ELEC 3509
Electronics II

Professor Q.J. Zhang

Department of Electronics

How to use NI Multisim

To Simulate a Circuit

plus

Two Exercise Examples for Practice

How to Start the NI Multisim Program

NI Multisim can be used for simulation and design of electronic circuits.

There are two ways to use the software: remotely or locally

To use it remotely:

Use VPN, and Remote Desktop Connection to login to DOE's Windows based computers (see slide 4)

start the Multisim program on DOE's computer (see slide 6).

To use it locally:

Download and install Multisim onto your own computer (see slide 5)

Start the Multisim program on your own computer (see slide 6).

To Run the Software Remotely:

Login to DOE's Window Based Computer/Server

Use your computer to connect to Carleton's VPN

Use "Remote Desktop Connection" on your computer to connect to DOE's Window based
server/computer

_____

Instructions for VPN: https://carleton.ca/its/help-centre/remote-access/

Instructions for Remote Desktop Connection:
http://doe.carleton.ca/
-> Computer Support
-> How to access DOE's Windows based servers remotely
-> How to Access a Local Desktop Computer using Remote Desktop Client

To Run the Software Locally:

Download and Install NI Multisim

Search "Multisim Download" in web to find download location. The official website for this software is by National Instruments (NI) in www.ni.com

You may need an NI User Account in their website to download the software. Sign up as a student.

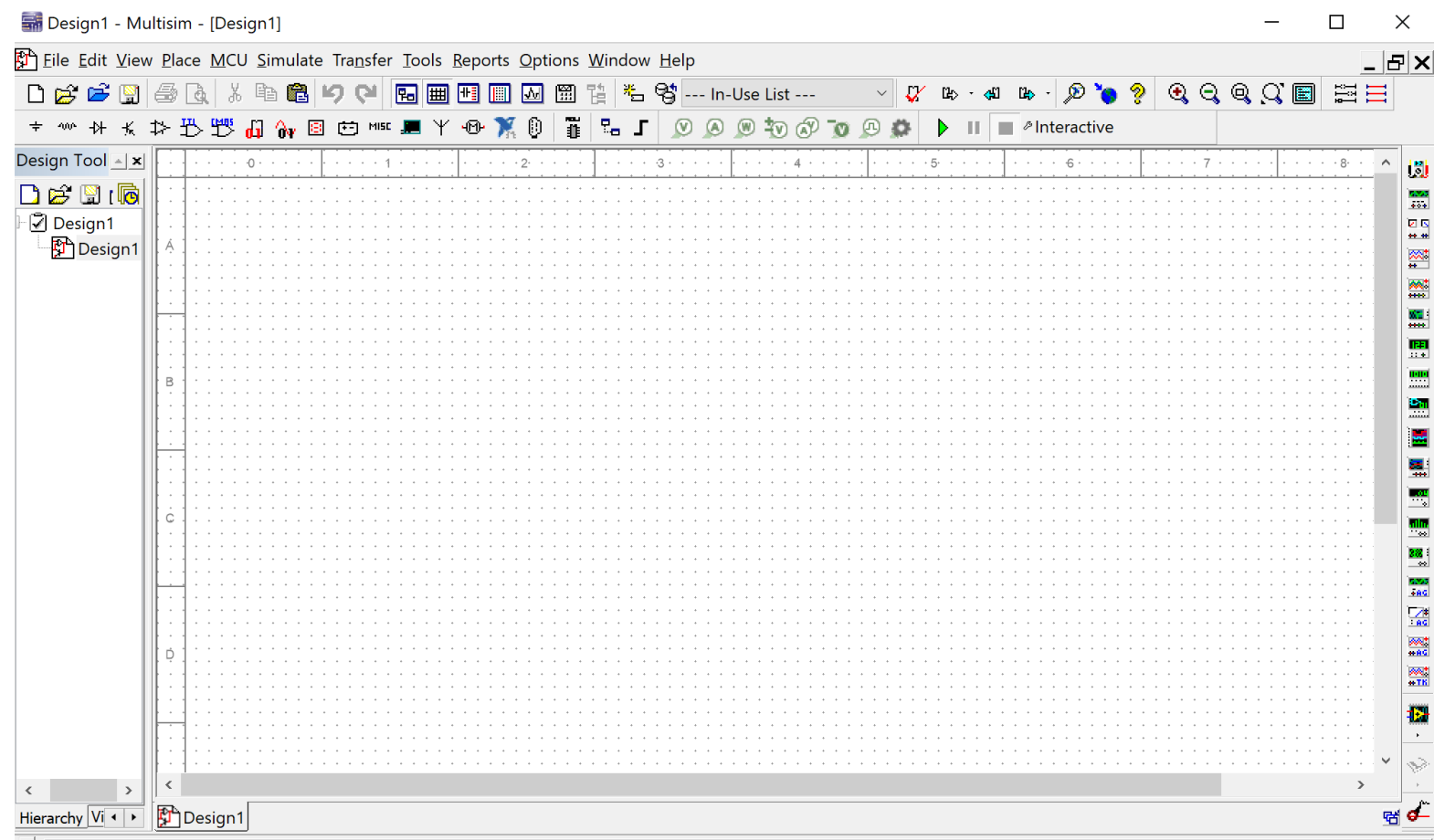Download NI Multisim 14.2 (Education Edition)

Install NI Multisim 14.2 on your computer.

Start Multisim

From the Start menu on the Windows based computer, find "NI Multisim 14.2".

Click it.

The main window of Muiltim will appear

Place a Resistor into the Schematic Window

Click the "Place Basic" icon (Fig. 1)
    a dialog window will appear (Fig. 2)
    select Resistor and click "OK" (Fig. 2)
    the resistor will be placed into the circuit schematic

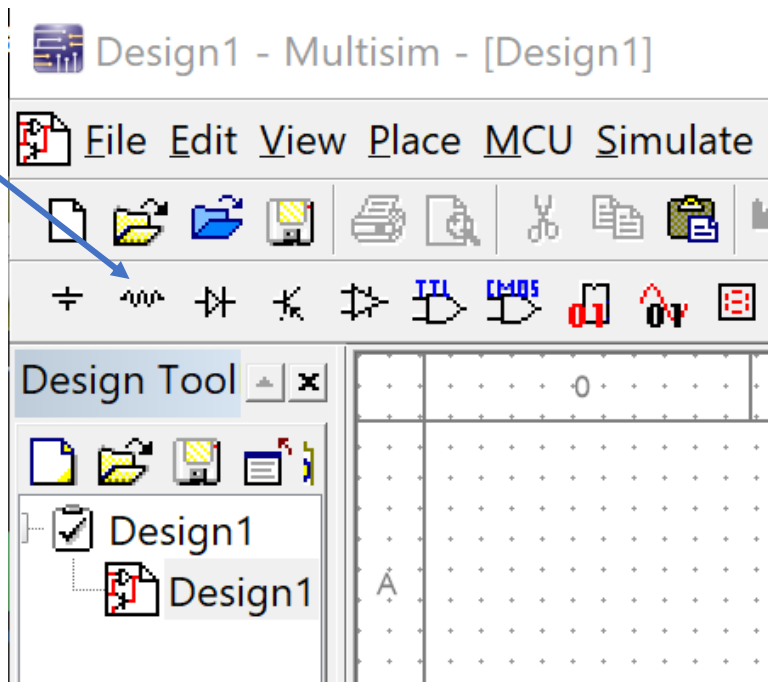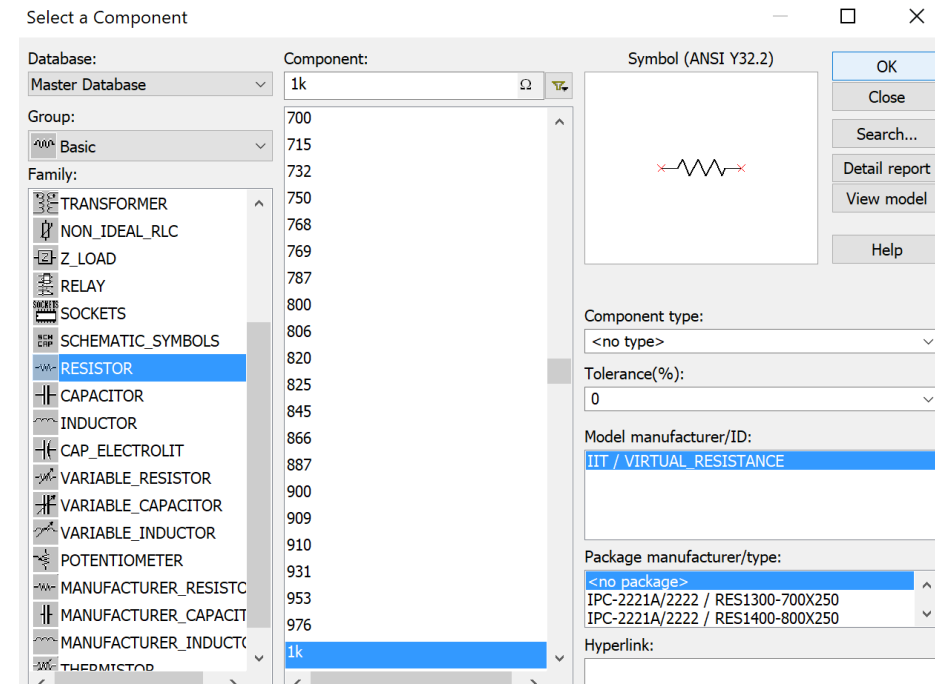Capacitors and inductors can be selected similarly

Place
Basic
icon

Fig. 1

Fig. 2

Modify the Value and Label of the Resistor

-> double click on the resistor symbol ⊢⊣WW⊢ (Fig. 1)

    -> a dialog window will appear, ⟶
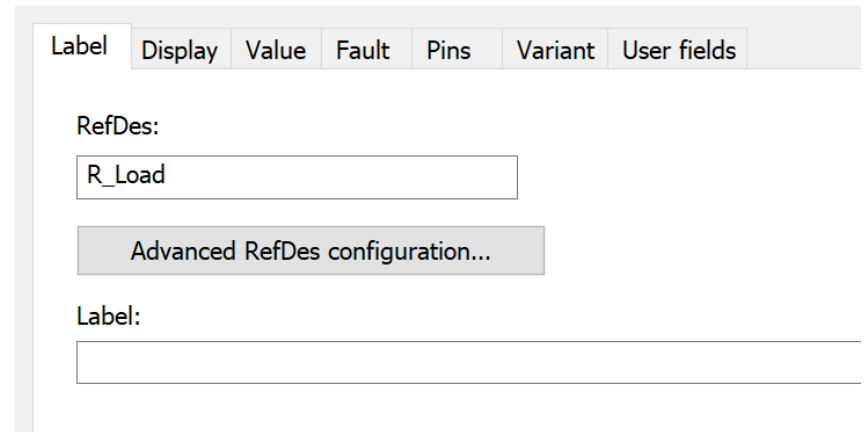   -> The window has Tabs such as "Label", "Value"
    -> click the "Label" tab, edit the name of the resistor
    -> click the "Value" tab, edit the value of the resistance.

-> right click on the resistor symbol ⊢WW→, then
    -> select "Rotate 90 degree clockwise"

Resistor

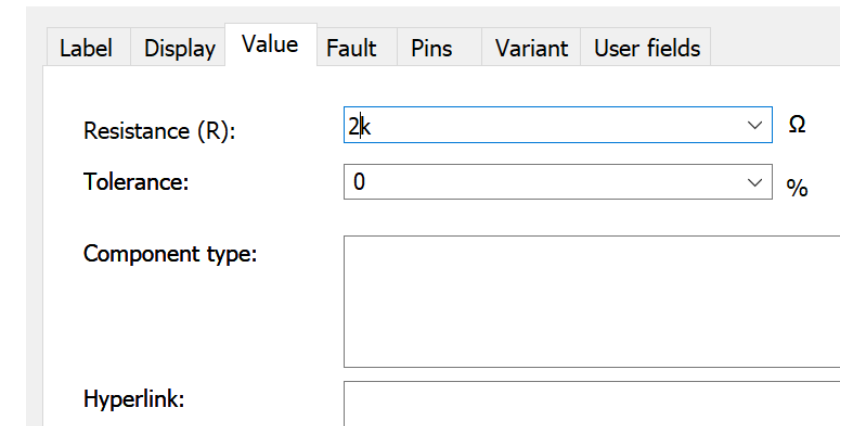| Label | Display | Value | Fault | Pins | Variant | User fields |

RefDes:

R_Load

Advanced RefDes configuration...

Label:

Resistor

| Label | Display | Value | Fault | Pins | Variant | User fields |

Resistance (R): 2k Ω

Tolerance: 0 %

Component type:

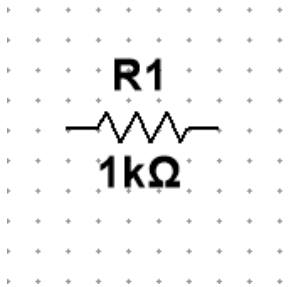Hyperlink:

R1
⊢WW⊢
1kΩ

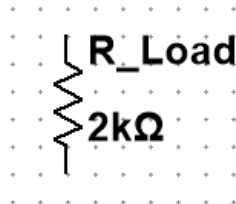Fig. 1: Original resistor

⌐ R_Load
⌇
⌇2kΩ

Fig. 2: The resistor is rotated (now vertical)
      Resistor label is changed to R_Load
      Resistor value is changed to 2K

Place Source and Ground into the Schematic Window

Click the "Place Source" icon   (Fig. 1)
    a dialog window will appear   (Fig. 2)
    select AC VOLTAGE   (Fig. 2)
    select GROUND   (Fig. 3)

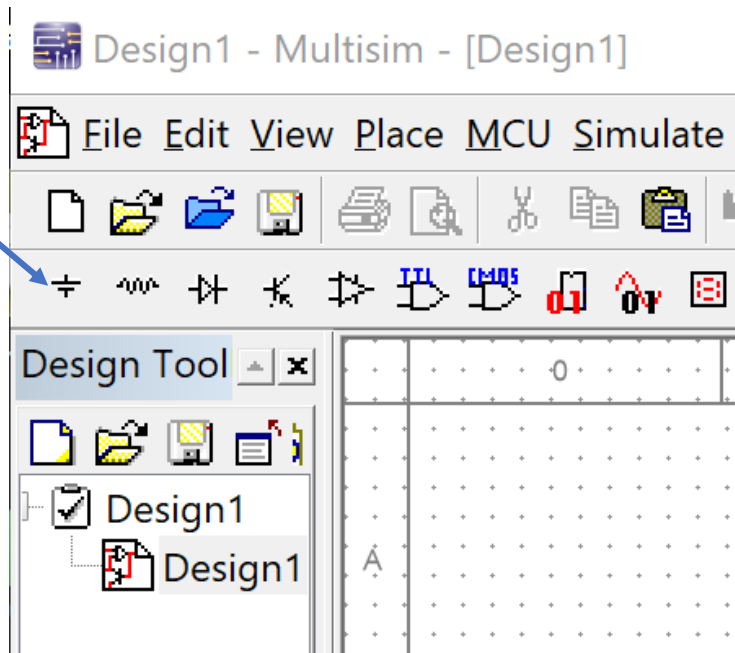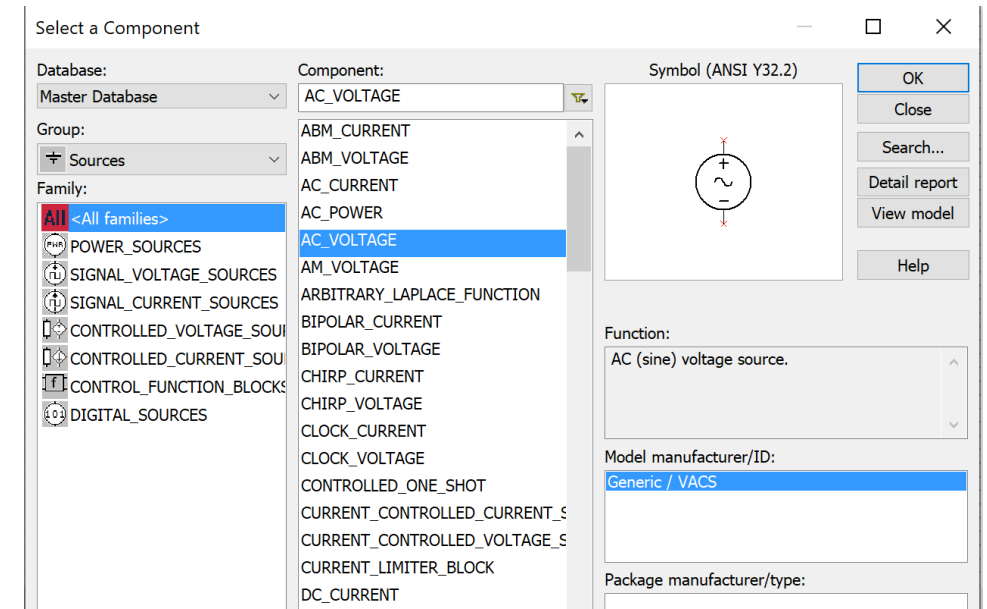Remark: for DC voltage source, select DC POWER
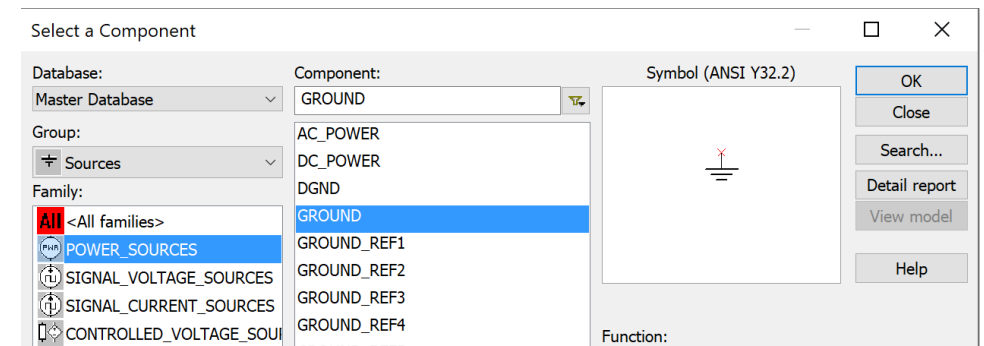
Place
Source
icon

Fig. 1

Fig. 2

Fig. 3

Place a Wire to Connect Components Together in the Schematic

From the "Place" menu, find "Wire" and left-click on "Wire" (Fig. 1)

then in the schematic window,
    left-click on a component node (start wire)
    left-click on another component node (end wire).
Now the two components are connected   (Fig. 2)
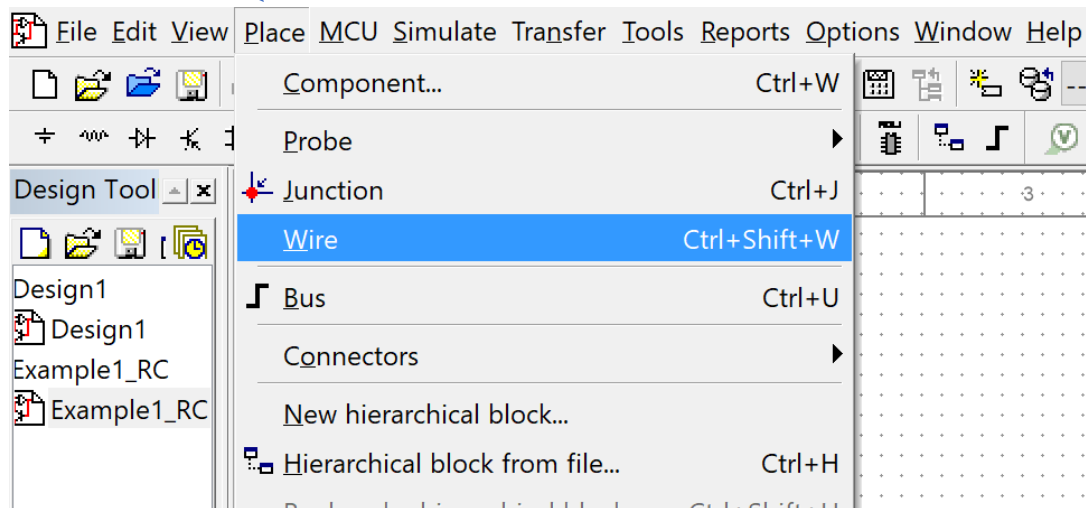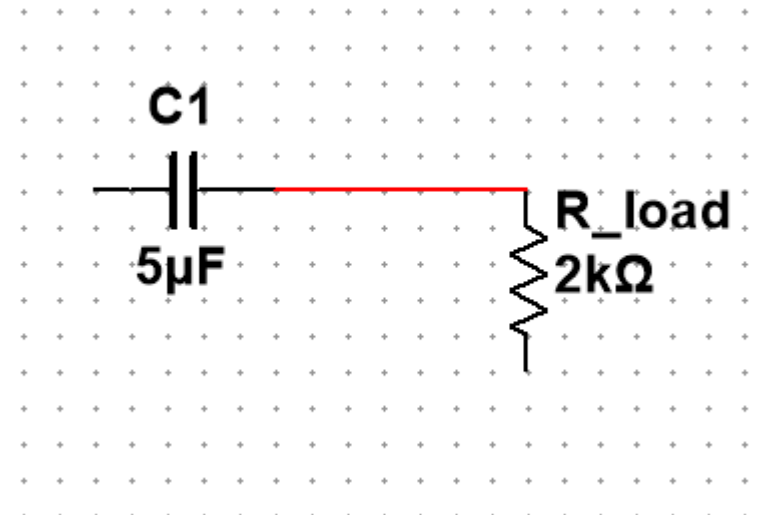
Place
menu

Fig. 1

Fig. 2

Create a Complete Circuit
(simple RC example)

-> place a resistor into the schematic
-> place a capacitor into the schematic
-> place an AC source into the schematic
-> place two grounds into the schematic
-> use wire to connect the components

-> Finally, get a voltage probe (Fig 1), and place it at the output node (at R_Load). Change the label of the probe to "Vout".

An alternative place to get a probe is from "Place" menu then select "Probe"

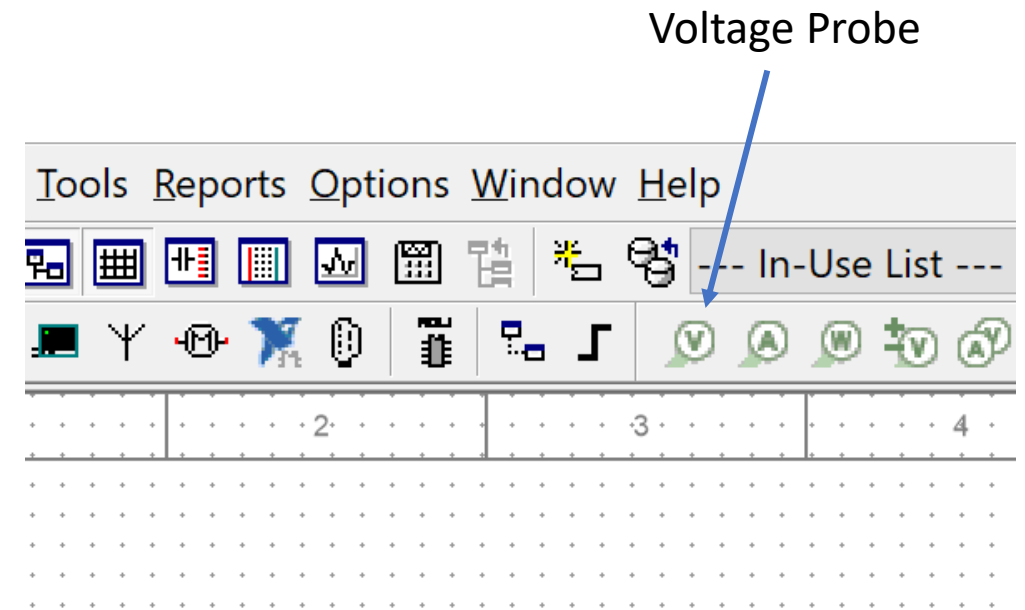Now you have the complete circuit schematic as shown in Fig. 2

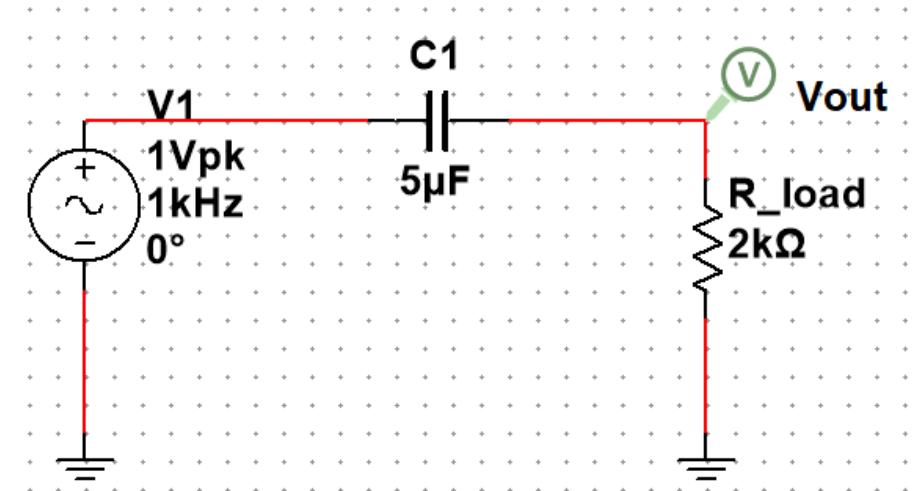Voltage Probe

Fig. 1: get a voltage probe

Fig. 2: the RC circuit

Define Analysis and Simulation Settings

Click the field for "Analysis and Simulation"
settings (default is "Interactive"), see Fig. 1

-> A dialog window, named "Analysis
and Simulation" will appear (Fig. 2)

-> select "AC Sweep"
-> enter values for
"Start Frequency",
"Stop Frequency"
"Number of points per decade"

Remark: other useful settings can be
similarly setup, such as:
DC Sweep,
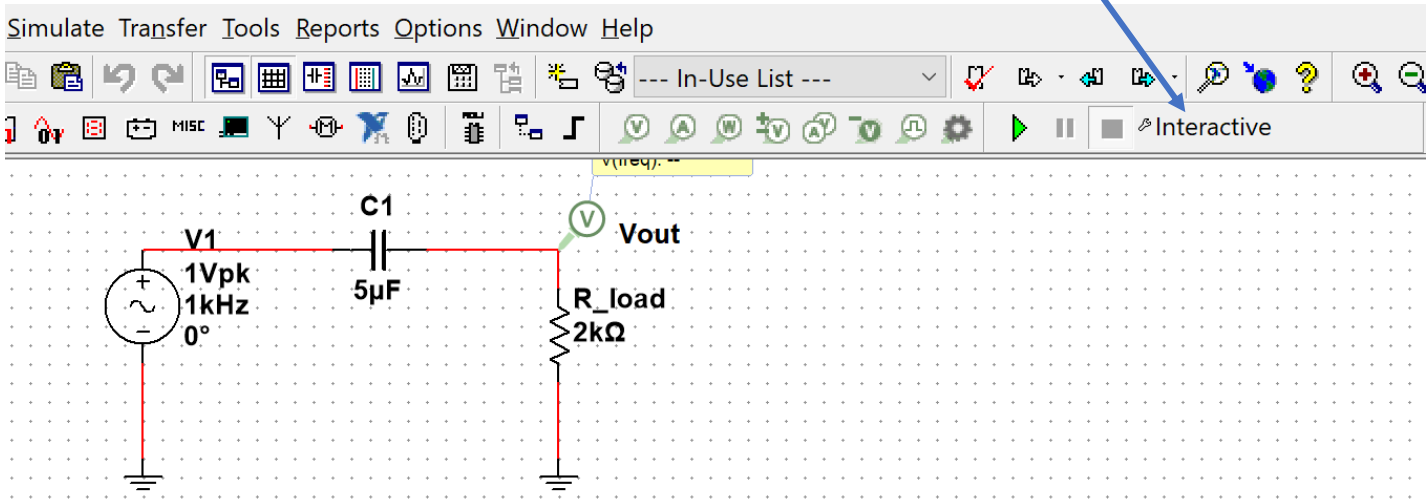DC Operating Point,
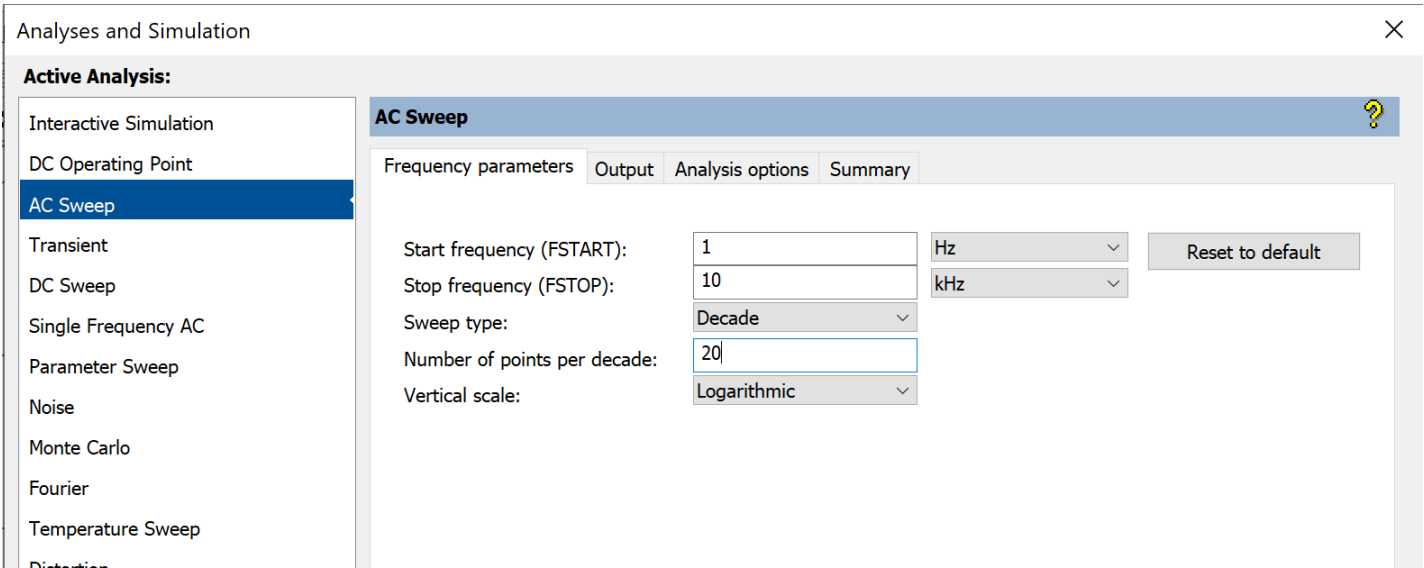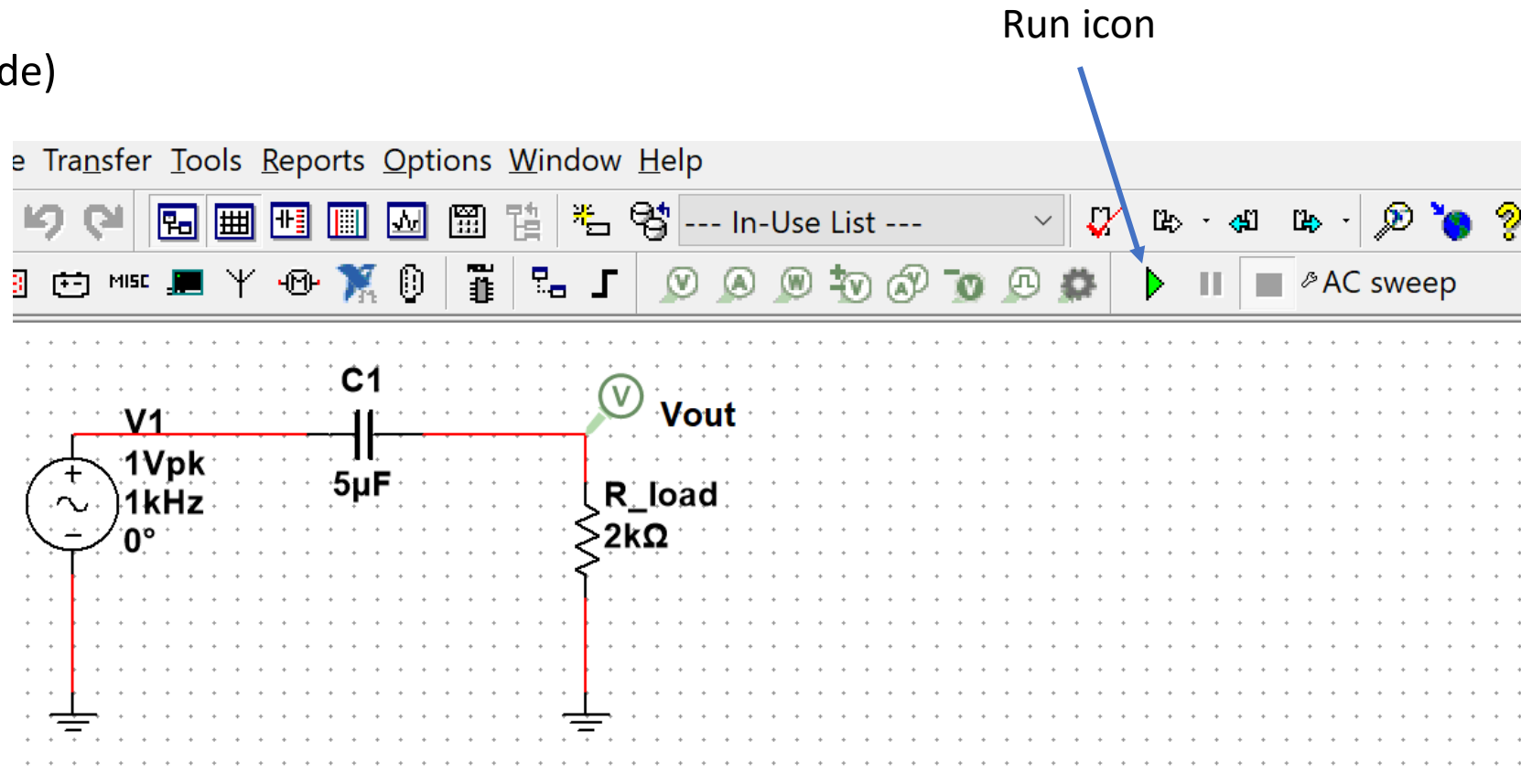Transient,
Interactive

Fig. 1

Fig. 2

Perform Simulation

Click the "Run" Icon (See Figure),
the simulation will start.

After simulation finishes,
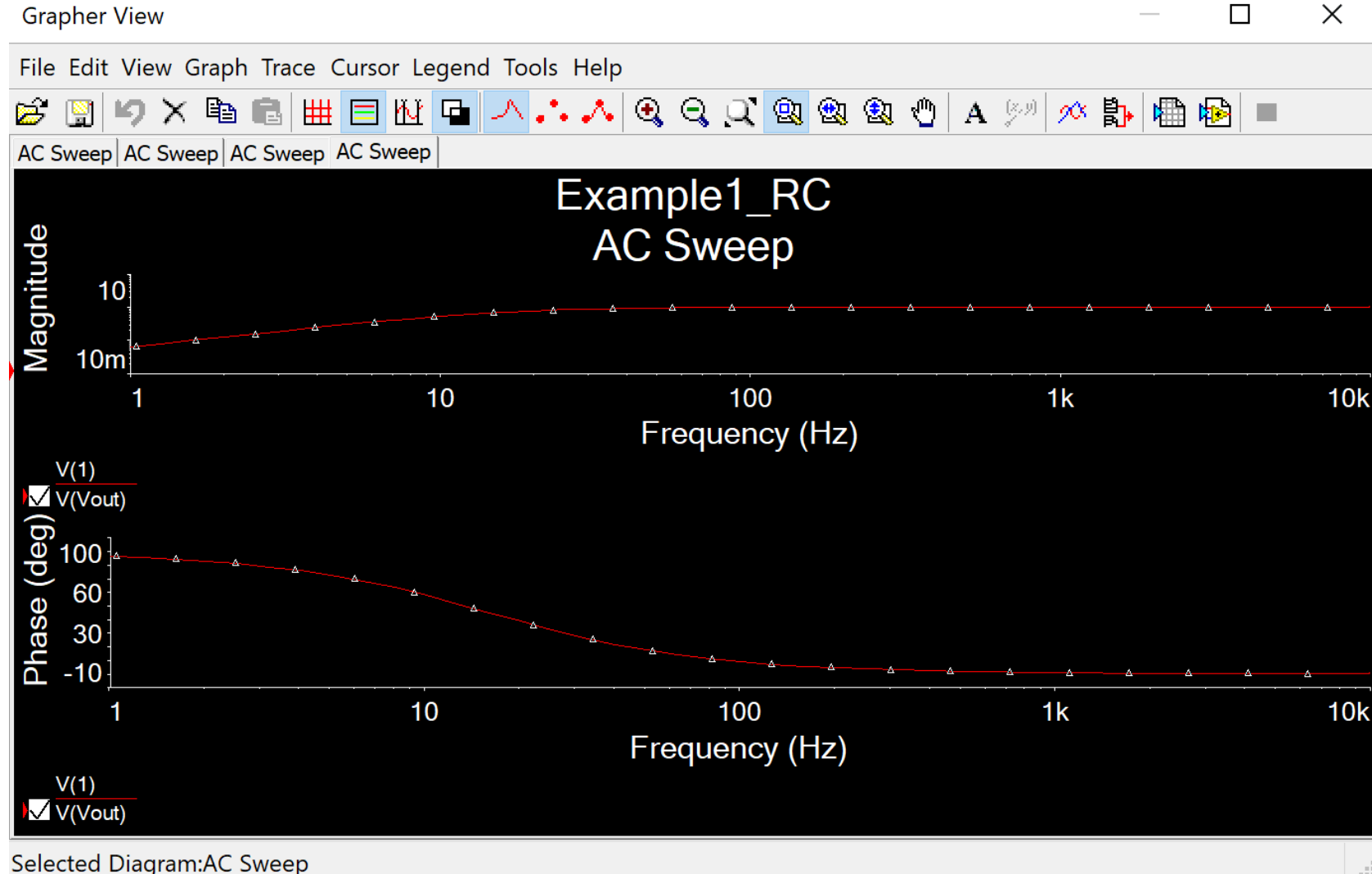a display window will pop-up (next slide)

Run icon

Display Results

After simulation is finished, the "Grapher View" window will appear.

The Grapher shows the probe signal, i.e., Vout versus frequency.

The horizontal axis is frequency in log scale. The frequency is from 1Hz to 10KHz.

The vertical axis represent the magnitude and phase of Vout.

How to Place a Transistor into the Circuit

Click the "Place Transistor" icon (Fig. 1)

A dialog window pops up (Fig. 2).
    select BJT NPN
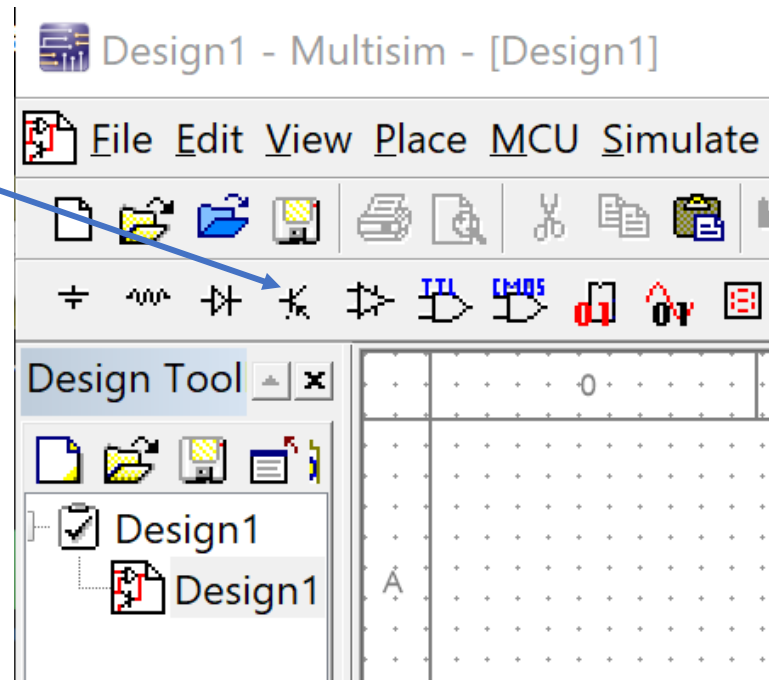   select "2N3904" which is a specific NPN BJT.

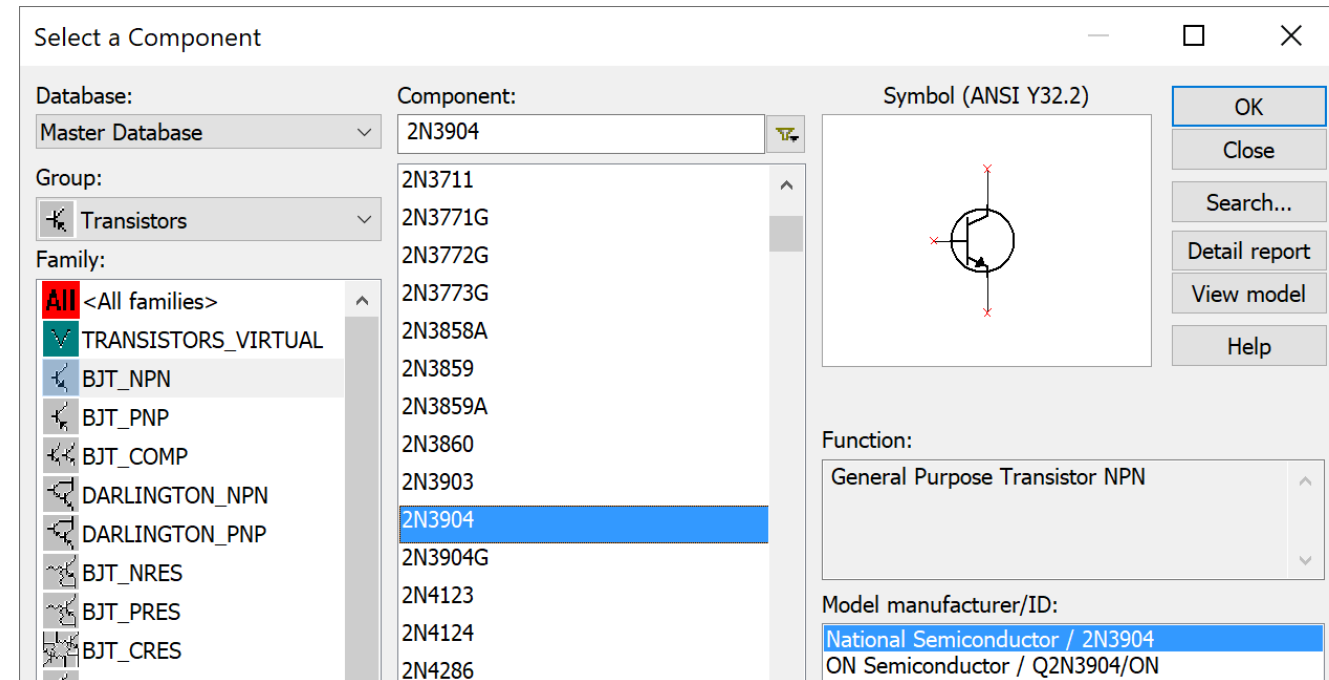Place Transistor icon

Fig. 1

Fig. 2

Exercise 1: RC Circuit Example

This example is a summary of the illustrative example used in earlier pages. You are encouraged to try it as an exercise.

Create the circuit schematic (Fig. 1). The source is an AC voltage source. The capacitor value is 5µF. The resistor value is 2KΩ. A voltage probe labeled as Vout is placed at R_load.

Define "Analysis and Simulation" setting (Fig. 2). Notice that the "AC Sweep" is selected. The sweep variable is frequency. The frequency range is set to be from 1Hz to 10KHz. We use 20 frequency points per decade.
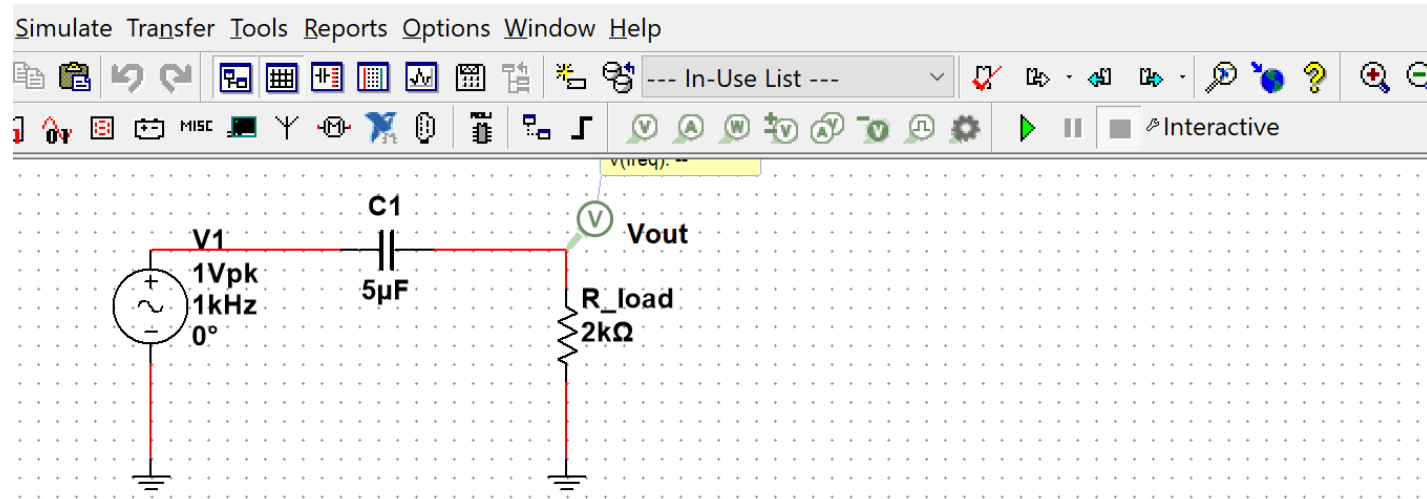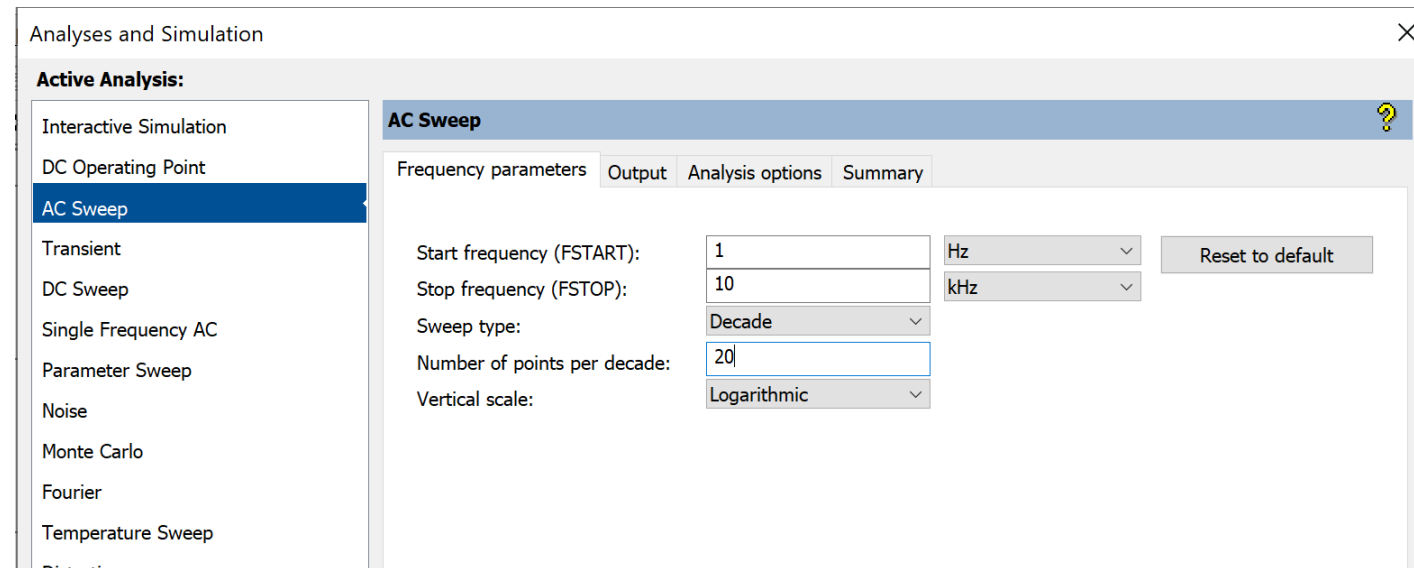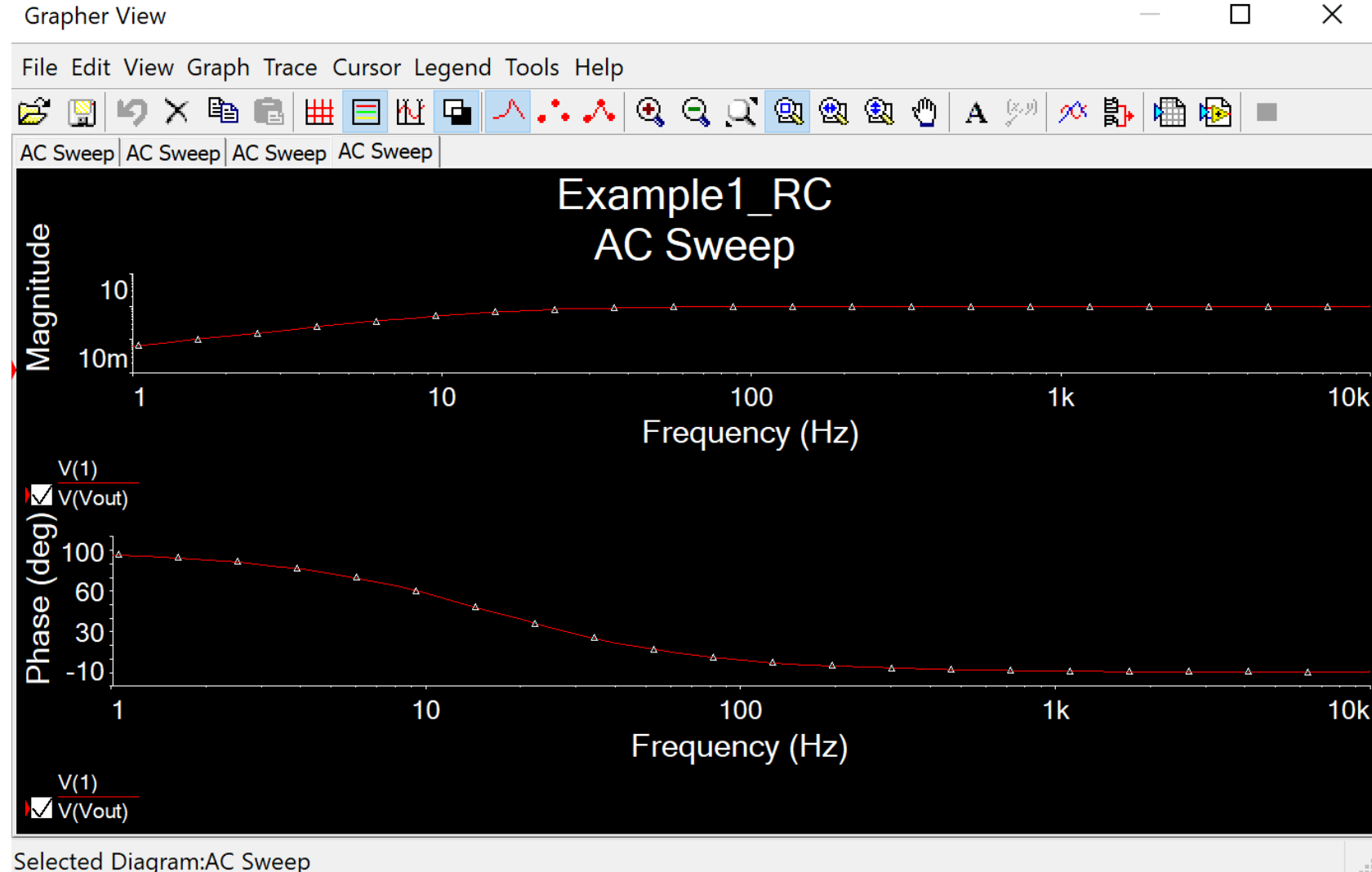

Fig. 1

Fig. 2

Exercise 1: RC Circuit Example

Perform simulation and display results as shown.

Horizontal axis is frequency from 1Hz to 10KHz in log scale

Vertical axis are probe values, i.e., magnitude and phase of Vout.

Exercise 2: DC Analysis of a BJT Circuit

Create the circuit schematic as shown in Fig. 1. The sources are DC voltage sources. We have renamed the two sources as VBB and VCC1. VBB is set to 3V, and VCC1 is set to 15V. The value of the resistors RB and RC are set to 230KΩ, and 5KΩ, respectively.

In "Analysis and Simulation" settings (Fig. 2), We have selected "DC Operating Point".

For variables of analysis (i.e., required solutions from simulation), we have selected collector current I(Q1[IC]), and all node voltages V(1), V(2), V(3), and V(4) (see Fig. 2)
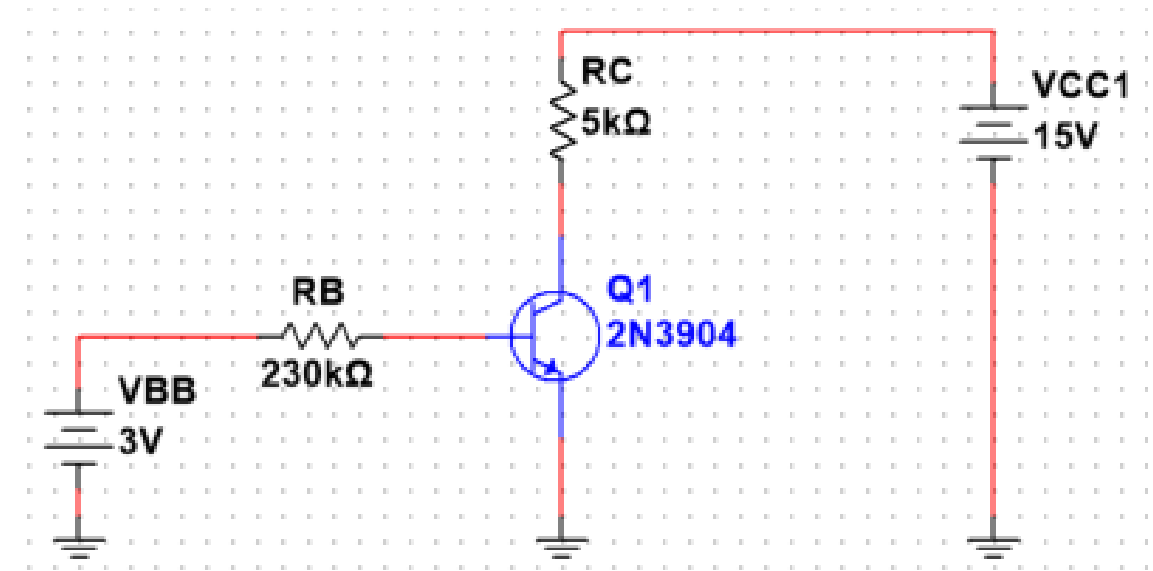
Then click the "Run" icon to start simulation
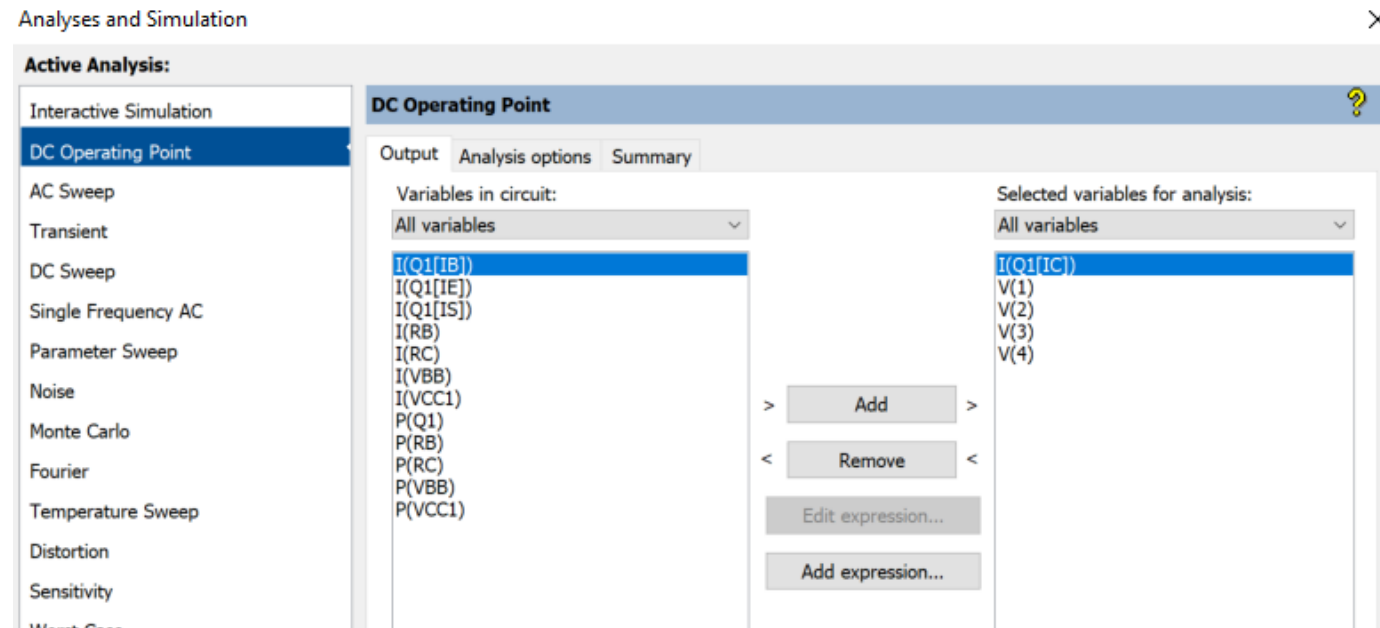

Fig. 1

Fig. 2

Exercise 2: DC Analysis of a BJT Circuit

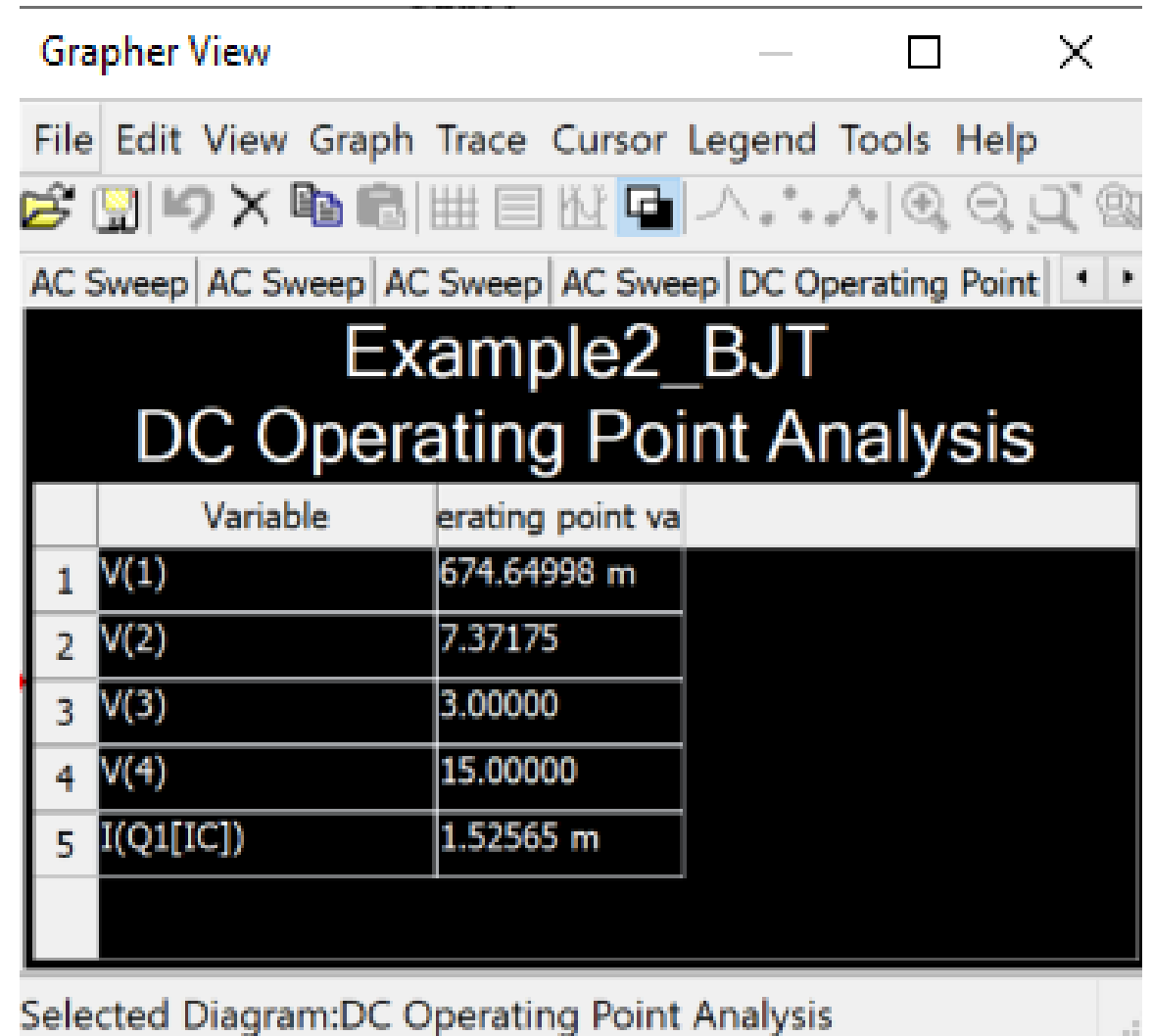Perform DC simulation and display results in the Grapher view, shown in the figure.

The values for all selected variables for analysis are listed in this Grapher view: I(Q1[IC]), V(1), V(2), V(3), and V(4).

Specifically, the solutions are:
    collector current = 1.52565mA
    collector voltage = 7.37175V
    base voltage = 0.67464998V

Grapher View

File Edit View Graph Trace Cursor Legend Tools Help

AC Sweep | AC Sweep | AC Sweep | AC Sweep | DC Operating Point

Example2_BJT
DC Operating Point Analysis

	Variable	erating point va
1	V(1)	674.64998 m
2	V(2)	7.37175
3	V(3)	3.00000
4	V(4)	15.00000
5	I(Q1[IC])	1.52565 m

Selected Diagram:DC Operating Point Analysis

Exercise 2: DC Analysis of a BJT Circuit

Alternative method to obtain collector voltage is to use multimeter.

Click Multimeter icon (see figure). Place multimeter into circuit, and connect it to the BJT's collector.

Double click the multimeter symbol. A multimeter-image will appear.

Select "Interactive" for the Analysis and Simulation setting.

Click "Run" icon to start simulation. Multimeter will show the reading of 7.372 V.

Click "Stop" icon to stop simulation.

Stop icon

Select "Interactive"

Multimeter icon

Double click the multimeter symbol, so that the multimeter image will appear

multimeter image