Electronic Circuit Simulation with LTspice
– A Hands-on Tutorial –

Karel L Sterckx, PhD
Bangkok University
Center of Research in Optoelectronics, Communications and Computational Systems
http://bucroccs.bu.ac.th
karel.s@bu.ac.th
Last Update: March 2020

An essential tool of modern electronic circuit design is electronic circuit simulation. Most simulators for analogue electronic circuitry are based on the open-source software SPICE, which stands for 'Simulation Program with Integrated Circuit Emphasis' and was developed at the University of California Berkeley (UCB). The first version of SPICE was released in 1973, and various companies have released commercial SPICE-based packages ever since. While there are commercial products that have a free *lite* version, these are usually severely restricted in number of components and nodes. This tutorial concentrates on LTspice, a widely used free tool that does not impose such restrictions. LTspice was developed by Linear Technology which, in March 2017, was acquired by Analog Devices. The initial version of the software was released in October 1999 and the latest stable release to date is LTspice XVII.
The tutorial explains the installation and operation of LTspice in a hands-on fashion. Before addressing these, however, the conventions that are used in this document will be explained. Since the tutorial relates to execution on the MS-Windows operating system, it is assumed that tutees are familiar with this operating system. Standard MS-Window operations such as the manipulation of windows (opening, closing, resizing etc.), therefore, are not detailed. The following conventions are used to denote software operation procedures:

- A left mouse button click, is referred to as 'click'
- A right mouse button click, is referred to as 'right click'
- Clicking and holding a mouse button is referred to as 'click-hold'
- Text in title bars, toolbars, buttons, windows, wizards and dialogues are written in bold font
  Examples: The **New Project Wizard** appears. Click the **Next** button.
- Names of menus, submenus and commands are written in bold-italic font. Accessing commands as menu items is denoted as a sequence of respective menu items separated by a right arrow
  Example: ***Menu Name*** → ***Submenu Name*** → ***Command Name***
- Highlighted text is information that needs to be entered, either through typing or selection

This tutorial relates to LTspice XVII. Where this version can be downloaded and how to install it is outlined in Section 1. How the software is operated is explained in Section 2, Section **Error! Reference source not found.** covers the importation of third party SPICE models into LTspice and, in Section 3, the user is directed to sources with extensive information of LTspice.

1. Download and Installation of LTspice XVII

This section outlines the download and installation of LTspice XVII. The procedure is as follows:

1) Visit the following URL:
   www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html
2) Scroll down to the orange button 'Download for Windows 7, 8 and 10', and click it.

3) Save the installation file 'LTspiceXVII.exe' to a desired location.
4) Click the 'Run' button
   Note: Alternatively, one may double click the installation file 'LTspiceXVII.exe'
5) The **LTspice XVII Installaer** wizard appears, which guides through the installation process.
6) **IMPORTANT:** Note that, somewhere during the installation process, you need to enter whether the processor of your computer is 64-bit or 32-bit. Make sure that you know the type your processor, and make the correct selection accordingly. The default choice is 64-bit.

2. LTspice XVII Hands-on

This section guides through the major aspects of LTspice XVII, i.e.

- Drawing a schematic circuit diagram
- Bias Point Analysis
- DC Analysis
- Parametric Sweep
- Transient Analysis
- AC Analysis
- Viewing simulation results

2.1. Drawing a schematic

7) If not already opened after installation, start LTspice by double clicking its icon on the desktop
8) Open a new schematic file by either
   - ***File → New Schematic***, or
   - Clicking the 'New Schematic' tool button. Note that the icon of the corresponding tool button is depicted in front of the tool's menu name.
9) Draw the circuit of Figure 1 below. The steps needed to draw this circuit are outlined underneath.

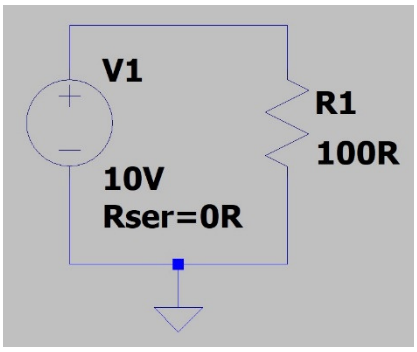


Figure 1: Resistor connected to a voltage source

VOLTAGE SOURCE

- Execute one of the following
  - ***Edit→ Component***, or
  - Click the 'Component' tool button
- In the **Select Component Symbol** dialogue that appears, select voltage as indicated in Figure 2 below, and click the **OK** button.
- Place the voltage source on the drawing area by clicking the left mouse button
- Click the right mouse button, or press the 'Esc' key, to switch from component placing mode to selection mode

- Move the cursor over the component. Note that the cursor changes from a cross shape into a hand with a pointing finger.
- Click the right mouse button, or press the ‘Esc’ key, to open the **Voltage Source** dialogue and set (via typing) the value of the DC voltage to 10V and the value of the series resistance, which is the internal resistance of the voltage source, to 0R.
- Note that the units V and R (which stands for Ω) do not need to be included for simulation purposes. For clarity, however, it is advisable to include the unit.
- Click the **OK** button

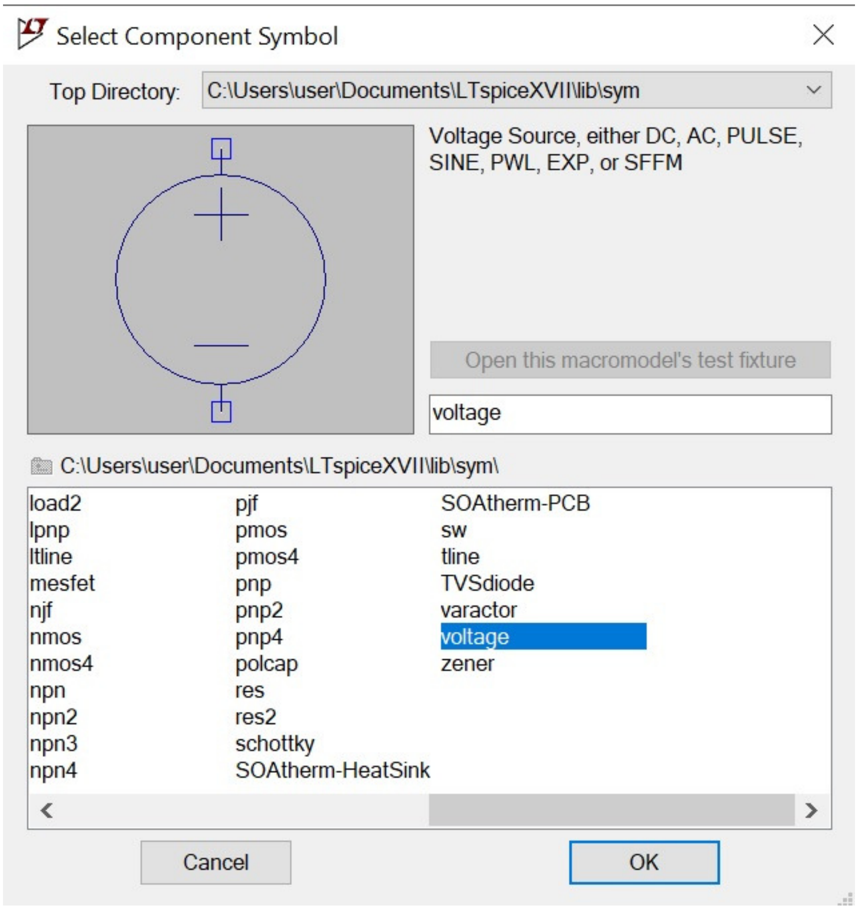
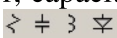


Figure 2: ‘Select Component Symbol’ dialogue

RESISTOR

- Commonly used components like resistor, capacitor, inductor and diode have a shortcut tool button to place the components, i.e. 
- Click the ‘Resistor’ tool button
- Before placing the component, let’s experiment with the alignment of the component, which needs to be executed before the component is placed.
  - Pressing the key ‘Ctrl’ + ‘R’ will *rotate* the component by 90 degrees clockwise
  - Pressing the keys ‘Ctrl’ + ‘E’ will *mirror* the component
- Place the component by clicking the left mouse button
- Click the right mouse button, or press the ‘Esc’ key, to switch from component placing mode to selection mode
- Move the cursor over the resistor. Note that the cursor changes from a cross shape into a hand with a pointing finger.
- Click the right mouse button, or press the ‘Esc’ key, to open the **Enter new Value** dialogue and set the resistance value of to 100R.

- Click the **OK** button
  Important: LTspice recognises the standard SPICE scaling symbols as displayed in Table 1 below.

Scale	SPICE Symbol	Name
10^{-15}	f	femto
10^{-12}	p	pico
10^{-9}	n	nano
10^{-6}	u	micro
10^{-3}	m	milli
10^{3}	k	kilo
10^{6}	MEG	mega
10^{9}	G	giga
10^{12}	T	tera

Table 1: SPICE Scaling Symbols

  Any other character written after a numerical value is ignored and has no meaning. For example: 100, 100R and 100E are recognised as 100Ω if the component is a resistor. Even 100V will be recognised as 100Ω. Note that the characters R and E are often used to denote Ω in electronic design software that does not allow the use of Greek characters. For reason of consistency use either. The SPICE symbols are case insensitive. The cases used in the Table 1 below follow the convention of scientific notation. The fact that the SPICE symbols are case insensitive explains why Mega is denoted as MEG.

Listed below or operations to edit a component, or part thereof. In the list below, the edit operations are invoked by means of a tool button. Alternatively, these edit operations can be selected from the **Edit** menu. The icon in front of each edit operation name matches the icon on the corresponding tool button.

- To *delete* a component
  - Press the 'Delete' key of the keyboard
  - The cursor changes into a pair of scissors, indicating delete mode
  - Clicking on a component whilst in delete mode will delete the component
  - To exit the delete mode, click the right mouse button, or press the 'Esc' key,
- To *move* a component
  - Click the 'Move' tool button, or press the 'F7' key
  - The cursor changes into the hand icon of the 'Move' tool button, indicating move mode
  - Clicking on a component symbol whilst in move mode allows the component to be moved
  - Clicking on the designator or the value of a component in move mode allows for that designator or value to be moved, respectively.
  - When a component, a designator or a value are able to move, they are also able to be rotated or mirrored.
  - To exit move mode, click the right mouse button, or press the 'Esc' key,
- To *duplicate* a component
  - Click the 'Copy' tool button, or press the 'F6' key
  - The cursor changes into the icon of the 'Copy' tool button, indicating copy mode
  - Whilst in copy mode, clicking on a component, its designator or its value, allows for duplication of the component.

- To exit copy mode, click the right mouse button, or press the ‘Esc’ key,
- To *undo* or *redo* a previous action
  - Click the ‘Undo’ tool button undo a previous action
  - Click the ‘Redo’ tool button redo a previous action

GROUND

- Click the ‘Ground’ tool button, or ***Edit → Place GND***, and place the ground in a fashion similar to how the voltage source and resistor were placed

WIRING

- Click the ‘Wire’ tool button, or ***Edit → Draw Wire***
- The cursor changes to a large cross, indicating the wire drawing mode
- Wires are added by clicking on the start and end point of a wire. There is no need to hold down the mouse button when moving between the two connection points. Junctions are added automatically when a wire ends at a point somewhere along the length of another wire.
- Add all wires to complete the circuit of Figure 1
- Note: To facilitate drawing convenience, make use of the zoom commands. To invoke these commands, click the relevant zoom tool button, or select the desired zoom command from the **View** menu.
- To exit the wire drawing mode, click the right mouse button, or press the ‘Esc’ key,

10) Save the schematic by executing one of the following
  - Click the ‘Save’ tool button, or
  - Press the Ctrl+S keys, or
  - Click ***File → Save***
11) Save the file as ‘Circuit-01’ in a location of choice. Advice: Save the design regularly.

2.2. Bias point analysis

12) Edit → SPICE Analysis
13) The **Edit Simulation Command** dialogue opens
14) Select the **DC op pnt** tab, where ‘op pnt stands for operation point (= bias point), and click the **OK** button
15) Place the text **.op** that appears to the left of the schematic as shown in Figure 3, and click the left mouse button. This invokes Bias point (or operating point) analysis.
16) Execute one of the following:
  - ***Simulate → Run***, or
  - Click the ‘Run’ tool button
17) A popup box appears, showing the currents through the various components, and the voltages at each node in relation to the signal ground.
18) Close the popup box
19) Move the cursor, which should have the shape of a small cross (if not, click the right mouse button or press the ‘Esc’ key), close to any wire that is not connected to the signal ground, and click and hold the left mouse button. Whilst holding the left mouse button, the cursor changes into a measurement probe as displayed in Figure 3.
20) Release the left mouse button and observe that ‘10V’ is being displayed in blue, indicating that the voltage of the wire in relation to the signal ground is 10 V.
21) Right click on this blue ‘10V’, which opens the **Displayed Data** dialogue
22) Under **Edit expression to display below**, delete the dollar sign $.
23) Under **Available data**, click **I(R1)**. The **Displayed Data** dialogue should now look as displayed in Figure 4 below.

24) Click the **OK** button. The value of the current flowing through the wire in the direction of R1 is now being displayed.

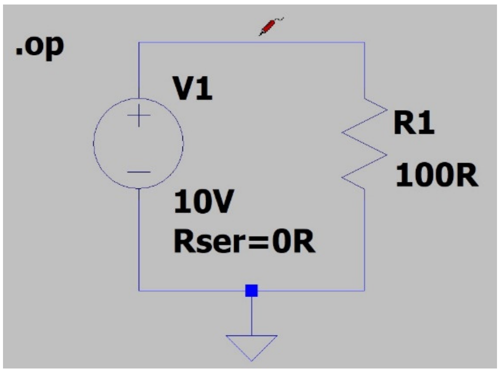


Figure 3: Placing a bias probe on a wire

25) Delete the text **.op** and the measured current value (= blue '100mA'). Note that the text **.op** can only be deleted after all measured current and voltage values, i.e. currents and voltages displayed in blue font, have been deleted.

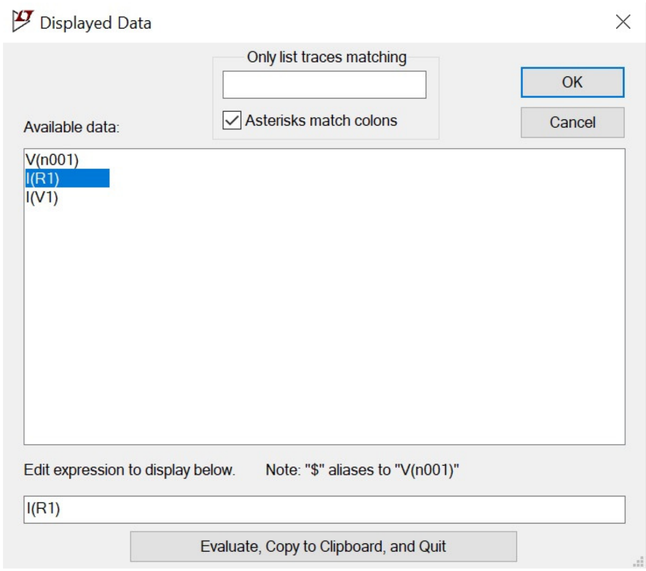


Figure 4: 'Displayed Data' dialogue

2.3. DC analysis

26) ***Edit → SPICE Analysis***
27) The **Edit Simulation Command** dialogue opens
28) Select the **DC sweep** tab
29) In the **1st source** tab, fill out and select the values as displayed in Figure 5 below
30) Click the **OK** button
31) Place the text **.dc V1 0 5 1** that appears to the left of the schematic as shown in Figure 6, and click the left mouse button. Now, a DC sweep will be executed when simulation is run.
32) Click the 'Run' tool button
33) The Waveform Viewer appears on top of the Schematics Editor
34) Select the Schematics Editor, and move the cursor anywhere near the wire that connects V1 with R1. Observe that the cursor changes into a measurement probe that symbolises a voltage probe.
35) Click the left mouse button. The Waveform Viewer now displays the voltage at Node 001.

36) Right click in the Waveform Viewer. In the menu that pops up, select ***View* → *Mark Data Points***, and observe what happens to the displayed graph.

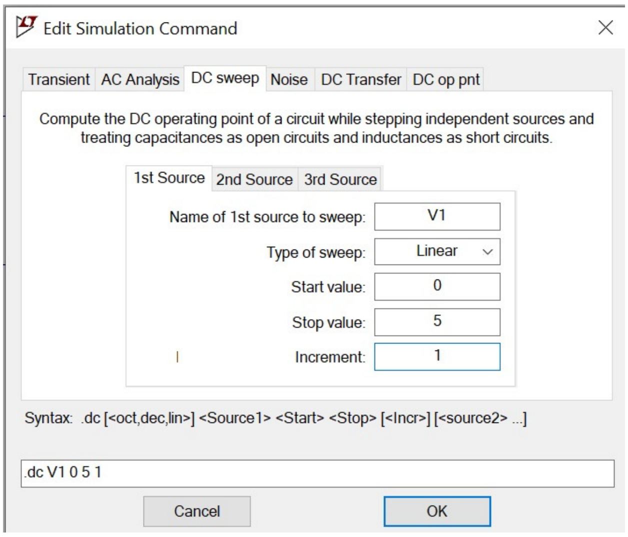


Figure 5: DC Sweep Setup

37) Select the Schematics Editor, and move the cursor over R1. Observe that the cursor changes into a current probe whereby the red arrow indicates the direction of the positive current.
38) Click the left mouse button. The Waveform Viewer now also displays the current through R1.
39) To add certain features to the Waveform Viewer, execute the operations listed below
- To add a grid: Right click in the Waveform Viewer and select ***View* → *Grid***
- To modify the appearance of a trace, move the cursor over its name. When the cursor changes into a hand with pointing finger, click the right mouse button. The appearance of the trace can be changed by means of the popup box that appears.
- When moving the mouse cursor over either the horizontal or vertical or axis, the cursor turns into a ruler, indicating that the attributes of that axis can be changed. Clicking the right mouse button opens a dialogue to manually enter the range and the nature (i.e. linear or logarithmic) of the axis
- To add cursors to a trace, move the cursor over the name of the trace to which a cursor needs to be added. When the cursor changes into a hand with pointing finger, click the left mouse button. The cursor can be moved via its vertical line. When moving the cursor, the number of the cursor appears. Up to two cursors can be added.

40) Close the Waveform Viewer and maximise the Schematics Editor

2.4. Parametric sweep

The previous section demonstrated the execution of a sweep of discrete voltage values. For every voltage value in the sweep, a bias point calculation was executed. This section will execute a second sweep, namely a sweep of discrete resistor values.

41) Right click on the value of the resistor R1 and, in the dialogue that appears, change this value from 100R to {R}, which indicates that this parameter will be specified by means of a sweep
42) Execute one of the following to insert a SPICE directive
- ***Edit* → *SPICE Directive***, or
- Press the 's' key

43) In the text box type that appears, type .step param R list 47k 100k 220k, which means that the DC analysis will now be carried out for resistor values of 47kΩ, 100kΩ, and 220kΩ, respectively
44) Click the **OK** button and place the directive under the .dc directive as shown in Figure 6 below

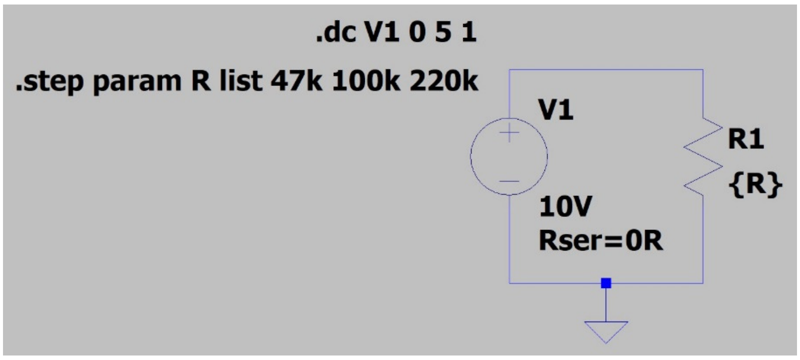


Figure 6: Parametric Sweep Setup

45) Click the 'Run' tool button
46) Place a current probe over R1 and observe what happens to the Waveform Viewer. It should now display three curves that correspond to the graphs for R=47k (red), R=100k (blue), and R=220k (green), respectively.
47) To save the Waveform Viewer display as a file that can be inserted into any MS-Office application, ***Tools*** → ***Write image to .emf file***
48) Close the Schematic Editor, which will also close the Waveform Viewer. Click **Yes** if asked whether changes should be saved.

2.5. Transient analysis

49) Open a new schematic file, and draw the circuit of Figure 7 below

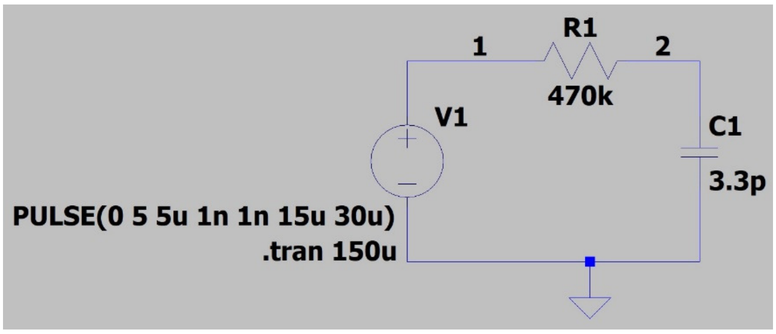


Figure 7: Low Pass Filter

50) Drawing R1 and C1 should be straightforward using the 'Resistor' and 'Capacitor' tool buttons, respectively.
51) The numbers 1 and 2 to the left and right of R1, respectively, are node numbers, which are added by means of the 'Node' tool button or ***Edit*** → ***Label Net***. Note that the signal ground is automatically set as Node 0.
52) Placing the voltage source is similar to placing the source in the circuit of Figure 1. Its setup, however, is different, and needs to be carried out as follows:
   - After placing the voltage source, move the cursor over the component. Note that the cursor changes from a cross shape into a hand with a pointing finger.
   - Click the right mouse button to open the **Voltage Source** dialogue

- Click the **Advanced** button to open the **Independent Voltage Source** dialogue
- Select **PULSE**, which allows for the specification of a block wave, and fill out the parameters as displayed in Figure 8 below.

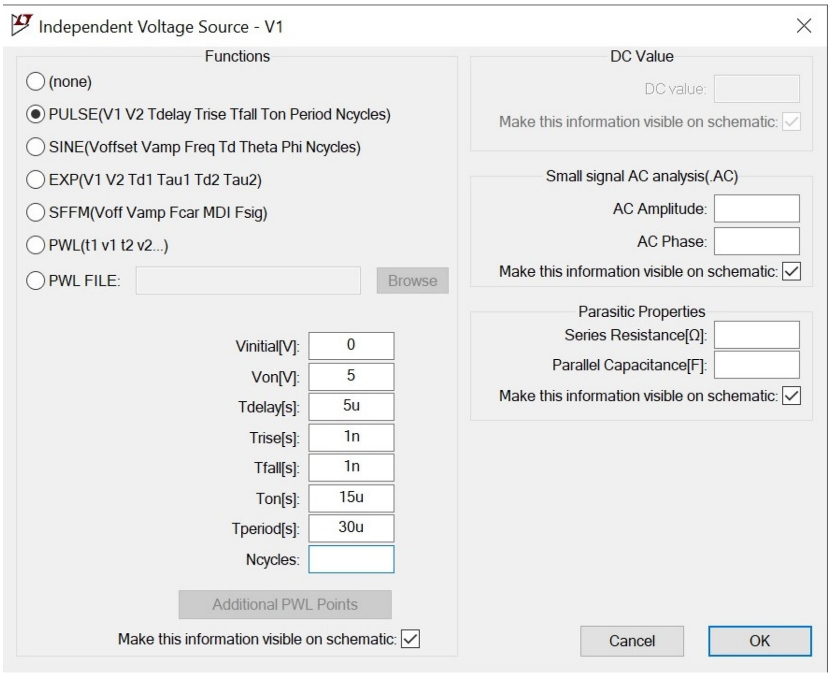


Figure 8: Setting Up a Block Wave

Note: Parameters explained
- Vinitial: Value before the block wave starts
- Von: High level of the block wave
- Tdelay: Time before the block wave starts
- Trise and Tfall: Rise and fall time, respectively. Note that these values cannot be set to 0 as they would cause a singularity. To cause minimal simulation error, the value needs to be set small. Note also that the smaller the value, the longer the simulation will take.
- Ton: Time that the block wave stays high
- Tperiod: Period of the block wave
- Ncycles: Number of cycles that will be displayed. There is no need to specify these as this will be set by the stop time of the transient analysis

- Click the **OK** button

53) Save the schematic as 'Circuit-02'
54) ***Edit → SPICE Analysis***
55) The **Edit Simulation Command** dialogue opens
56) Select the **Transient** tab
57) In the **Stop Time** box, type 150u
Note: Transient analysis explained
The transient analysis calculates the behaviour of a circuit over time, namely from time zero to an end value (specified under **Stop Time**). The transient analysis is using an internal time step that is adjusted as the analysis proceeds. In intervals where there is little change, the internal step is increased and during intervals where there is a lot of change, the internal step is decreased. This is done automatically. The only control the user has is to specify the maximum time interval between two subsequent calculation points, which is accomplished via the **Maximum Timestep** text box. The default value of the maximum time interval

equals the end value divided by 50. If the default value is used, the **Maximum Timestep** text box may be left blank.

58) Click the **OK** button.

59) Place the text **.transient 150u** that appears to the left of the schematic. When simulation is run, a transient analysis will be executed over a period of 150 µs.

60) Click the ‘Run’ tool button

61) To display the voltage over R1

- Move the cursor to the wire on the left of R1, i.e. Node 1
- When the cursor changes into a red voltage probe, click and hold on its wire
- Drag the mouse to the wire on the right of R1, i.e. Node 2
- When the cursor changes into a black voltage probe, release the left mouse button.

62) Display the voltages over V1 and C1 in a similar fashion. Start at Node 1 and Node 2, respectively, and end at Node 0 (= signal ground) in both cases.
Note: Clicking and releasing the red voltage probe is sufficient when the second node is the signal ground.

63) Delete the signal trace of V1

- Select the Waveform Viewer
- Press the ‘Delete’ key, which turns the cursor into a pair of scissors
- Click on the designator V(1), which stands for Voltage at Node 1 (in respect to Node 0)
- Exit the delete mode by clicking the right mouse button, or press the ‘Esc’ key,

64) Close the Waveform Viewer and maximise the Schematics Editor

65) Move the cursor over V1 and when it changes into a hand with a pointing finger, click the right mouse button to open the **Voltage Source** dialogue

66) Click the **Advanced** button to open the **Independent Voltage Source** dialogue

67) Select **SINE**, which allows for the specification of a sine wave, and fill out the parameters as displayed in Figure 9

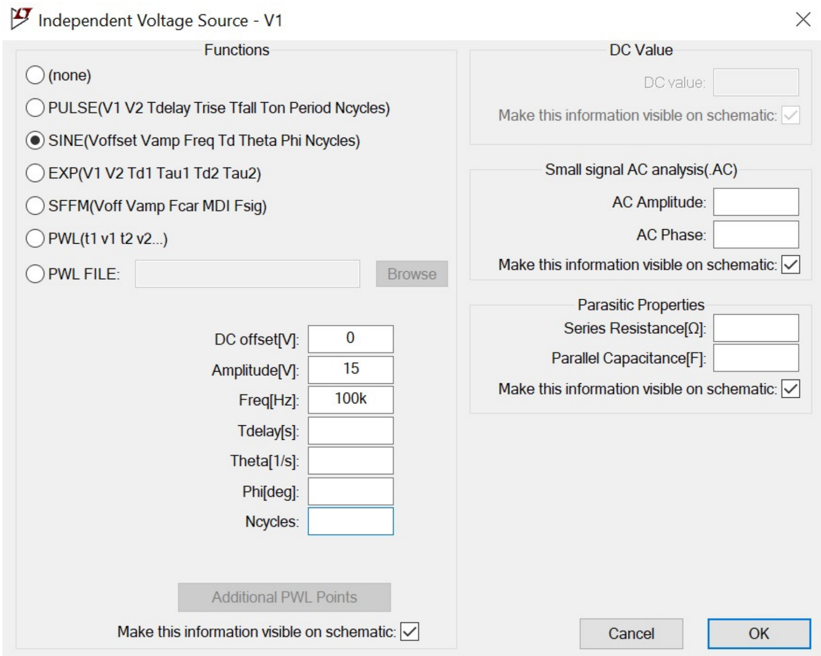


Figure 9: Setting Up a Sine Wave

68) Click the **OK** button

69) Change the stop time of the transient analysis from 150 µs to 50 µs

70) Click the ‘Run’ tool button

71) Display the voltages over V1, R1 and C1 as outlined in Steps 61) and 62)

2.6. AC analysis

72) Close the Waveform Viewer and maximise the Schematics Editor
73) Move the cursor over V1 and when it changes into a hand with a pointing finger, click the right mouse button to open the **Independent Voltage Source** dialogue
74) Under Small signal AC analysis (AC), in the AC Amplitude text box, type 10
75) Click the **OK** button
76) ***Edit → SPICE Analysis***
77) The **Edit Simulation Command** dialogue opens
78) Select the **AC Analysis** tab, and fill out the parameters as displayed in Figure 10 below

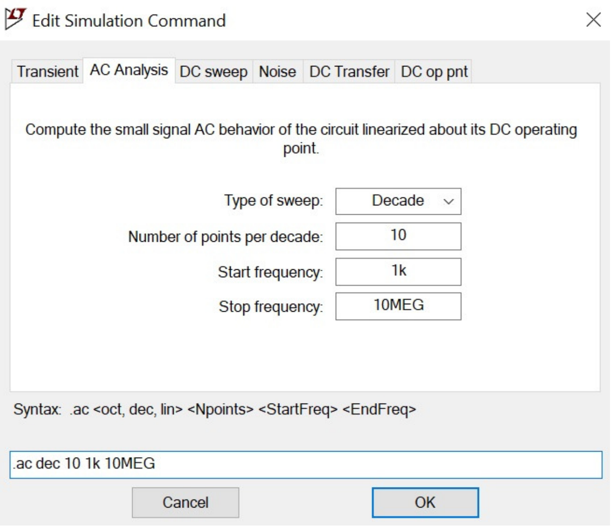


Figure 10: AC Sweep Setup

79) Click the **OK** button, and place the text **.ac dec 10 1k 10MEG** underneath the text **.tran 50u**
80) Note that a semi colon (;) appears in front of the **;tran 50u** directive, indicating that the transient analysis has been disabled
81) Click the 'Run' tool button
82) Display the voltage over C1
83) Note that both the magnitude and phase of V(2) (= V_{C1}) are displayed. To display these graphs in two separate plots, execute as follows:
    - Right click anywhere in the Waveform Viewer
    - In the popup menu that appears, select **Add Plot Pane**
    - Note that a second plot appears above the plot of V(2)
    - Once again, display the voltage over C1, which should now also appear in the upper plot
    - Move the cursor to the right axis of the upper plot. When the cursor changes into a ruler, click the right mouse button.
    - In the **Right Vertical Axis** dialogue that appears, click the **Don't plot phase** button. The upper plot should now only display the magnitude
    - Move the cursor to the left axis of the lower plot. When the cursor changes into a ruler, click the right mouse button.
    - In the **Left Vertical Axis** dialogue that appears, click the **Don't plot magnitude** button. The lower plot should now only display the phase.

3. Further Reading

For more extensive information on LTspice, users are directed to the following two resources:

- A collection of technical articles and videos related to LTspice, which can be accessed from the LTspice home page, i.e. www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html, underneath ‘Technical Articles & Videos’
- A users' group dedicated to LTspice, which can be reached at https://groups.io/g/LTspice