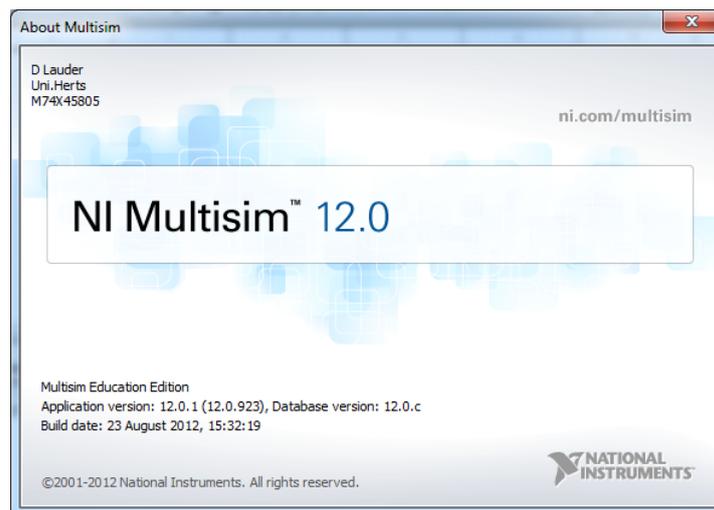


An Introductory Guide to Circuit Simulation using NI Multisim 12



This booklet belongs to: _____

This document provides a brief overview and introductory tutorial for circuit simulation using NI Multisim Version 12 and later. Multisim is currently installed on all the PCs in labs D411 & D421, Projects Lab, C406 and elsewhere but may not be available in the LRC.

The National Instruments website contains full details of Multisim
<http://www.ni.com/multisim/>

A student version can be purchased (version 13).

<http://www.ni.com/multisim/try/>

Contents

	page
1 OVERVIEW	3
1.1 Introduction	3
1.2 Circuit Simulation	3
1.2.1 DC Analysis	3
1.2.2 AC Analysis	3
1.2.3 Transient Analysis	4
1.2.4 Network Topology	4
1.2.5 Branches & Nodes	4
1.2.6 Active Devices	4
2 MULTISIM TUTORIAL	5
2.1 Getting Started	5
2.2 Circuit Analysis	7
2.3 Using the Cursors	8
2.4 Transient Analysis	9
2.5 Further investigations	10

1 Overview

This guide introduces some of the main features of the Multisim analogue and digital simulation software which is currently installed on PCs in the School of Engineering and Technology. This initial chapter provides a basic overview of circuit simulation. Chapter 2 describes an introductory tutorial to Multisim.

1.1 Introduction

Multisim is an analogue and digital circuit simulator that has similarities with Altium Designer, TINA and PSpice. These programs all calculate the voltages and currents of a circuit under a variety of different conditions. The first analogue simulation program SPICE was developed at the University of California at Berkeley in the early 1970s and has become a *de facto* standard in the area of analogue simulation. Multisim uses 'SPICE-like' models to describe the behaviour of all network elements.

Computer-based circuit analysis is a powerful alternative to 'bread-boarding' and extensive circuit measurements. The circuit under test can be modified on the computer to obtain revised response predictions. Once optimised, the circuit may be built and tested against these predicted results with a considerable reduction in design cost overheads.

Multisim includes a schematic capture 'front-end', which means that you can 'draw' a circuit diagram on-screen and then perform your analyses directly from this 'captured' schematic. You may also copy files and images into other Windows compatible programs (e.g. Microsoft Word).

The various analysis tools provide the 'number crunching' part of the package and will compute all nodal voltages and branch currents for a wide range of DC, AC and transient conditions. All these results are then used to plot waveforms which are displayed in a separate 'Diagram Window' or viewer which works interactively with the results of an analysis. If you think of Multisim as a 'software breadboard' then think of Diagram Window as the 'software oscilloscope', although that analogy is only a very small part of the tools Multisim provides you with.

1.2 Circuit Simulation

Three main classes of analysis exist;

1. DC or steady-state
2. AC or frequency domain
3. TRANSIENT or time-domain

1.2.1 DC Analysis

D.C. Analysis is used to determine d.c. steady-state conditions, with all inductances shorted and capacitances open circuit. It is particularly valuable in non-linear circuits or when making repetitive calculations with variable components. Since temperature affects semiconductor devices, this can also be taken into account.

The sensitivity of bias conditions to component changes is of great interest to circuit designers when considering the effect of value spreads on production runs. Multisim provides a range of analyses, optimization, transfer and temperature to list a few, to aid the designer.

1.2.2 AC Analysis

This is mainly used for frequency response and all components are assumed to be linear. Active devices are replaced by small-signal models. The parameters of the model are calculated at the operating point, so a d.c. analysis must be carried out first. The program knows that small-signal parameters vary with the d.c. conditions. The analysis follows the same pattern as linear d.c. analysis with the extra complication of complex numbers. The output from an a.c. analysis is conveniently a graph, perhaps a Bode diagram of gain & phase.

1.2.3 Transient Analysis

This is the most complicated type of analysis, because it involves reactive components, non-linear components, and time variation. The CPU time consumed is therefore greater than in d.c. or a.c. analysis. Usually, a d.c. analysis is necessary to establish the initial conditions in a non-linear circuit. The output indicates the time variation of a circuit response, probably in graph form.

The circuit excitation must also be specified. This may be an algebraic function, or a piecewise-linear function such as a ramp or square wave. The simple 'step' function is the most used example.

The non-linear distortion in a transistor amplifier is determined by exciting the circuit with a sine-wave. The Fourier coefficients of the resultant output may also be calculated.

The behaviour of the circuit in the time domain is plotted by moving forward in iteration steps and at each step calculating the change in capacitor charge and inductor current. Thus every step requires a new non-linear iterative analysis. The process may become so slow that the determination of a suitable iteration step is of prime concern.

Warning; poorly specified analysis commands can cause excessively long computational runs often producing misleading results. Always consider your analysis carefully, are the predictions what you would expect?

1.2.4 Network Topology

In order to calculate all the currents in a simple resistance network it is necessary to define equations based on Kirchhoff's Laws and then solve them. This is done by the use of topological matrices and matrix-handling routines which are easy to use in computer programs, but would be daunting on paper! The matrices which define the topology of the network are derived easily from the input data. The results of the calculations are held in vectors, and those requested by the user are printed out.

1.2.5 Branches & Nodes

A *branch* is either a single circuit element, or a simple group of elements having a node at each end. Branch currents and branch voltages are important variables in network analysis.

A *node* is a point where branches join. Generally, one node is designated 'ground' and is numbered '0'. Your circuit should *always* include a ground node using the 'earth' symbol. SPICE based analysis programs such as Multisim will probably generate an error message if there is no ground node. Node voltages are measured relative to ground and are of most use to the designer.

The topology of a network is a line drawing of the nodes and branches. The nodes and branches are given arbitrary numbers, but the direction of a branch is clearly of importance. The network must be 'continuous', i.e. there must be a path from every node to ground. There are never more 'live' nodes than branches.

1.2.6 Active Devices

Three terminal devices such as transistors are always replaced in the program by equivalent circuits which consist of controlled sources and passive or non-linear elements. The particular model chosen depends on the type of analysis.

Where semiconductor and other non-linear devices are involved, the amount of computing effort is greatly increased. The characteristic of the device has to be defined by the user, part of the process of 'modelling'. It is generally satisfactory to model diodes and BJTs by their exponential (Ebers-Moll) equations and similar equations exist for FETs.

Where a large and complex device such as a monolithic circuit (i.e. an operational amplifier) is concerned, it is usual to use a *macro-model*, that is a circuit which has the same characteristics as the real device but which has far fewer components. The components of the macro-model are idealised, and could not exist as hardware.

2 Multisim Tutorial

This tutorial introduces you to the three key steps to creating analogue circuit simulations;

- creating a circuit diagram and selecting analysis options using the *schematic editor*,
- selecting a range of *analysis options* to simulate circuit performance,
- displaying the results within the "*Grapher*" Window.

You will begin by investigating some simple passive circuits which may then be edited to create more complex networks. These exercises may be carried out using any computer in rooms D411, D421 or C460 (projects lab).

It has been assumed that you are familiar with the Windows 7 Operating System environment. You are expected to have your own USB memory stick for backups. You can also use your student network U: Drive

Note: the 'enter' or 'return' key is represented in these notes using the '↵' symbol.

2.1 Getting Started

Switch on the computer and login to the network. Once the Windows 7 desktop appears, launch Multisim from the *start* menu using:

Start > All Programs > National Instruments > Circuit Design Suite 12.0 > Multisim 12.0
(You may see a later version). Alternatively, *Start > Search programs and files* and type *Multisim*

The Multisim User Interface window should now open as shown in figure 1 below.

Before going any further, save your work to an appropriately named file such as RC_1.ms12 Note exactly where you are saving the file so that you can retrieve it later. Save to your student network U: Drive if possible or a USB stick. These simple .ms12 files are quite small.

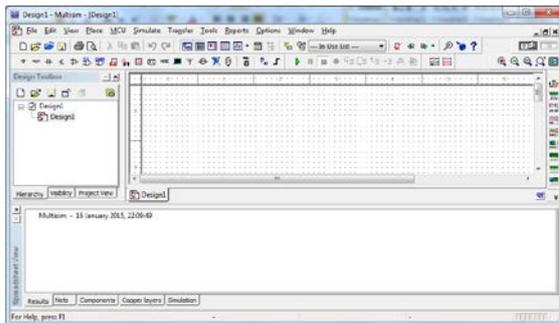


Figure 1

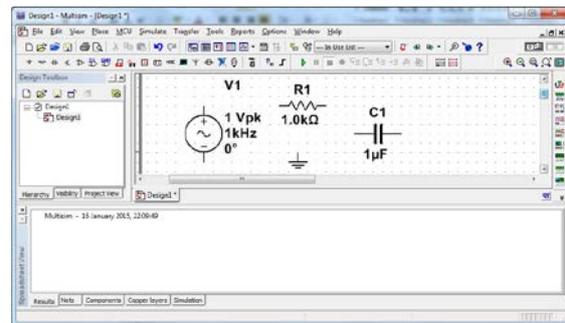


Figure 2

The first circuit you will learn how to simulate is a simple R-C filter network. In the Design Window, place four circuit elements into your schematic as shown in figure 2. You are selecting a resistor, capacitor, AC voltage generator and ground (or earth) symbol.

1. Move the mouse along the component toolbar and find pop-up text **Place Source and Place Basic** (These are the "earth symbol" and "resistor" buttons on the **toolbar**, not the top menu)
 - Place Source** > POWER_SOURCES > GROUND > OK
 - Place the ground symbol then
 - Place Source** > SIGNAL_VOLTAGE_SOURCES > AC_VOLTAGE > OK
 - Place the AC voltage source then close the "Select a component" window.
 - Place Basic** > RESISTOR > OK
 - Place the resistor then
 - Place Basic** > CAPACITOR > OK
 - Place the capacitor then close the "Select a component" window.
2. Select each component and position it on the screen in the order shown in figure 3 below. To move component symbols around, hold down the left mouse button whilst you move the

component. Note that a blue dashed box appears around the component when it is selected. C1 should be placed fairly low down as shown, leaving space above to add a probe.

Note also that the capacitor C1 in Fig 3 has been rotated through 90°. Select the component with the right mouse button then select 'rotate 90 degrees clockwise'. The component rotates but the component identification does not rotate. A text box has also been added. From the top menu **Place > Text**.

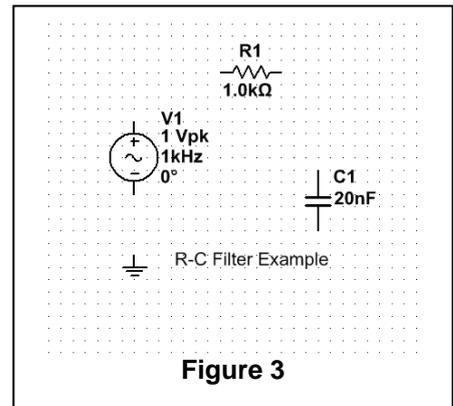


Figure 3

- Now you need to 'wire' all the components together as illustrated in figure 4. The auto wire tool starts as soon as you move the mouse over any component node. Click the mouse to start a wire then move to another node and click again to end the wire. With a little practice you can easily join components together but always do a final click when you get to the node at which you need to stop wiring. You can cancel an unwanted wire with the **Esc** key.

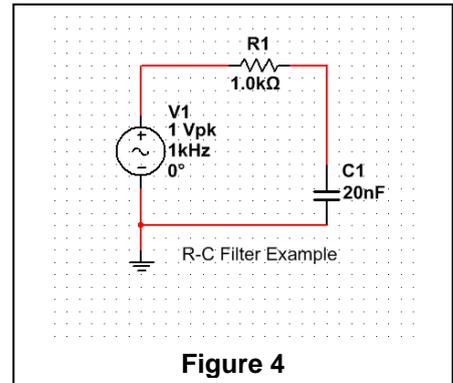


Figure 4

- The components have default values that may need to be changed. Double click the resistor symbol to open a dialog box. On the **Value** tab, the value should be **1k**. This is the default value so there is no need to change it. The resistor should be labelled **R1**. If not, then change the **RefDes** (Reference Designator) on the **Label** tab. Click **OK** to exit the dialogue box.

For the capacitor, change the **value** to 20 nF and check that the **RefDes** is **C1**. The component properties should look like figure 5(a) and 5(b) below.

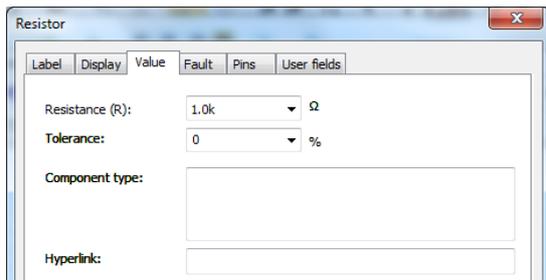


Figure 5(a) R setup

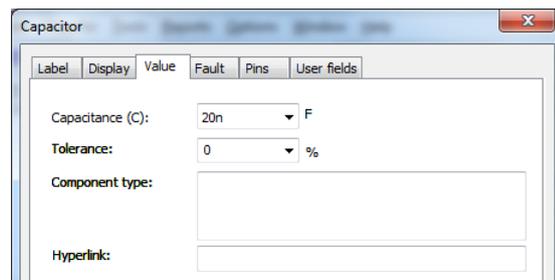


Figure 5(b) C setup

- Finally, double click on the AC voltage generator symbol to open the AC_VOLTAGE dialog box, as in figure 6 below. Check that **Voltage (pk): 1 volt, Frequency (F): 1 kHz**. These are the default values so you don't need to change them at the moment.
- Extensive HELP files are accessible from the "?" icon in the Multisim window. Investigate the **Help** menu before continuing with the next part of this tutorial.

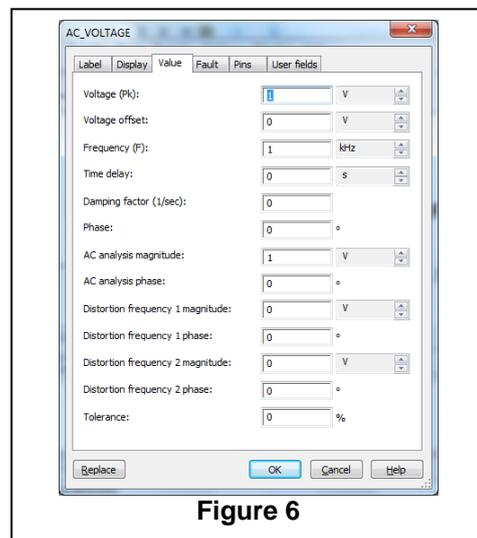
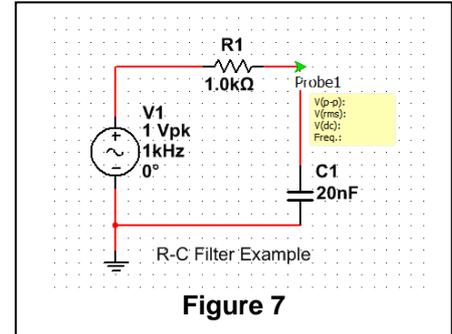


Figure 6

2.2 Circuit Analysis

We start by using the Analysis simulation mode in Multisim, not the virtual instruments. First, we need to add suitable ‘probes’. These gather information about the voltage, current or phase of the signals at the nodes they are connected to. This data can then be used to generate graphical plots.

- Figure 7 shows our first-order RC low pass filter circuit complete with a 1 volt peak sinusoidal AC source. An AC voltage probe has been added to the node where R1 joins C1. To do this on your schematic first find the double down arrow symbol on the right hand edge of the screen that has a pop-up label **More buttons** Then select **More Buttons > Measurement Probe > AC voltage** Place the measurement probe on the corner of the wire that joins R1 to C1 and click the mouse to connect the probe to the wire.



- A frequency domain simulation provides data about the frequency response of the circuit. First the analysis options must be set, so start from the Multisim main menu and select **Simulate > Analyses > AC Analysis > Frequency parameters** This should open a dialog box as shown in figure 8. Set **Start frequency: 100 Hz, Stop frequency 1 MHz, sweep type Decade, number of points per decade: 10, Vertical scale: Decibel** as shown. Then click on **OK**.

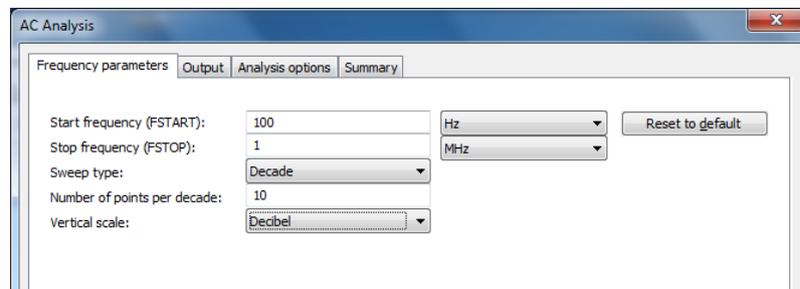


Figure 8

- You need to select which node to analyse. **Simulate > Analyses > AC Analysis > Output** as shown in **Figure 9**.

This lists node voltages, component currents and phases. We need the voltage at Probe1. Find **V(probe1)** in the list of all variables on the left.

Click **V(probe1)** to select it then **> Add >** to move it to the list on the right. When you have done this, it should look like Figure 9.

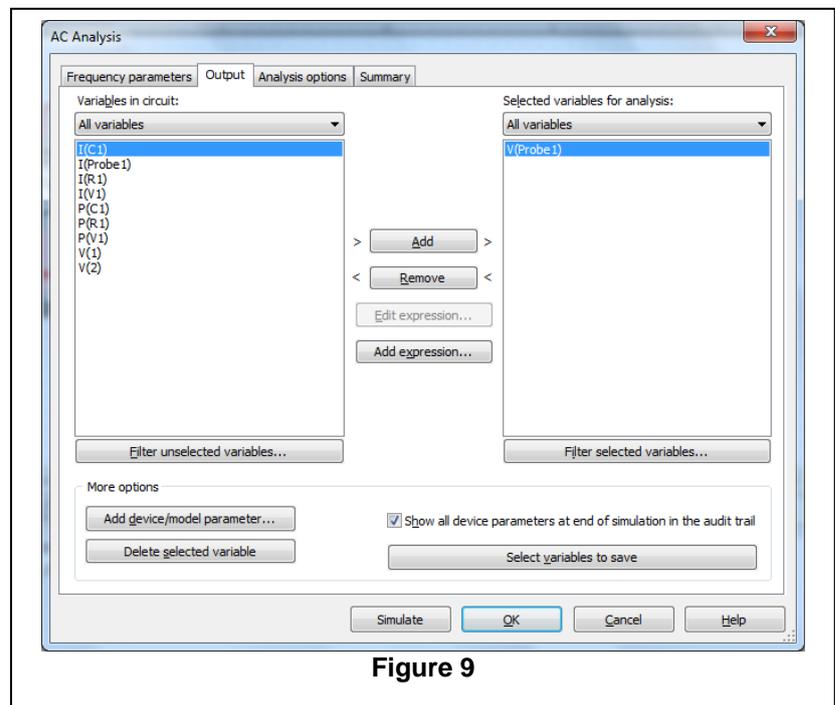


Figure 9

- Click Simulate. The analysis runs and a "Grapher View" window opens automatically to show a frequency and phase response or Bode Plot, figure 10. Use the appropriate buttons to change the black background to white and to turn on grid lines for both graphs.

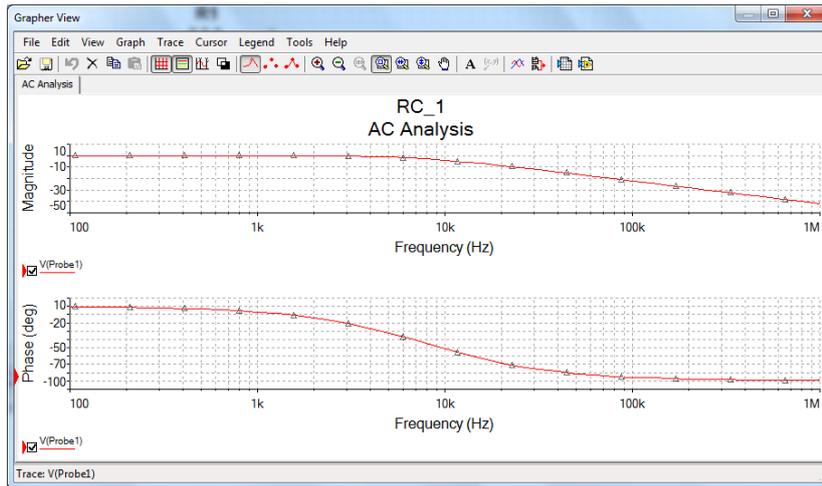


Figure 10: "Grapher" View – Bode Plot

2.3 Using the Cursors

In order to read accurate 'X' and 'Y' values from a trace, it is necessary to use the **cursors**. By placing a cursor on a trace, you can move it along the trace and read off the 'X' and 'Y' values at any point. Explore the menu items in the Grapher Window, especially **Cursor**. From the initial display (figure 10) let's add cursors as shown in Figure 11 below.

- Select the top trace (frequency response curve) then **Cursor > Show Cursors**. Two cursors appear on top of each other at the lowest frequency. Slide one of these to the right. Cursor 1 is red and Cursor 2 is blue; you can use either one. A pop-up window shows "X" and "Y" values. If you are using the Cursor 2 (blue) then you need to read the x2 and y2 values. At what frequency is the -3 dB point, i.e, where the frequency the magnitude is 3 dB *less than* the low frequency gain? Record your result.
- Select the lower trace (phase response curve) then **Cursor > Show Cursors**. Now move a cursor along the lower trace and note which box shows the data for the lower cursor. Find the phase shift at the same frequency as the -3dB point on the upper trace. What is the phase shift in degrees at this frequency? What does circuit theory tell you?

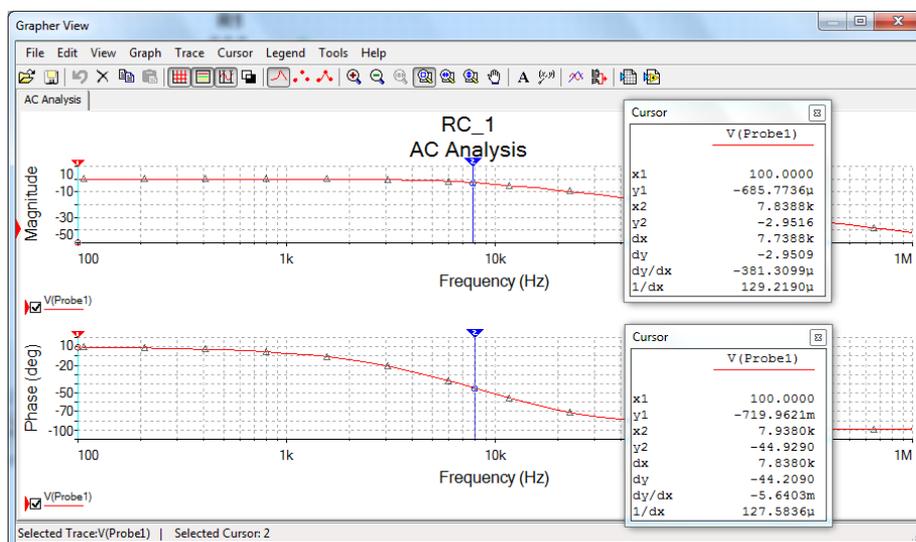


Figure 11: Using the cursors

Before you continue ensure that you have recorded the following results from the Bode Plot:

- The Low Frequency (LF) gain on dB.
- The -3 dB corner frequency.
- The phase response at the -3 dB point.

Note that the "Grapher" view has a "Copy Graph" button that allows you to insert a graph into a Word document.

2.4 Transient Analysis

Now let's continue the investigation of the RC network with an analysis of its transient, or time domain, behaviour. The transient analysis produces a waveform like you would see on an oscilloscope. You can run a transient analysis with a sine wave source or any other waveform but in this case, we shall use a pulse waveform. This show the exponential rise and fall of the output waveform. Use **File > Save As** to save your **RC_1.ms12** file with a new name such as **RC_2.ms12** Then make changes to **RC_2.ms12** file and keep the original **RC_1.ms12** file unchanged.

1. First we need to change the type of source and add a second probe.

In the *Design* window, keep the same values for R1 and C1 (1kΩ and 20nF) but delete the sine wave source. Replace it with a pulse source using **Place Source** (use the "earth symbol" button on the toolbar, not **Place** from the top menu) Then

Place Source >
SIGNAL_VOLTAGE_SOURCES >
PULSE_VOLTAGE > OK

Place the pulse voltage source then close the "Select a component" window.

2. Double-click on the pulse voltage source and set the values as shown in Figure 12.

Initial value: 0 V
Pulsed value: 1 V
Delay time: 10 us
Pulse width: 0.1 ms
Period: 1 ms

Check that rise time, fall time are the default value of **1 ns** (note ns for nanosecond not ms for millisecond)

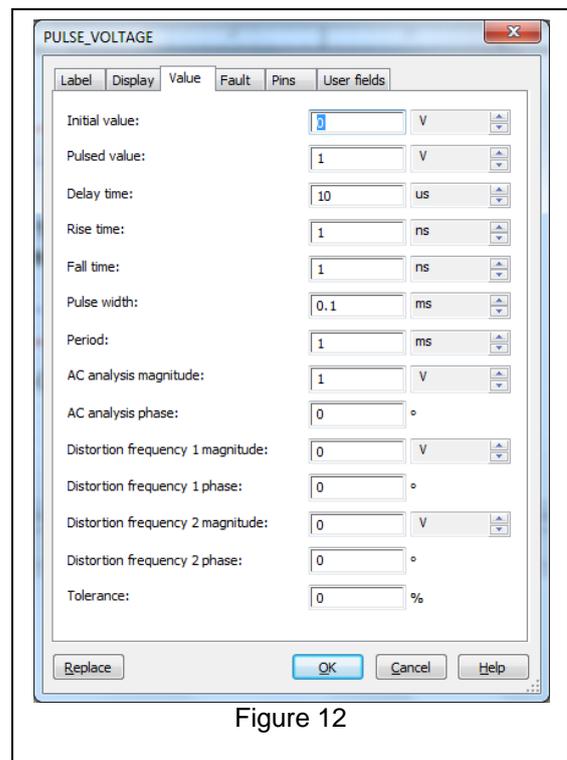


Figure 12

3. Connect the wires to the pulse voltage source and add another AC voltage probe (Probe2) to the point where the voltage source joins R1. Use **More Buttons > Measurement Probe > AC voltage**

Your schematic should now look like Figure 13 below.

4. You now need to set up the Transient Analysis, using:

Simulate > Analyses > Transient Analysis.

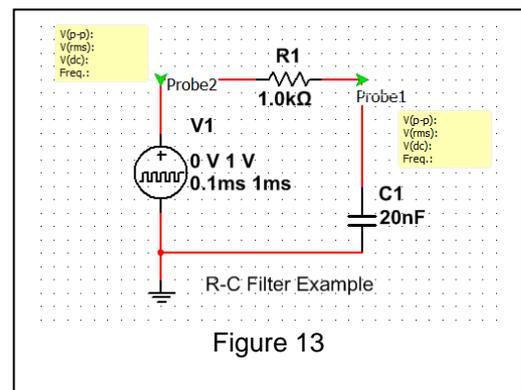


Figure 13

- Set the Transient Analysis Parameters as shown in Figure 14(a) below, i.e.:
Start Time (TSTART): 0 s
Stop time (TSTOP): 0.00025 s
 Tick the box for

Maximum time step settings: (TMax)
 then set **Minimum number of time points: 100**
 Note that the stop time is 250 μs but you need to enter it in seconds as **0.00025 s**.

Alternatively, you can enter the time in scientific notation, e.g. 250e-6 for 250 μs or 1e-3 for 1 ms.

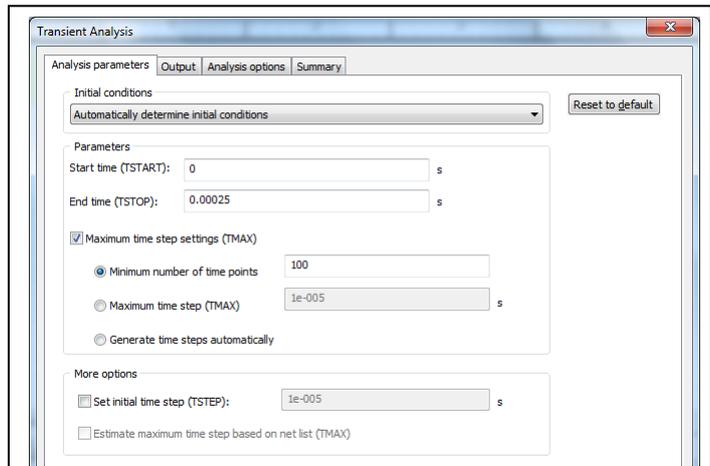


Figure 14(a) Transient Analysis Start/Stop time

- Set the Transient Analysis Output as shown in Figure 14(b) below, i.e.:
Selected variables for analysis V(Probe1), V(Probe2)

- Click **Simulate**. The analysis runs and a "Grapher View" window opens automatically to show a transient response, see figure 15. Use the appropriate buttons to change the black background to white and to turn on grid lines.

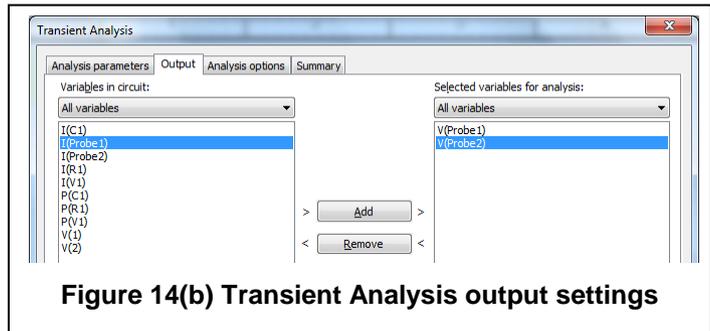


Figure 14(b) Transient Analysis output settings

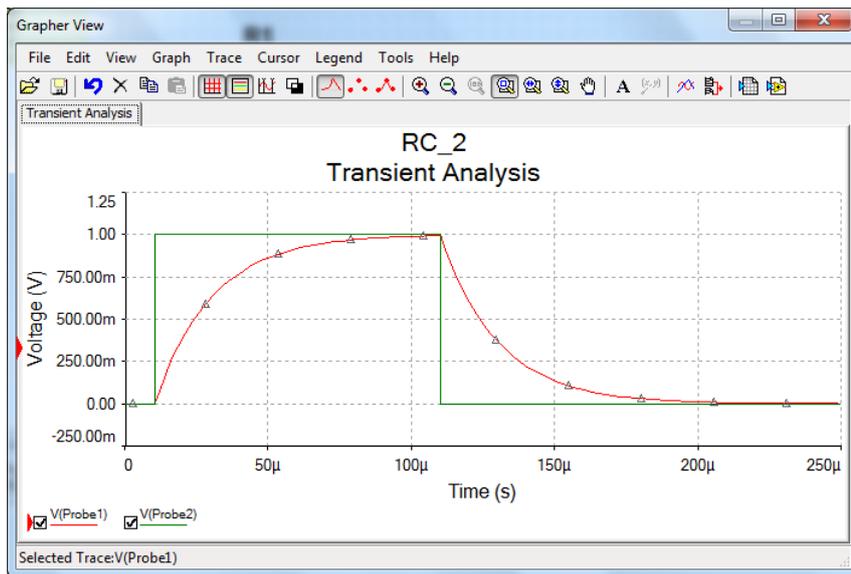


Figure 15 Pulse Response.

- What does the resulting transient analysis, figure 15, tell you about the behaviour of the C-R circuit when driven by a pulse waveform?
 What is the significance of the 100 μs pulse width chosen, relative to the C_1R_1 time constant?

2.5 Further investigations

Modify your initial circuit schematic to create other simple component combinations, for example:

- a high-pass filter (interchange the positions of R1 and C1)
- a band-pass filter (cascade a low pass and a high pass filter).