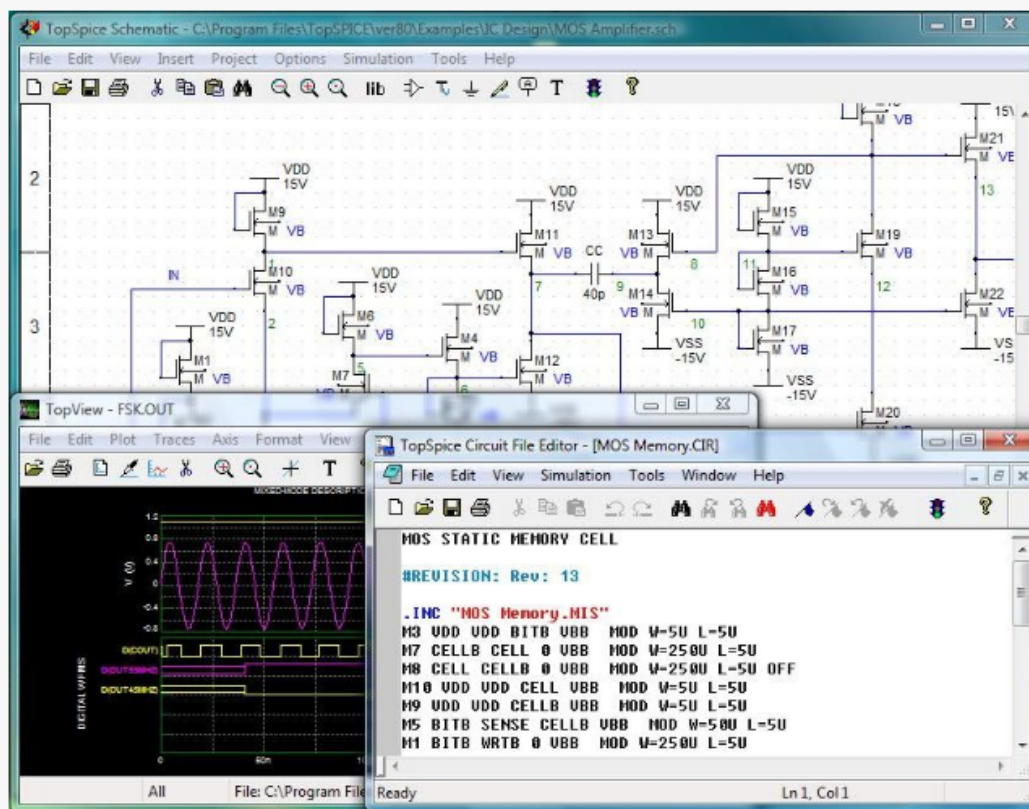
SPICE – Types of Analysis

◆ **DC Analysis:** Determines the DC operating point of the circuit with inductors shorted and capacitors opened.

◆ **AC Small-Signal Analysis:** Computes the AC output variables as a function of frequency. The program first computes the DC operating point of the circuit and determines linearized, small-signal models for all the non-linear devices in the circuit. The resultant linear circuit is then analyzed over a user-specified range of frequencies. **(i.e., view voltages and currents vs. frequency)**

◆ **Transient Analysis:** Computes the transient output variables as a function of time over a user-specified time interval. **(i.e., view voltages and currents vs. time)**

◆ **Pole-Zero Analysis:** Computes the poles and/or zeros in the small-signal AC transfer function.

◆ **Sensitivity Analysis:** Calculates the difference in an output variable by perturbing each parameter of each device independently.

◆ **Noise Analysis:** Calculates the noise contributions of each device to the output port.

◆ **Small-Signal Distortion Analysis:** Computes steady-state harmonic and intermodulation products for small input signal magnitudes.

How does SPICE work?

1. **Describe your circuit** in a text file ("**.cir**" extension) called a netlist **OR** draw the circuit using graphical symbols on a schematic page. --- Number each node, Label each component and give it a value.

2. **Decide on the type of analysis you want performed.** (AC, Transient, DC, noise, etc.)

3. **Run the simulation.** SPICE reads the netlist and then performs the requested analysis: (e.g., AC, DC, or TRANSIENT RESPONSE). The results are stored in a text output file ("**.out**" extension) or a binary data file.

4. **View the results** of the simulation in a text output file (".out") using a text editor. Most SPICE programs provide a graphical viewer to plot the waveforms stored in the binary data file.

Inside a Typical SPICE File

◆ **The heart of your SPICE file is the netlist, which is simply a list of components and the nets (or nodes) that connect them together).**

Example for RC Low-Pass-Filter:

nodes


```
LPFILTER.CIR   SIMPLE RC LOW-PASS FILTER
*
VS       1        0        AC       1       SIN(0VOFF 1VPEAK   2KHZ)
*
R1       1        2        1K
C1       2        0        0.032UF
*
* ANALYSIS
.AC     DEC      5 10 10MEG
.TRAN   5US   500US
*
* VIEW RESULTS
.PRINT  AC       VM(2) VP(2)
.PRINT  TRAN     V(1) V(2)
*
.PROBE
.END
```

AC analysis at 5 points per DECade in the frequency range from 10 Hz to 10 MHz.
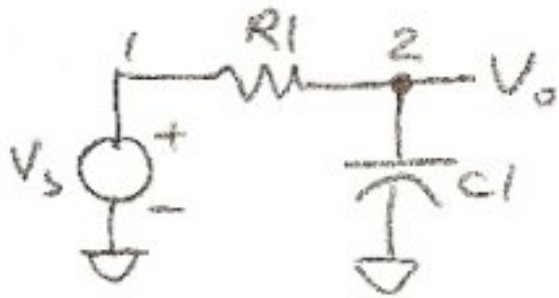
Transient analysis for a duration of 500 us and print out the results at 5 us intervals.

Prints the voltage magnitude and phase at node 2 for the AC analysis and the voltages at nodes 1 and 2 for the time analysis.

TopSpice Installation

STEP 1: Go to Penzar's Website

THE OHIO STATE UNIVERSITY

ELECTRICAL & COMPUTER ENGINEERING

STEP 2: Download TopSpice (Demo Version)

Free Software Downloads

Demo Software For Windows XP/7/8/10

- **TopSpice 8 Demo version 8.85 [ts885demosetup.exe]** (25MB, 2021-2-18)

If using Edge browser, please download it as ZIP file ts885demosetup.zip

To install the product, either run or open the downloaded setup file **ts885demosetup.exe in your computer** and follow the setup program screen prompts.

If you download it as a ZIP file, extract the setup file first.

Do not run the installation setup file directly from this webpage.

DEMO VERSION LIMITATIONS

Simulator limits: number of nodes 64; number of top level transistors and subcircuits 20; number of subcircuit transistors 20; total number of top level components 30 (excluding resistors and capacitors); number of subcircuit definitions (macromodels) 15; maximum data memory usage 10MB.

Schematic Editor limits: schematic files with more than 3 pages cannot be created but they can be viewed and edited. Model encryption utility not included.

Post-processor plot limits: maximum number of simulated data points per trace 32,000.

The demo model library only includes the device models needed to run the demo example files (they are found under Help).

The manuals cannot be printed (they are found under Help).

Penzar Development Home

STEP 3: Install TopSpice on your PC

THE OHIO STATE UNIVERSITY
ELECTRICAL & COMPUTER ENGINEERING
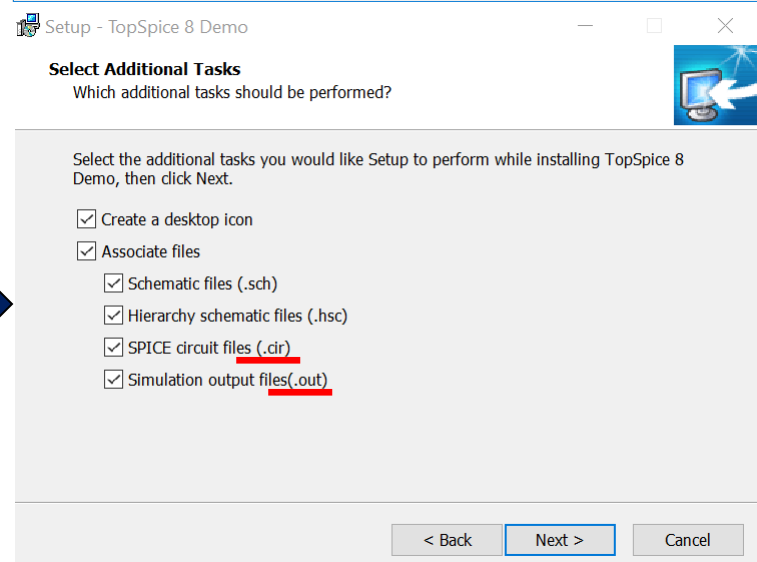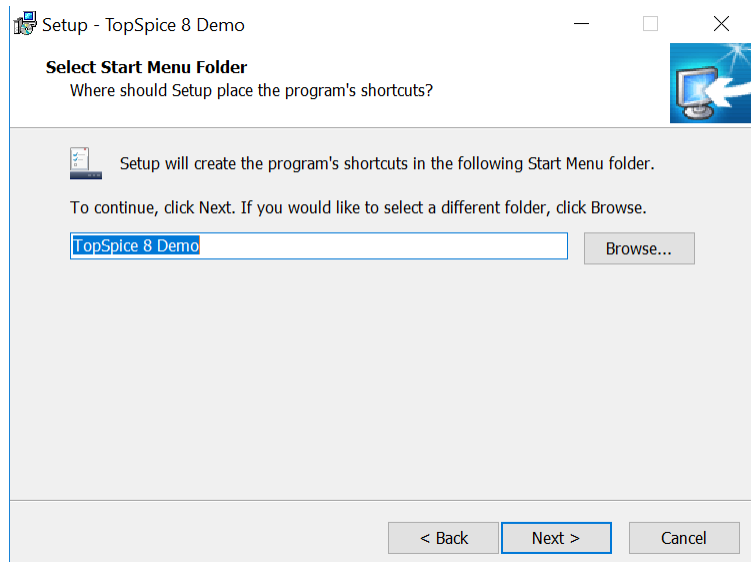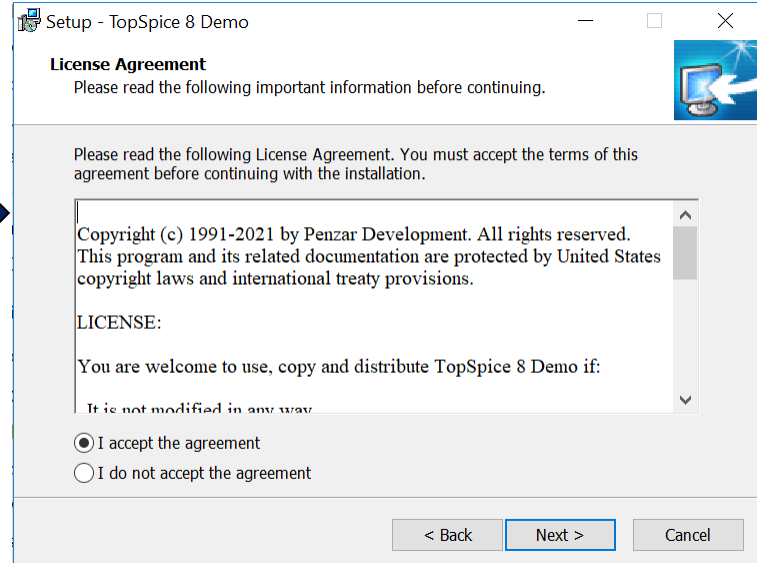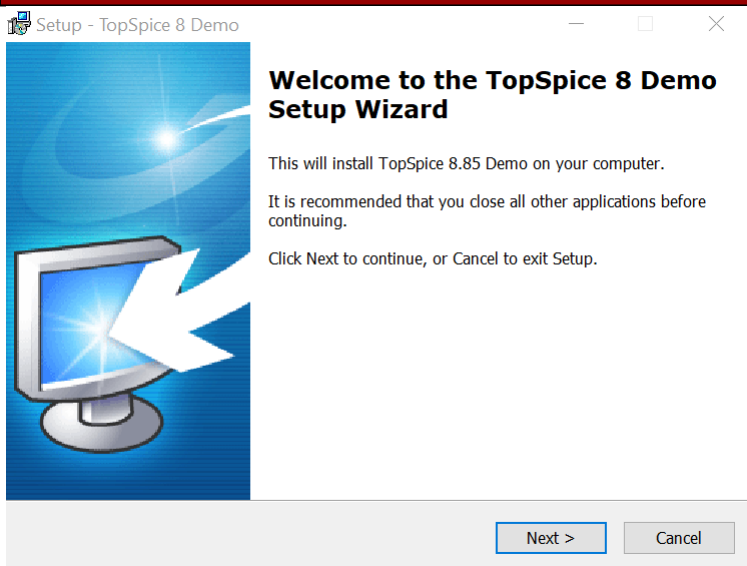
STEP 3: Install TopSpice on your PC
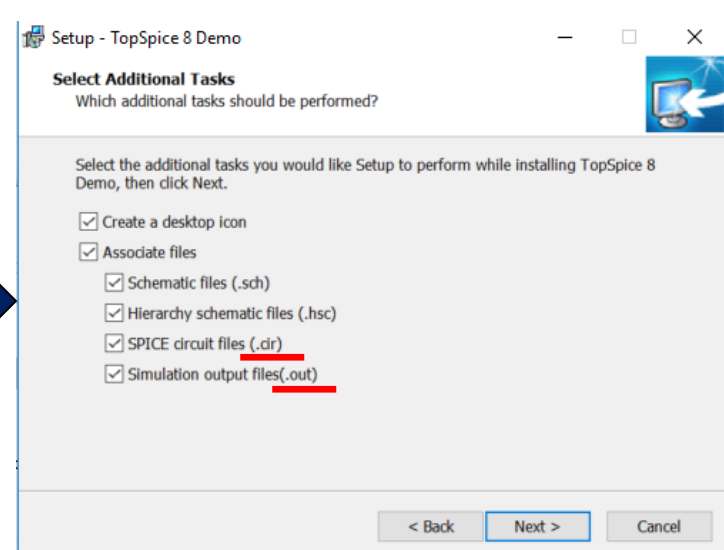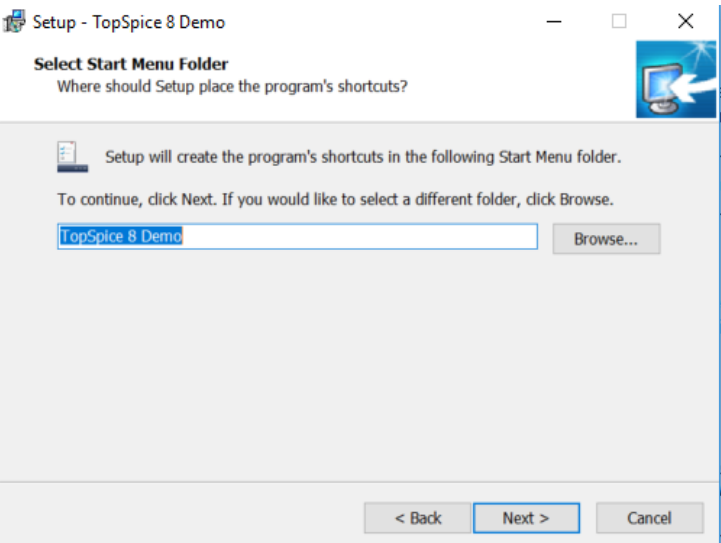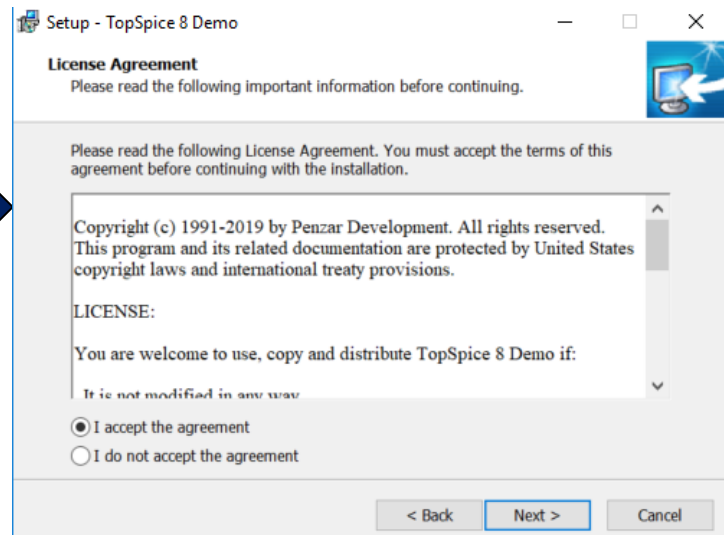
STEP 4: TopSpice Launch Screen

The recommended way to start TopSpice is to open the "TopSpice 8" icon/tile in the Start menu "TopSpice 8" folder or on the Windows desktop.
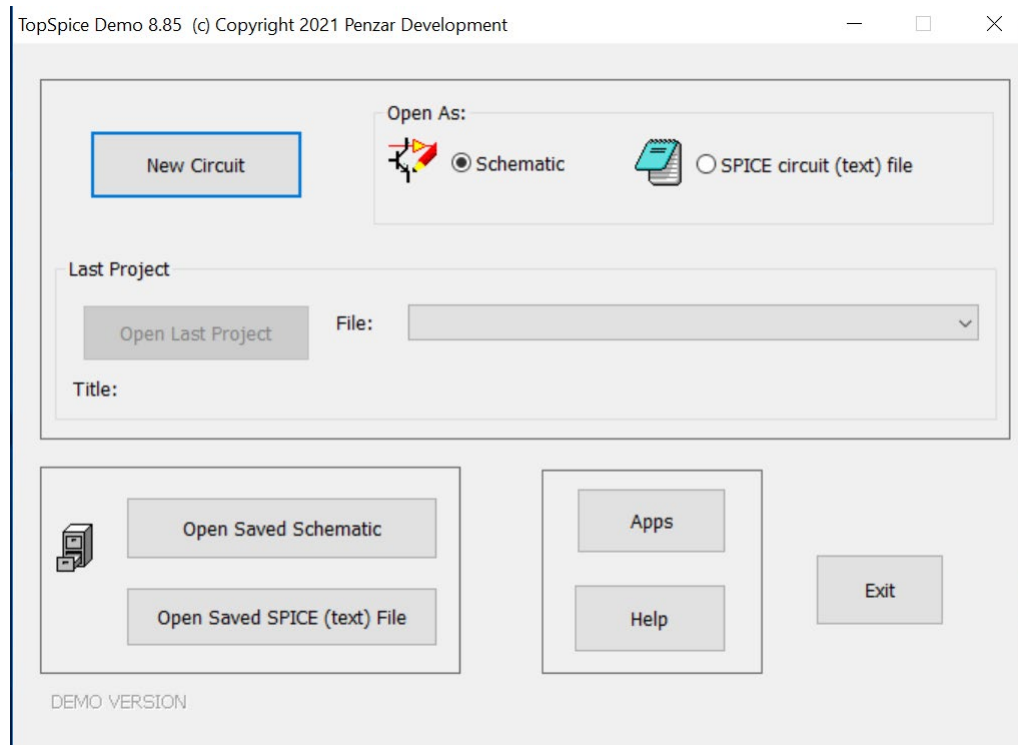
The TopSpice launch screen menu simplifies the task of starting a new or previously saved circuit design project, switching between schematic or circuit file editors, and opening the example circuit files.

TopSpice Example #1
Simulate a simple DC circuit using a Netlist

Example #1 [1 / 2]

1

I1 = 1.2 A

R2 = 5 Ohm

I2 = 1 A

V1 = 12 V

R1 = 10 Ohm

+

−

R3 = 7 Ohm

2

0

New Circuit

Open As:
Schematic
SPICE circuit (text) file

Last Project
Open Last Project   File: C:\Users\Ugur Olgun\Documents\TopSpice\Demo\Examples\O
Title: OP AMP TEST CIRCUIT

Open Saved Schematic

Open Saved SPICE (text) File

Apps

Help

Exit
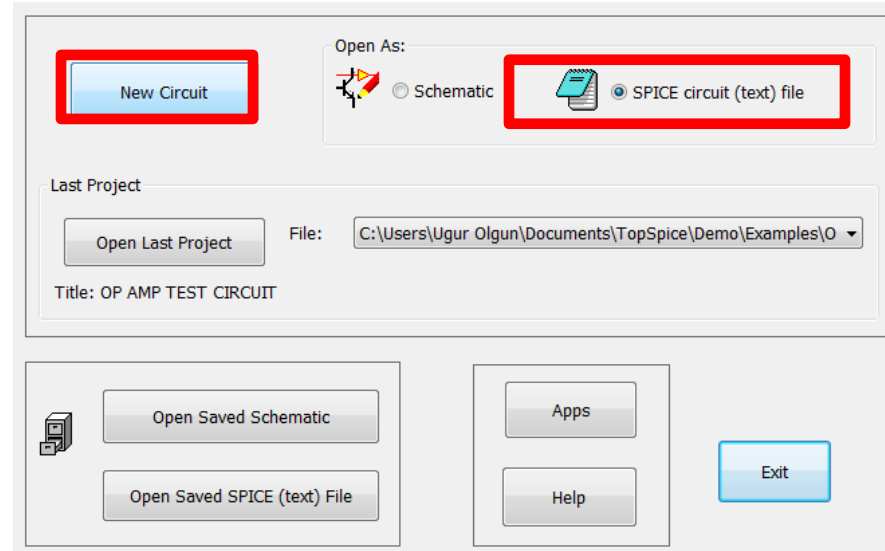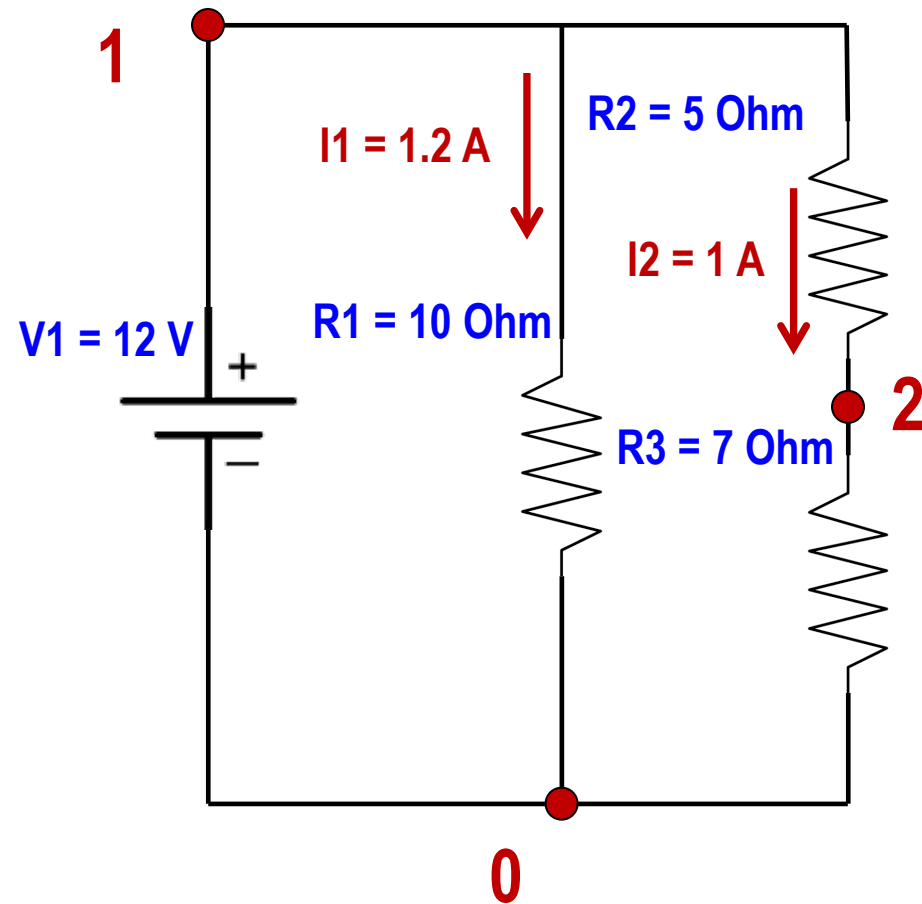
Name
Sample1.CIR

TopSpice Circuit File Editor - [Sample1.CIR]
File   Edit   View   Simulation   Tools   W

```
SMALL SAMPLE CIRCUIT

V1 1 0 DC 12
R1 1 0 10
R2 1 2 5
R3 2 0 7

.END
```

Example #1 [2 / 2]

1

I1 = 1.2 A

R2 = 5 Ohm

I2 = 1 A

V1 = 12 V

R1 = 10 Ohm

+

−

R3 = 7 Ohm

2

0

TopSpice Circuit File Editor - [Sample1.CIR]

File   Edit   View   Simulation   Tools   Window   Help

Run Simulation   F9

Simulator Engine Option   ▶

Plot Output Data   F11

Browse Output File (*.out)   F12

Open Misc. SPICE Commands File (.mis)

SMALL SAMPLE

U1 1 0 DC 12
R1 1 0 10
R2 1 2 5
R3 2 0 7

.END

TopView

No data found in this file

Do you want to browse .OUT file?

Yes   No

NODE	VOLTAGE
V(1)	1.20000000E+01
V(2)	7.00000000E+00

SOURCE	CURRENT
I(V1)	−2.20000000E+00

RESISTOR	CURRENT
I(R1)	1.20000000E+00
I(R2)	1.00000000E+00
I(R3)	1.00000000E+00

TopSpice Example #2
Simulate an IDEAL inverting amplifier configuration using a circuit schematic

$$\frac{v_O}{v_I} = -\frac{R_2}{R_1}$$

(5) $i_2 = i_1 = \frac{v_I}{R_1}$

(3) $i_1 = \frac{v_I}{R_1}$

(4) 0

(1) 0 V

(2) $v_I = 0$ (Virtual ground)

(6) $v_O = 0 - \frac{v_I}{R_1} R_2$

$v_O = -\frac{R_2}{R_1} v_I$

Example #2 [1 / 7]

Insert part

Insert wire

Example #2 [2 / 7]

NOTE: ideal op-amp model
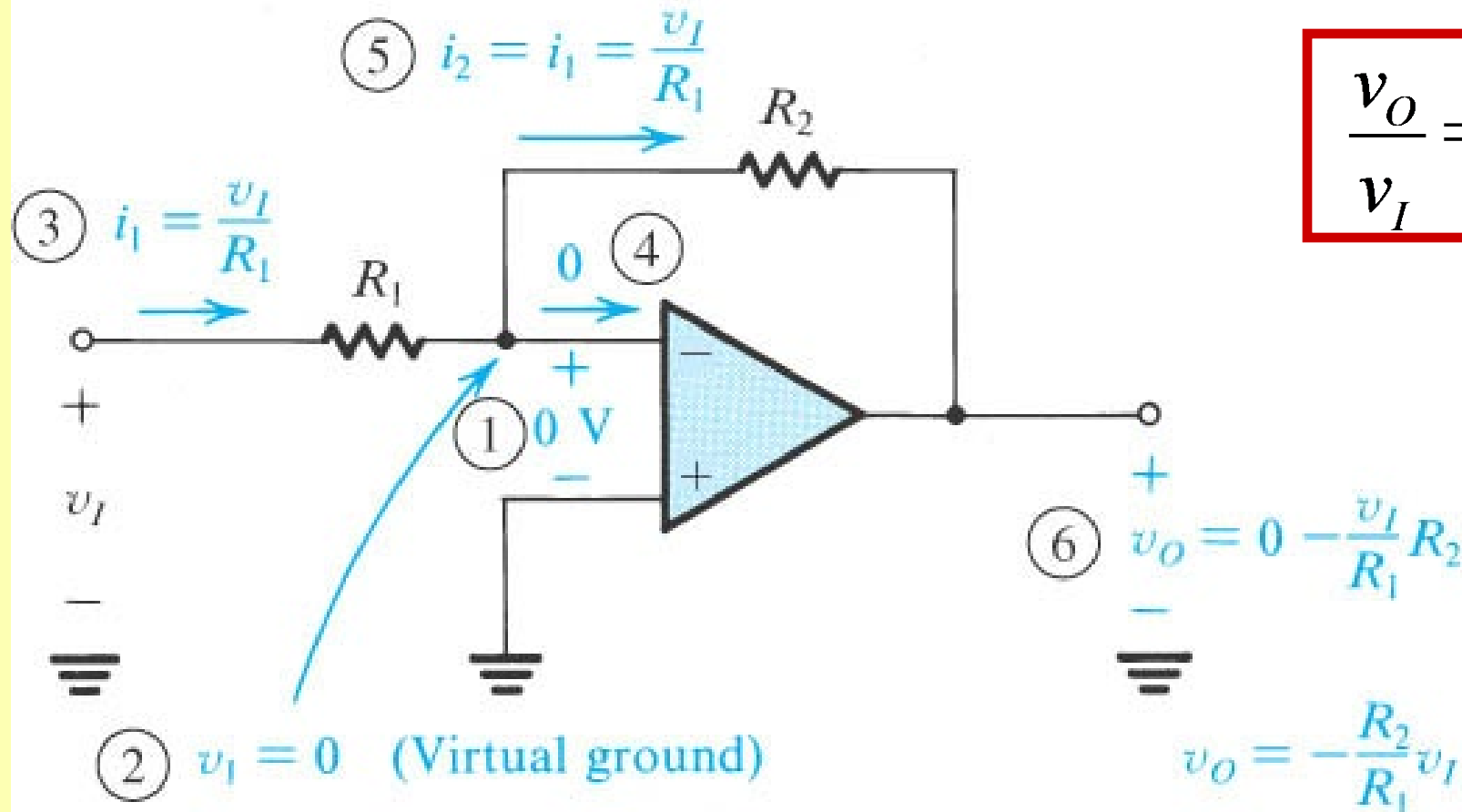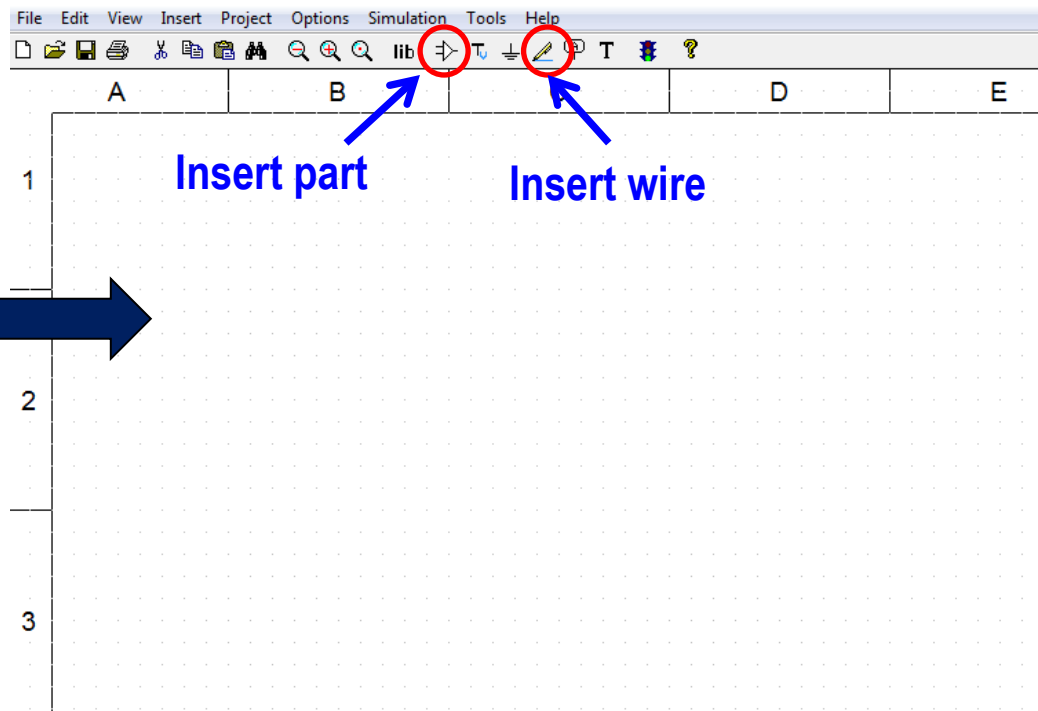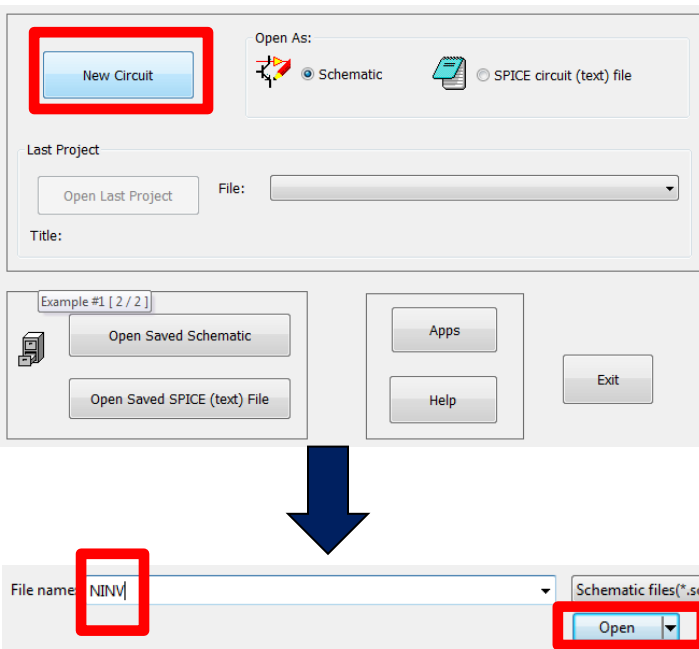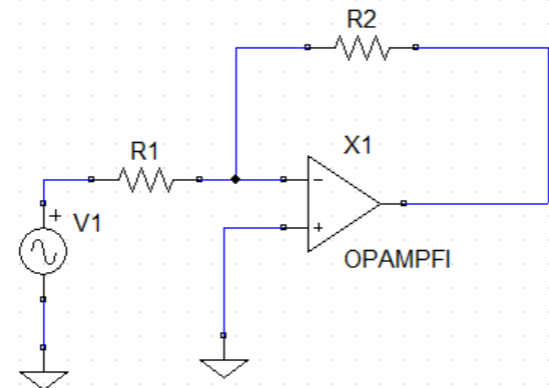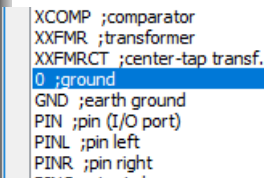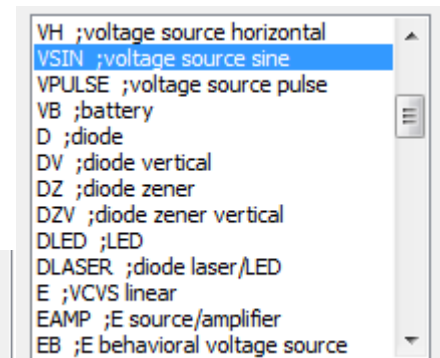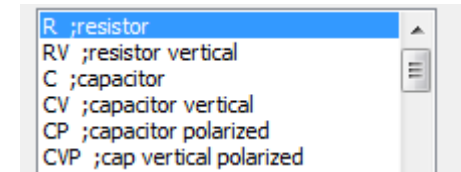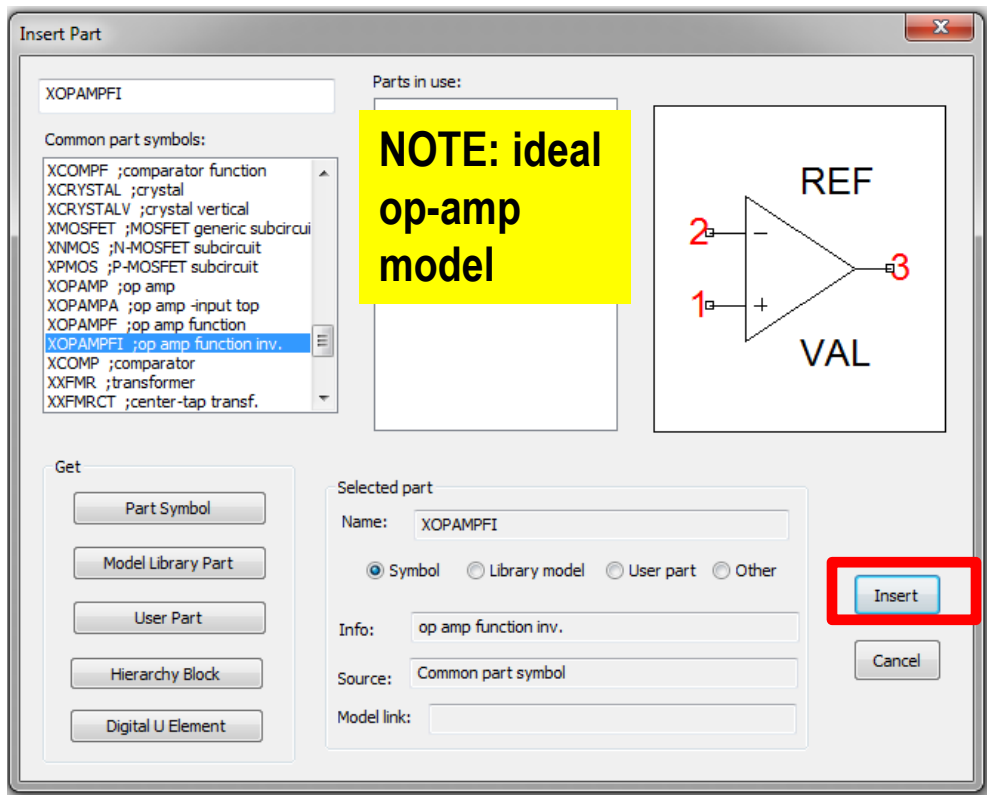
Draw wires
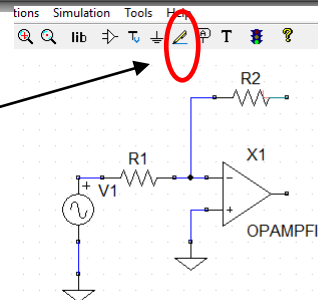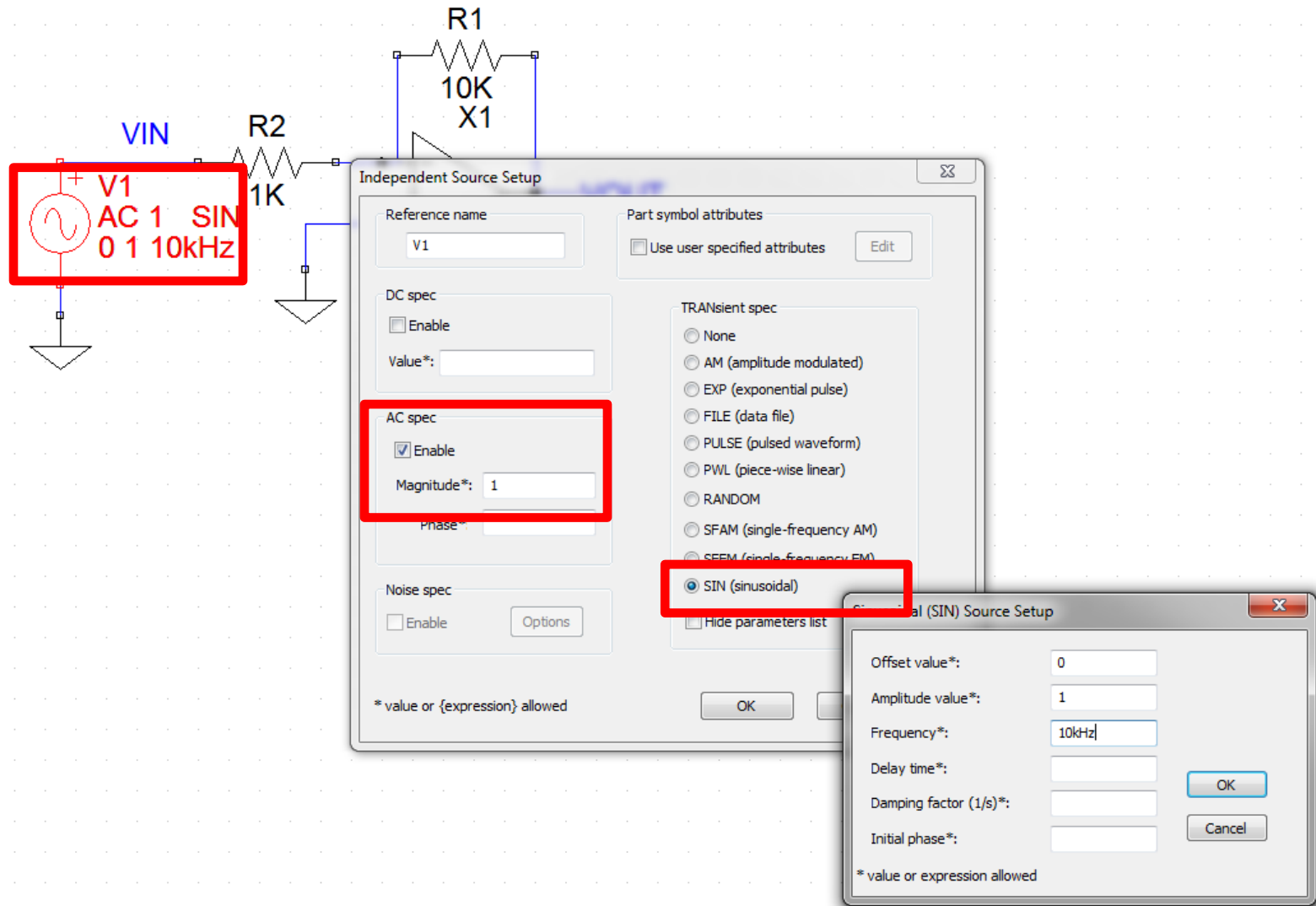
Example #2 [3 / 7]

Example #2 [3 / 7]
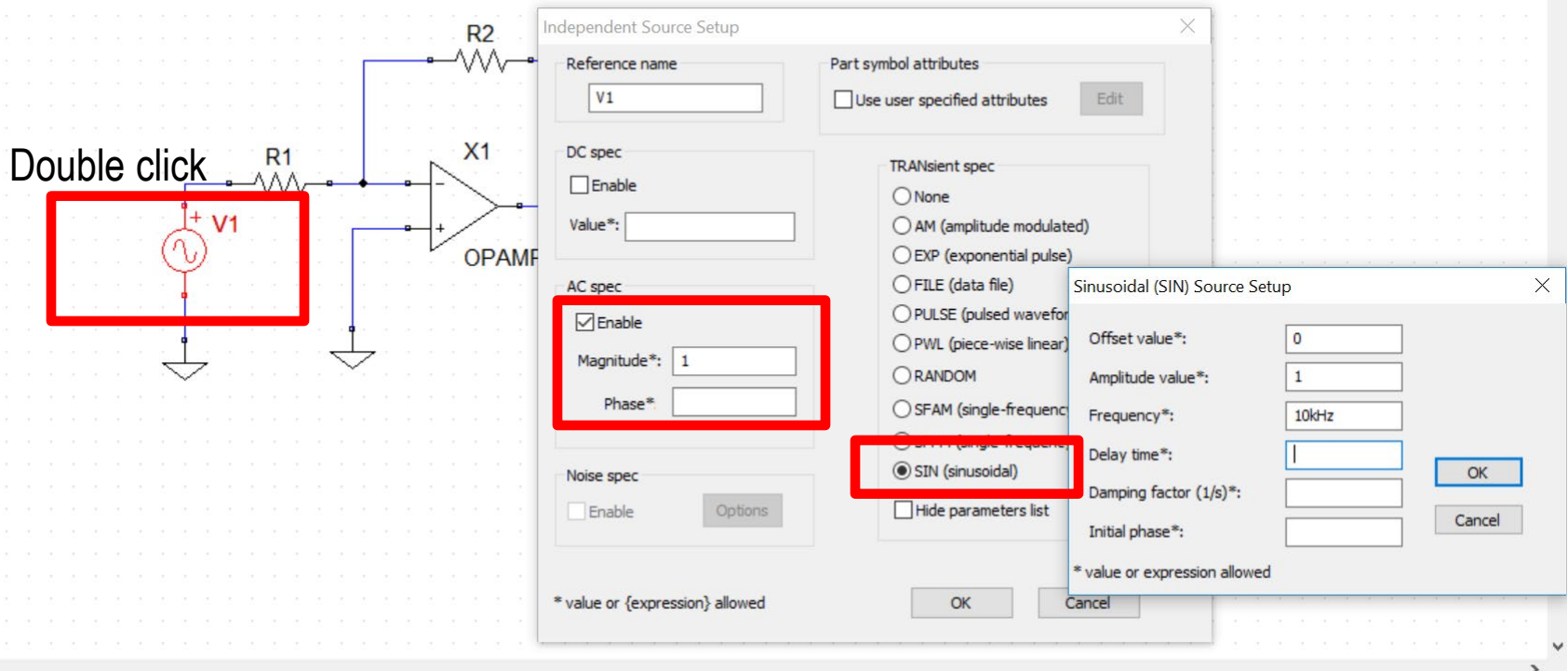
Example #2 [4 / 7]

Enter R1 value
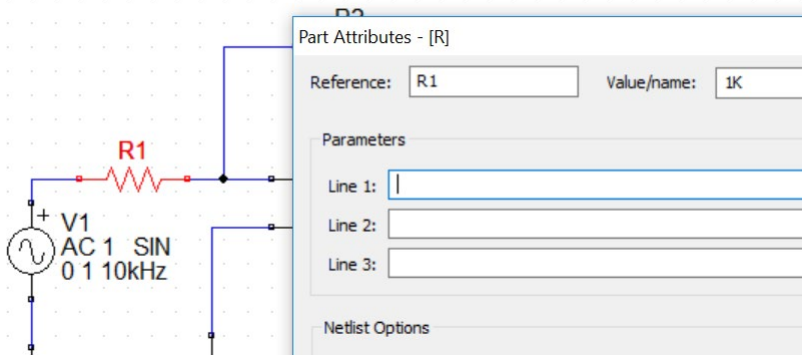
Enter R2 value
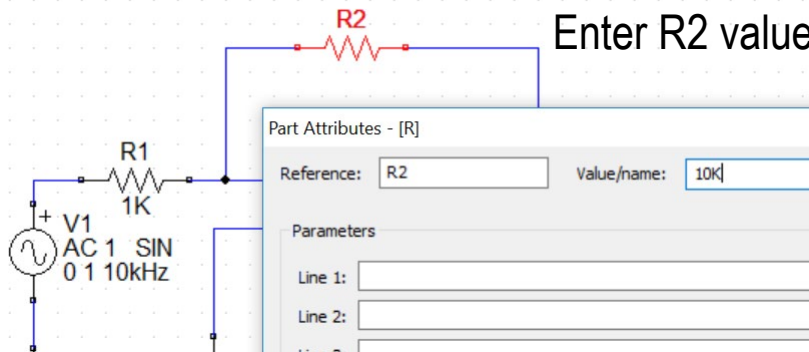
Label Input and output voltages
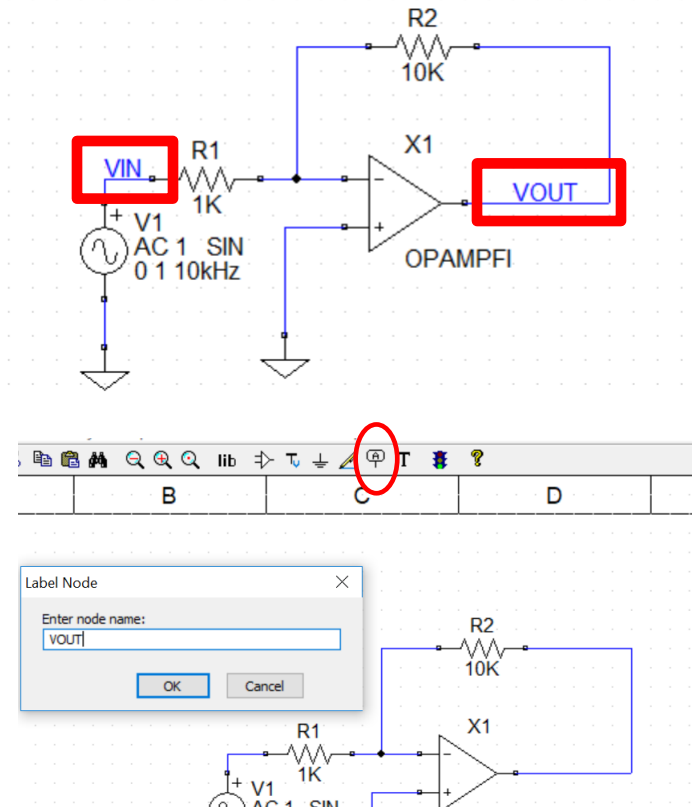
Example #2 [5 / 7]

Example #2 [6 / 7]

THE OHIO STATE UNIVERSITY
ELECTRICAL & COMPUTER ENGINEERING
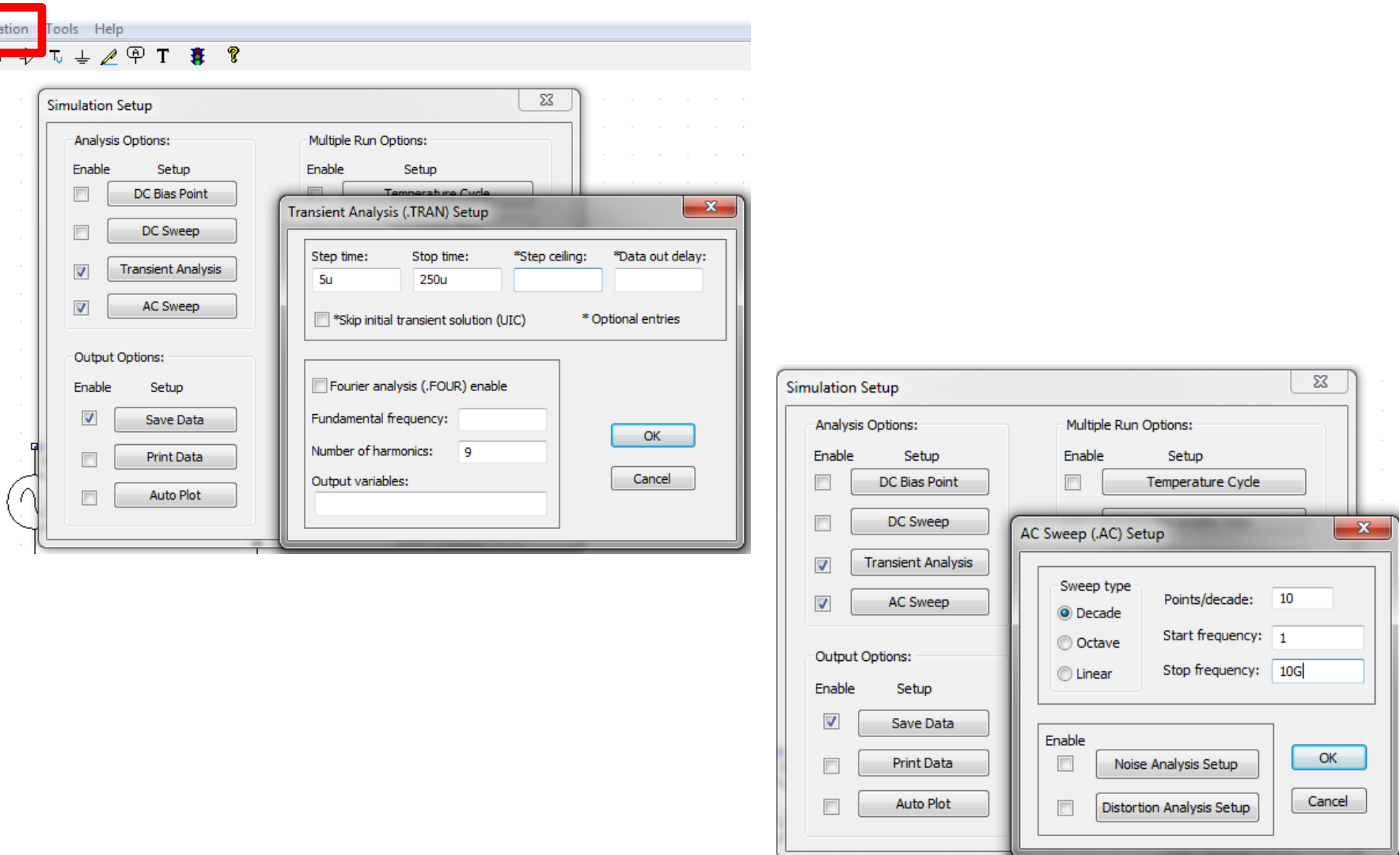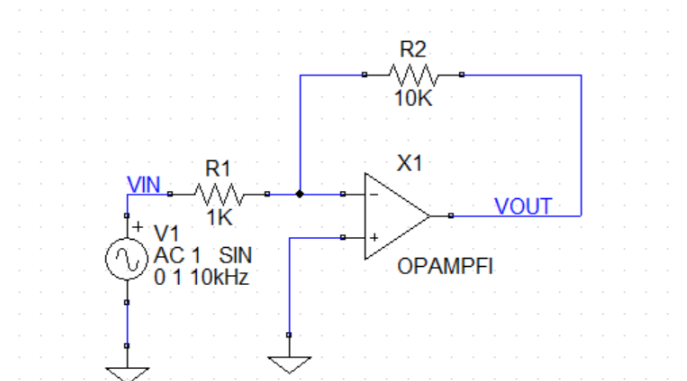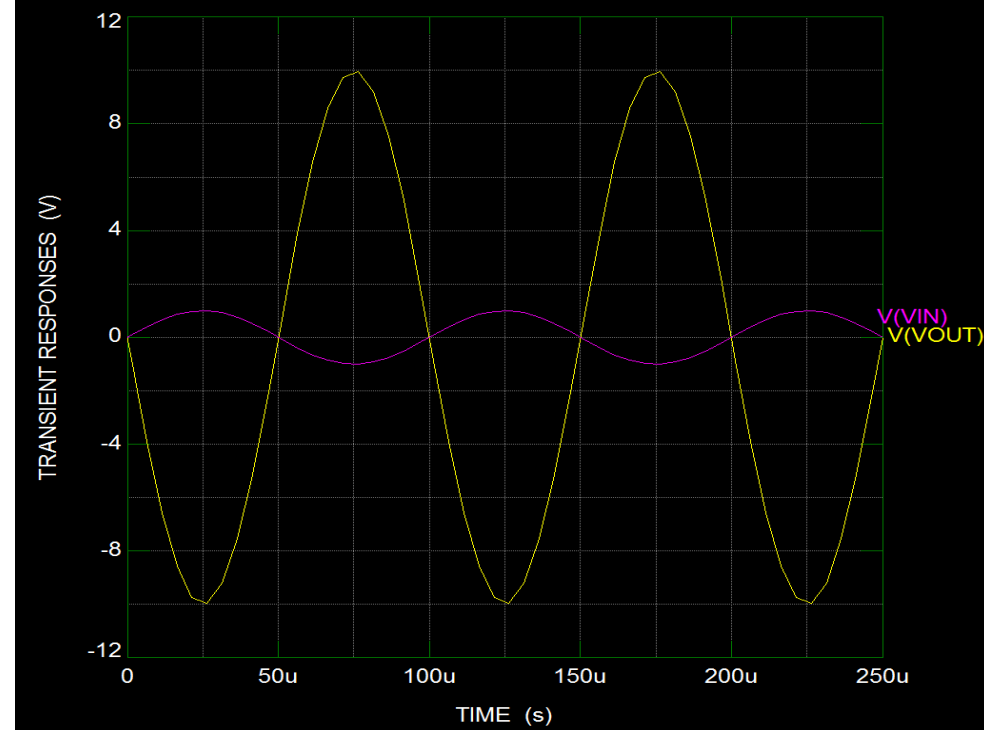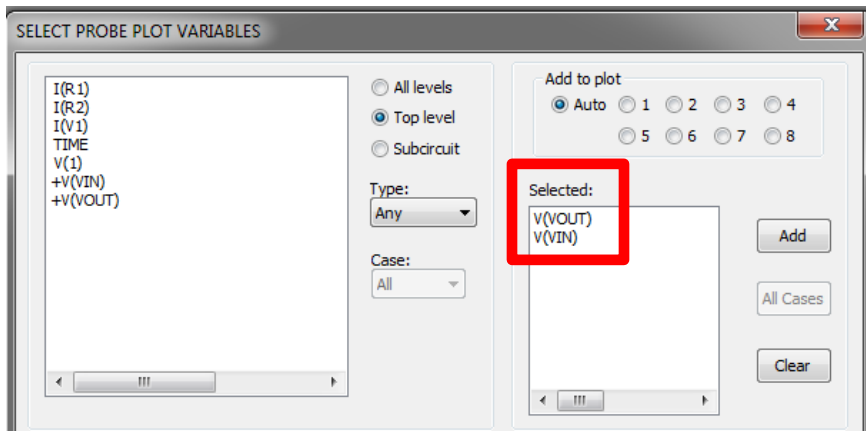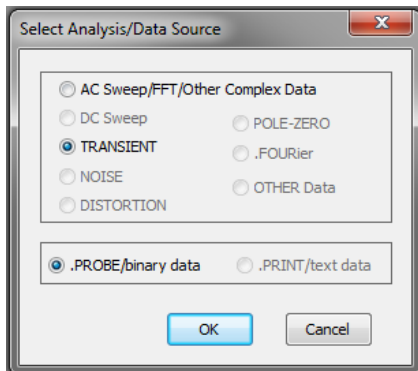
Example #2 [7 / 7]
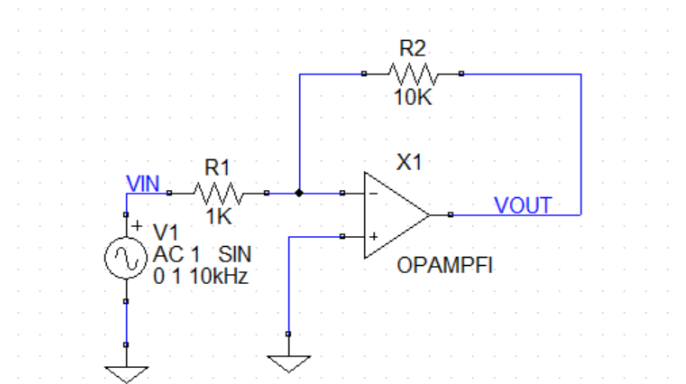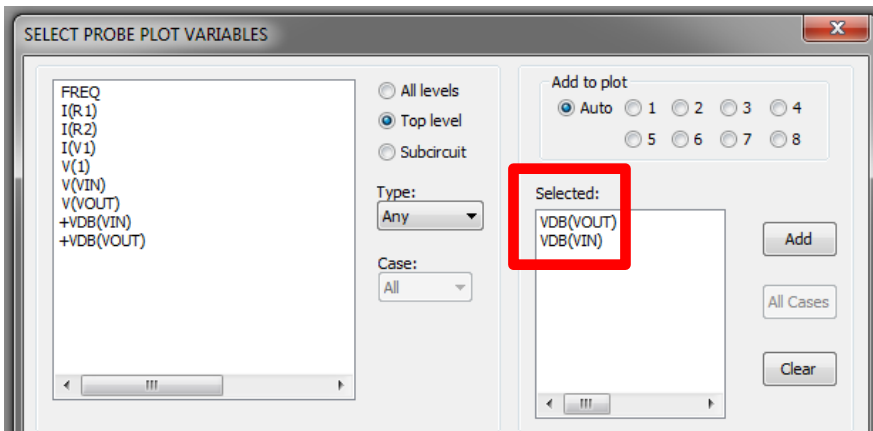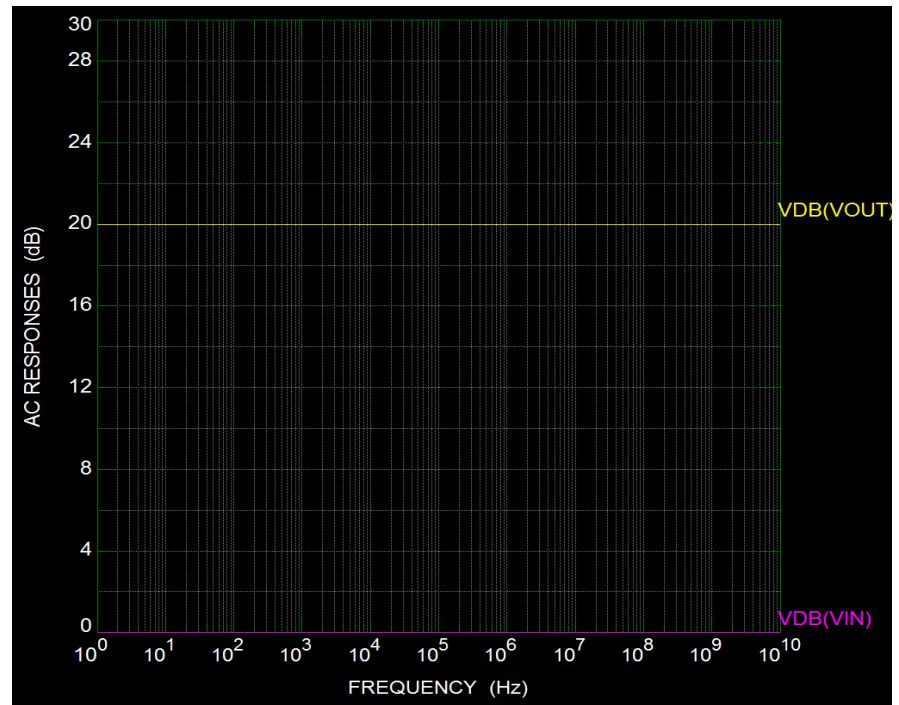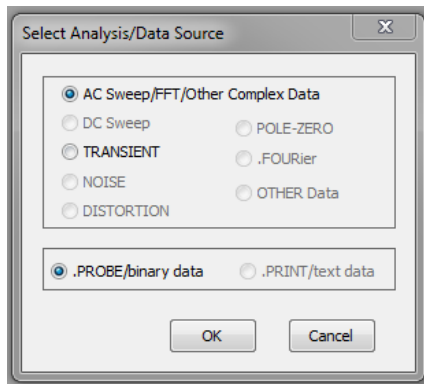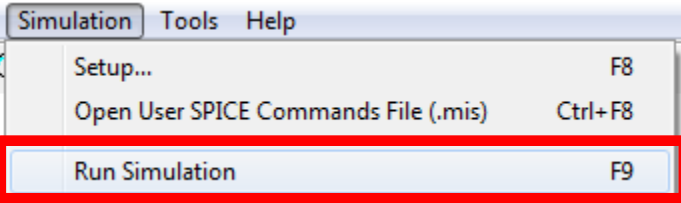
TopSpice Example #3
Simulate a NON-IDEAL inverting amplifier configuration using a circuit schematic

$$v_O = A(v_2 - v_1) = -Av_1$$

$$i_1 = \frac{v_I - v_1}{R_1} = \frac{v_1 - v_O}{R_2} = i_2$$

algebra

$$\frac{v_O}{v_I} = -\frac{R_2}{R_1} \times \frac{1}{1 + \left(1 + \dfrac{R_2}{R_1}\right)\dfrac{1}{A}}$$
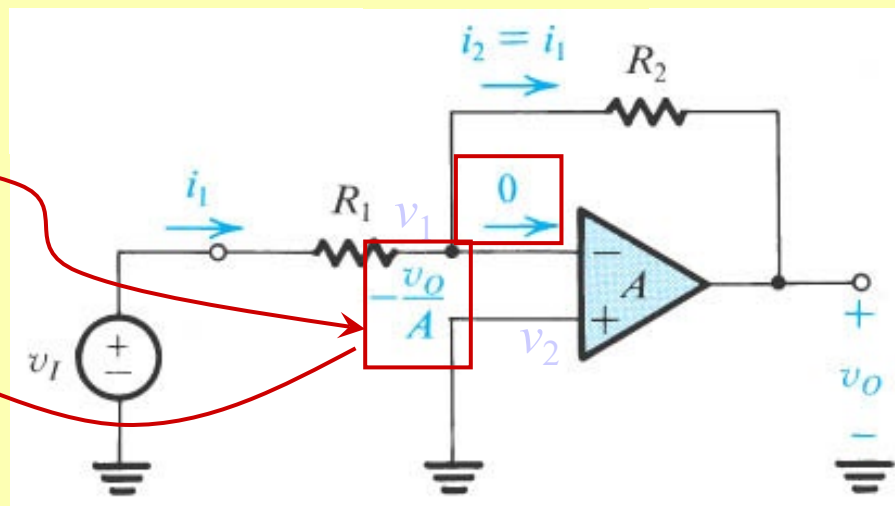
Calculate GBW for the inverting amplifier configuration when:

$$A(s) = \frac{A_0}{1 + \dfrac{s}{\omega_b}} \qquad \boxed{\omega_t \equiv A_0 \omega_b}$$

$$\frac{V_o(s)}{V_i(s)} = \frac{-R_2/R_1}{1 + \left(1 + \dfrac{R_2}{R_1}\right)\dfrac{1}{A(s)}}$$
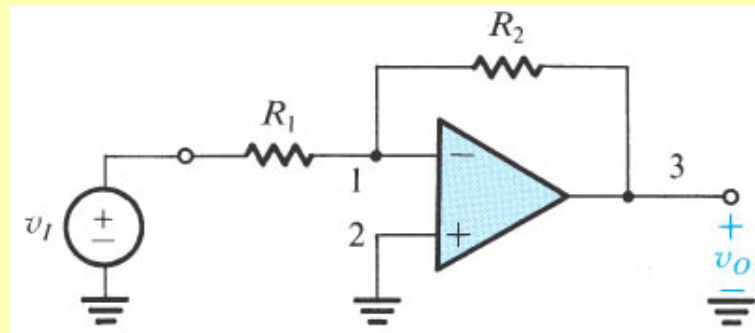
For $A_0 \gg 1 + \dfrac{R_2}{R_1}$

$$\frac{V_o(s)}{V_i(s)} \approx \frac{-R_2/R_1}{1 + \left(1 + \dfrac{R_2}{R_1}\right)\dfrac{s}{\omega_t}} = \frac{-R_2/R_1}{1 + \dfrac{s}{\omega_{3dB}}} \quad \text{with } \omega_{3dB} = \frac{\omega_t}{1 + \dfrac{R_2}{R_1}}$$
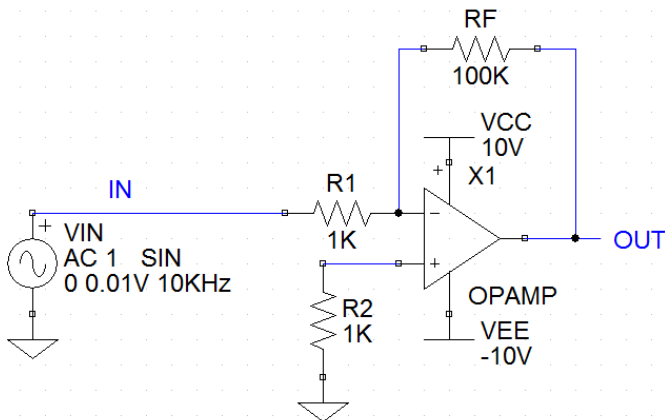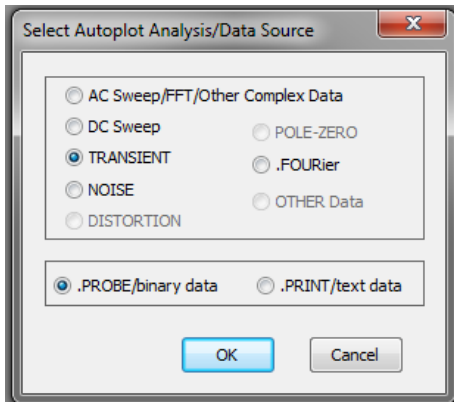
Example #3 [1 / 4]

Example #3 [2 / 4]

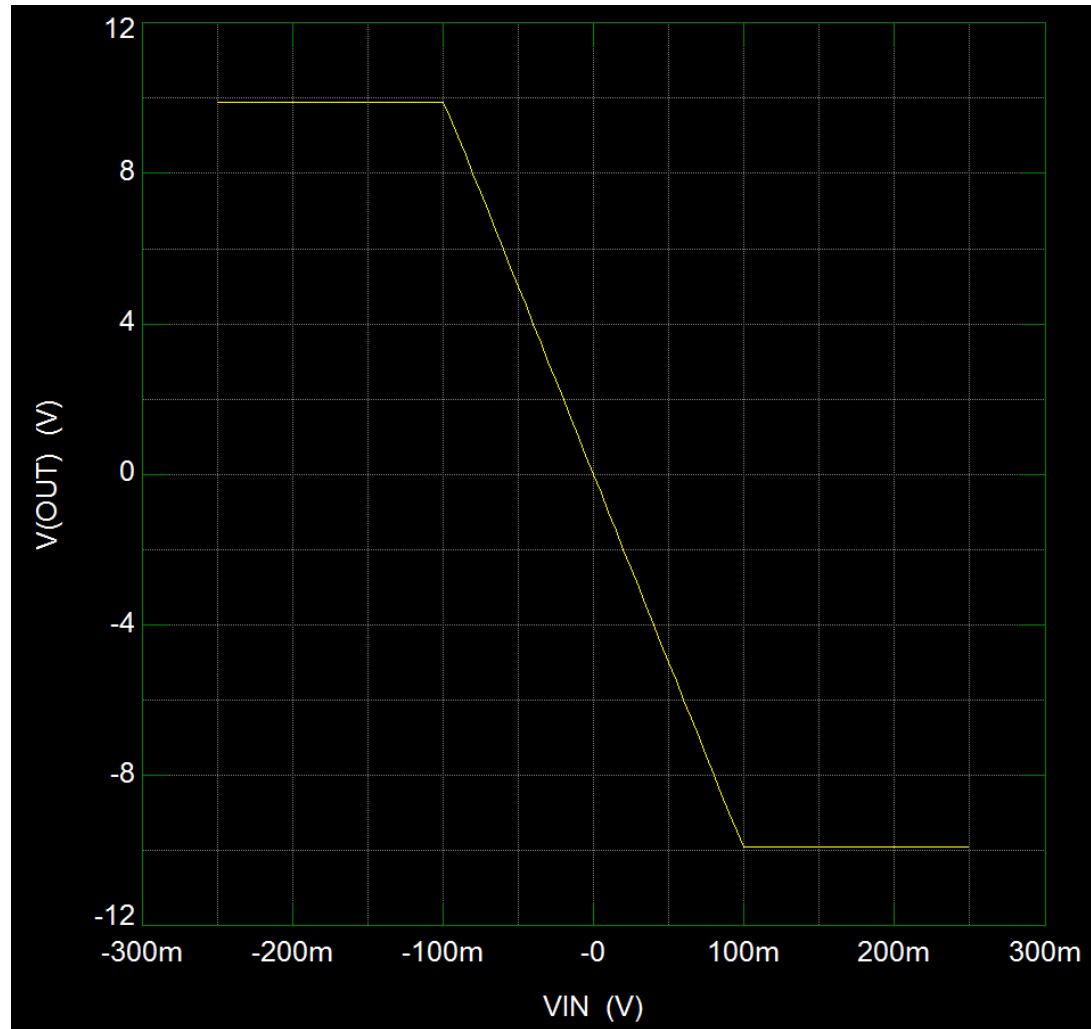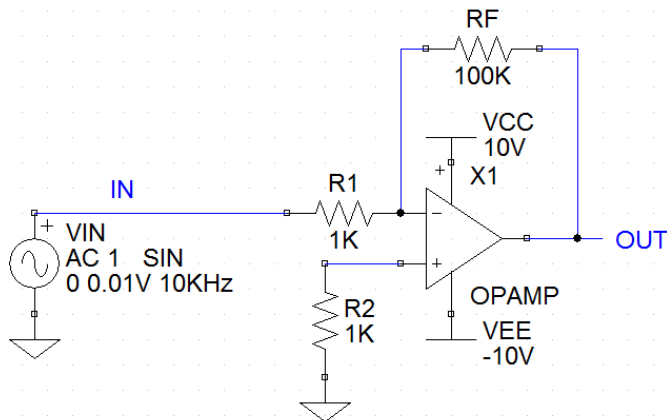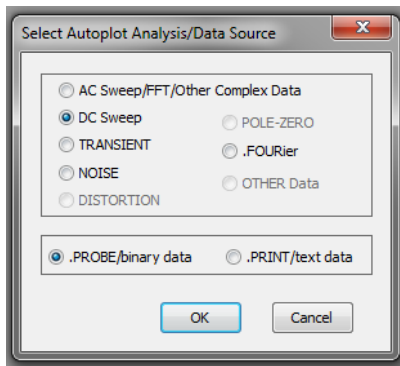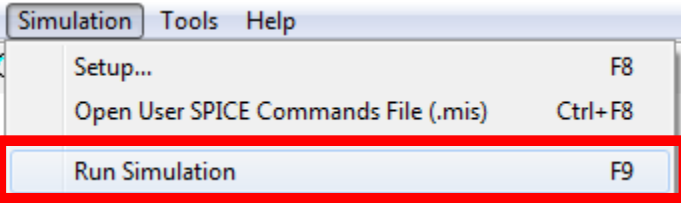# Example #3 [3 / 4]

Example #3 [4 / 4]

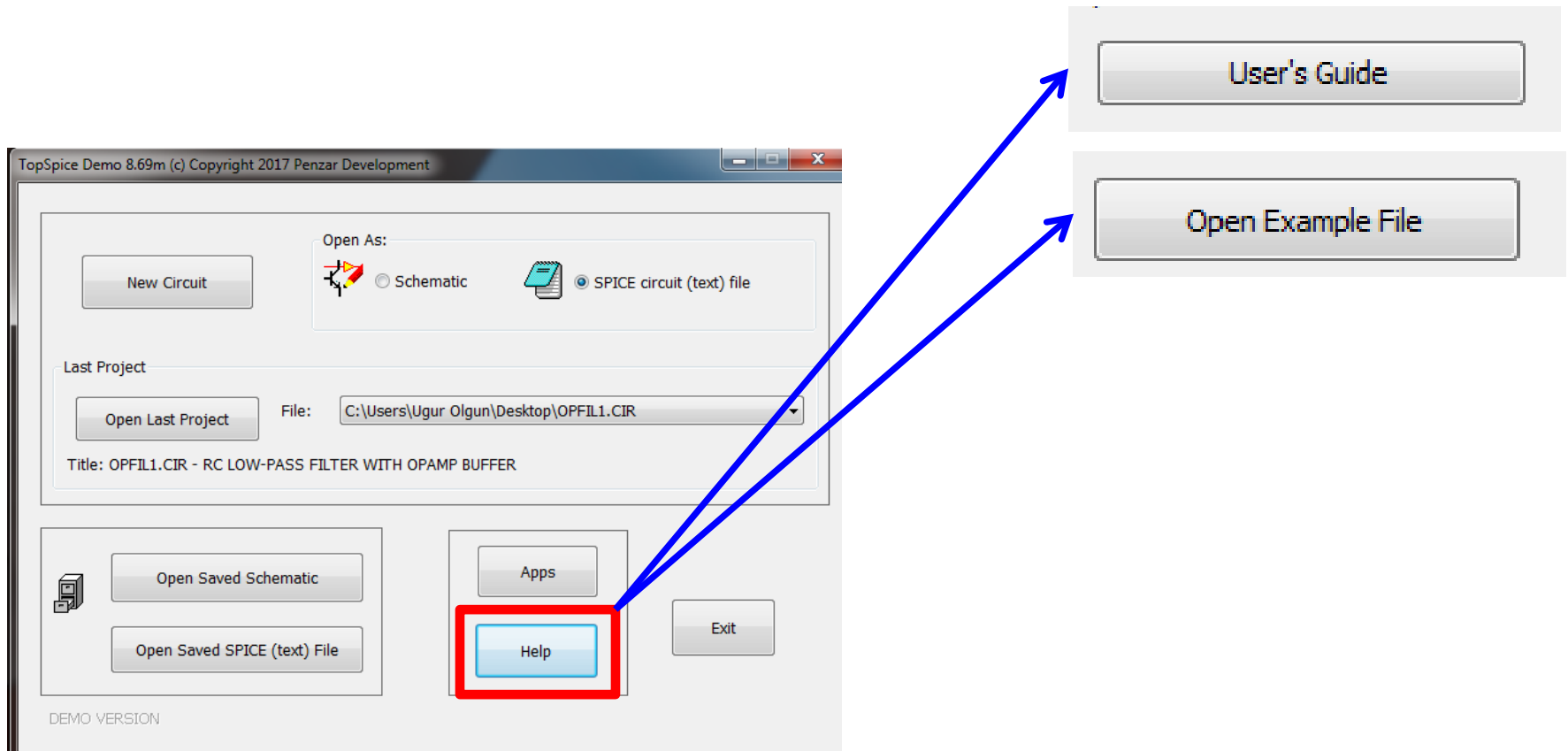# TopSpice
Additional Resources

Additional Resources

THE OHIO STATE UNIVERSITY
ELECTRICAL & COMPUTER ENGINEERING

Model Library: Available Parts

Part number	Model name	SPICE dev.	Symbol	Device type	Description
1N4001	1N4001	D (D)	D	DEMO	Diode
1N4148	1N4148	D (D)	D	DEMO	Diode
1N750	1N750	D (D)	DZ	DEMO	Zener diode
2N2222	2N2222	Q (NPN)	QNPN	DEMO	Bipolar transistor
2N2907A	2N2907A	Q (PNP)	QPNP	DEMO	Bipolar transistor
2N6851	2N6851	M (PMOS)	MPMOS	DEMO	MOSFET
2N7000	2N7000	M (NM...	MNMOS	DEMO	MOSFET
555	555	X	X555	DEMO	Timer IC behavioral model
7400	7400	X	XNAND	DEMO	2-input NAND
7402	7402	X	XNOR	DEMO	2-input NOR
7404	7404	X	XINV	DEMO	inverter
*	74HC	U (UG...	*	DEMO	CMOS 74HC series delay model
74HC00	74HC00	X	XNAND	DEMO	2-input NAND
74HC02	74HC02	X	XNOR	DEMO	2-input NOR
74HC04	74HC04	X	XINV	DEMO	inverter
74HC05	74HC05	X	XINV	DEMO	inverter open drain
74HC08	74HC08	X	XAND	DEMO	2-input AND
*	74HCAD	O (ATOD)	*	DEMO	CMOS 74HC series I/O model
*	74HCDA	O (DTOA)	*	DEMO	CMOS 74HC series I/O model
*	74HCODIO	O (UIO)	*	DEMO	CMOS 74HC series I/O model
*	74HCUIO	O (UIO)	*	DEMO	CMOS 74HC series I/O model
*	COMP	X	XCOMP	ANALOG	Generic comparator
*	DAC4BIT	X	XDAC4BIT	DEMO	4-bit DAC
*	DBRIDGE	X	XDBRIDGE	Diode	Rectifier diode bridge
*	LXFMRCT	X	XXFMRCT	DEMO	Linear transformer with center tapped secondary
MPS3903	MPS3903	Q (NPN)	QNPN	DEMO	Bipolar transistor
*	OPAMP	X	XOPAMP	DEMO	Generic op amp model (floating supplies)
*	OPAMPA	X	XOPAMPA	DEMO	Generic op amp model IN- top
*	OPAMPF	X	XOPAMPF	DEMO	Ideal op amp function
*	OPENLOOPGAIN	X	XMETEROLG	ANALOG	Open loop gain measurement macro
*	SPARAM2P	X	X2PORT	DEMO	General 2-port s-parameters table network model
*	ST	X	XST	ANALOG	Schmitt trigger
*	STI	X	XSTI	ANALOG	Schmitt trigger inverting
*	THERMISTOR	X	XRES	DEMO	Thermistor table model
TLC393	TLC393	X	XCOMP	DEMO	TLC393 comparator
*	TTL	U (UG...	*	DEMO	TTL 7400 series I/O model
*	TTLAD	O (ATOD)	*	DEMO	TTL 7400 series I/O model
*	TTLDA	O (DTOA)	*	DEMO	TTL 7400 series I/O model
*	TTLUIO	O (UIO)	*	DEMO	TTL 7400 series I/O model
UA741	UA741	X	XOPAMP	DEMO	UA741 op amp
*	VCO	X	XFUN	DEMO	Analog behavioral model of VCO
*	XFMR	X	XXFMR	ANALOG	Ideal linear transformer (no losses)
*	XFMRI	X	XXFMR	ANALOG	Ideal transformer

Adding from Model Library