

# Electronic CAD Practical work

## Weeks 1 and 2

Dr. Martin John Burbidge  
Lancashire UK  
Tel: +44 (0)1524 825064  
Email: [martin@mjb-rfelectronics-synthesis.com](mailto:martin@mjb-rfelectronics-synthesis.com)  
© Martin Burbidge 2006

## Introduction to and familiarization with circuit simulation techniques.

### ***Primary Objectives:***

- To familiarize the user with the SPICE (Simulation Programme with Integrated Circuit Emphasis) simulation package.
- To introduce *net list* and *schematic capture* methods of describing a circuit using simple passive circuit examples.
- To briefly introduce various types of analysis and stimuli.
- To explain why it is sometimes important to use net lists as opposed to schematic capture.

### ***Aims:***

By the end of the first two weeks students should be able to:

- set up a circuit simulation.
- draw and carry out simulations of various circuits presented during the remainder of the course.

## Exercises

**Note:** In the following exercises you are expected initially to enter the circuit description along with its associated stimuli and analysis statements as a textual description, that is in *net list* format. To do this proceed as follows:

Select -> **File** -> **New** -> **Netlist**

Then place the textual circuit description between the **Circuit** and **.end** lines

### **Exercise 1.0 Operating point analysis of a potential divider**

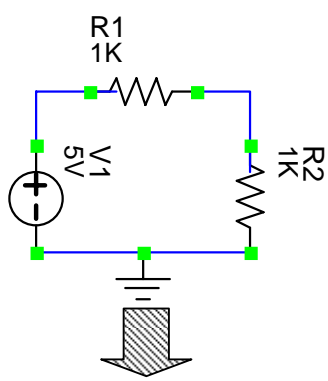
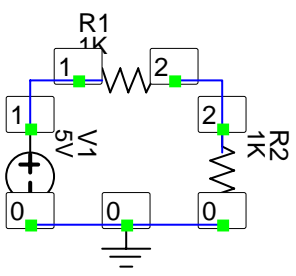
**Note:** SPICE style simulators always perform an operating point, or DC, analysis on a circuit before any other analyses are performed. This analysis calculates the DC

bias points within the circuit and is analogous to a real circuit settling at its DC operating point when the power is applied. The operating points will only be printed if no other analyses are specified or a **.OP** or **.AC** statement are included in the circuit file.

### 1.1 Circuit and net list entry:

The circuit shown in the frame below is a simple potential divider. Before entering the circuit description and performing a simulation do the necessary hand calculations to evaluate the node voltages and the current through the resistors in the circuit.

**Note:** Before attempting any simulation it is always good practice to try and do some simple hand calculations to evaluate the circuit. In the case of this example the circuit is simple and the user would immediately know if the simulator was misbehaving. However, in complex integrated circuits the user may not know how the circuit will operate. In such a case, initial first order approximations are used to gain a “feel” for the expected circuit operation, then the calculations are compared with the more accurate simulation results.

Simple Potential divider schematic	Corresponding net list
 <p data-bbox="289 1407 682 1440"><u>Schematic with node numbering</u></p> 	<pre data-bbox="836 1039 1380 1648"> **** main circuit * 5Vdc source across nodes 1 and 0 V1 1 0 DC 5V * 1K resistor across nodes 1 and 2 R1 1 2 1K * 1K resistor across nodes 2 and 0 R2 2 0 1K **** End of main circuit *Statement to instruct the simulator to *perform an operating point analyses. .OP *.PROBE ;Required by other simulators *.END ;Must always appear *(included automatically in *this case) </pre>

### **1.1.1 Net list entry:**

**Note:** It is not necessary to type in the lines beginning with an \*. These are comment lines and are included to improve readability. However, if you ever have to produce complex net lists, it is always a good idea to include some comments to aid your memory of the circuit you have analysed.

Enter the above net list (in between the circuit and .END line), investigating how the nodes and components of the schematic description map onto the net list description.

Run the simulator engine from the toolbar:

#### **Simulation -> Run**

A window should appear if the simulation is successful.  
Add the calculated values to the window by:

#### **Edit -> Edit Plots**

A dialogue box should then appear with the available currents or voltages in the left hand pane. Click on the right arrow button to transfer the values to the right hand pane. Then click on OK. The DC operating voltages and currents should be displayed.

**Note:** The current through the voltage source is stated as  $-2.5\text{mA}$ . This is due to the current being measured from node 1 to node zero.

After you have run the simulation successfully try changing some of the statements. A suggestion would be to try changing the voltage source statement as follows:

**V1 1 0 DC 5V >>> T1 1 0 DC 5V**

and then re-simulate. View the messages in the log window.  
Try to insert other single errors and view the results.

#### **Notes on net list conventions:**

- **Node 0;** this is always the reference node for the circuit (i.e. ground).
- When declaring a voltage or current source or any two terminal devices with its connections the positive node of the device is given first.
- For other notes on syntax and component declaration refer to the additional handout and reference texts.
- With a complex net list it is good idea to include comments. This is done by including a "\*" at the start of the line. Some simulators also accept a ";" at the end of a statement (for in line comments).

- **For a complete net list additional statements may be required by the simulator to control operations such as printing and plotting. When using this particular simulator the statements are included automatically. If using a different simulator refer to the appropriate manuals and texts.**

### **1.1.2 Schematic capture:**

Initially make sure all the windows within the main window are closed.

Then select:

**File -> New -> Mixed signal (Xspice circuit)**

A schematic window should appear.

Place the components in the window as they appear in the above schematic. This is done as follows: -

***On the toolbar select:***

**Devices -> Then the appropriate parts as below:**

*Voltage source*  
*Resistor simple*  
*Resistor simple*  
*Ground*

**Note:** To rotate the component when it is placed, select the component, *right click* on it, and then select the appropriate option.

After the components are placed wire them together using the wiring tool

***Fifth button from the left below the tool bar.***

**Note:** After each wire has been placed, *right click* to end the individual wire. When all wires are complete return to the selection mode by clicking the black arrow next to the wiring tool button.

After the wiring is complete set the component parameters by double clicking on them. Set the parameters as follows:

**Voltage source ->** Select the radio button **Constant (use DC value)** and set the DC value above it to **5**.

**Resistors ->** Set the Resistance value to **1k**.

### **Simulation**

To run the operating point simulation proceed as follows:

**From the toolbar select:**

**Simulation -> Setup Simulations ->** In the dialogue box click the **.OP checkbox**. Then select the **Run Now** button.

The circuit should simulate as in the previous example.

If it does not simulate view any messages in the log window and check the circuit before re-simulating.

**Save the circuit for your reference as: SchemWeek\_1\_1**

## **1.2 Creating a Net list from the schematic:**

To create net lists from a schematic perform the following steps.

**From the tool bar select:**

**File -> Create net list document.**

A text window should appear with a SPICE net list corresponding to the circuit you have just entered. Study the net list and compare it with the one you entered manually, it should be very similar.

**Save the circuit for your reference as: NetWeek\_1\_1**

### **Notes on net list generation:**

- The generated net lists include the line .OPTIONS with various parameters. The parameters specify the convergence parameters for the simulator. These modifiers control how well the simulator can produce a solution at each calculation point. For the course we will try to keep the default values. However, convergence problems are something you should be aware of if you have to carry out any complex simulations.
- The options statement can include other parameters not related to convergence.
- **Why bother with net lists when you have got schematic capture:**
  - A SPICE style net list is the common form of interface between a schematic capture package and the simulation engine.
  - If you have produced a circuit net list for one circuit, you should be able to (with perhaps some modification) simulate it with another simulator.
  - If you have a description at the net list level it is possible to write programs (in C, Perl, etc.) to modify the net list. This can be useful for more advanced forms of analyses and simulations, such as fault injection etc.
  - It is always useful to be able to understand a net list. Sometimes the schematic capture does not work as expected and produces “bad” net lists.

## **1.3 Extra Work:**

Try to familiarize yourselves with the simulation and capture oriented options available in the simulation package. Before next week try to carry out two tutorials that are included in the package.

These are found as follows:

**Select help -> Help contents -> Tutorials.**

And try to work through:

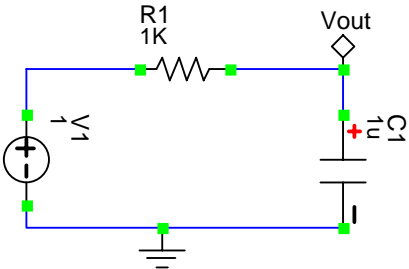
Analog Tutorial #1 and Analog Tutorial #2

## Week 2

### 2.0 Analysis of a simple RC passive network:

For this exercise you are expected to enter the circuit using skills and techniques obtained in the last exercise

#### 2.1 Circuit and net list description:

Circuit schematic	Netlist
	<pre>***** main circuit *Input source * <b>Change</b> *<b>accordingly (see text)</b> V1 1 0 DC 1 R1 1 Vout 1K C1 Vout 0 1u  *Analysis type *<b>Change</b> *<b>accordingly (see text)</b> .dc V1 0 5 .1  .end</pre>

### Circuit equations for the simple low-pass RC network:

The simple equations for modeling the RC network in the time and frequency domain are given below.

Time domain (Transient and steady state):

$$v_{OUT} = v_{IN} (1 - e^{-\frac{t}{RC}})$$

where the  $RC$  product is the time constant of the circuit.

**Note:**  $V_{out}$  reaches approx. 99% of  $V_{in}$  in 5 time constants.

Frequency domain:

*Transfer function:*

*Phase difference:*

$$\frac{v_{out}}{v_{in}} = \frac{1}{1 + j\omega RC} = \frac{1}{1 + j\frac{\omega}{\omega_c}}$$

$$\phi(v_{out}) = -\tan^{-1}(\omega RC)$$

Where  $\omega_c$  is the cutoff frequency (see below)

*The 3dB point, breakpoint or cutoff frequency:*

$$f_{3dB} = \frac{1}{2\pi RC}$$

### **2.1.1 Net list entry:**

Enter the net list shown above. (Note it is not necessary to include the comment lines)

#### **2.1.1.1 DC sweep analysis:**

Make a note of the:

Independent voltage source line: **V1 1 0 DC 1**

which specifies a DC voltage source (called V1) across nodes 1 and 0 with a value of 1 volt,

and the analysis directive line: **.DC V1 0 5 0.1**

which informs the simulator to evaluate circuit operation using a DC sweep. The sweep is carried out on the voltage source V1 which starts from 0 volts and ends at 5 volts. The resolution of the ramp is 0.1 volts (i.e. the input V1 is swept from 0 to 5 volts in 0.1 volt increments).

Run the simulation and observe the output.

As the capacitor is open circuit to DC the output voltage is the same as the input voltage.

#### **2.1.1.2 Step Response of low pass RC network:**

Change the independent voltage source line to:

**V1 1 0 PWL (0 0 1m 0 1.01m 1 10m 1)**

**Note:** This line specifies an input source changing from 0 to 1 volt at 1ms and holding this value for 10ms. The time for the 0 to 1V transition is 10us.

**The stimulus is called a Piecewise Linear Source (PWL)**

See the additional handouts for a more detailed description.

Change the analysis directive line to:

**.TRAN 10u 10m 0 1u uic**

**Note:** this directive informs the simulator to perform a simulation from 0 to 10ms  
See the additional handout for a more detailed description of the transient analysis statement.

Before running the simulations calculate how long it should take for  $v_{OUT}$  to reach 99% of  $v_{IN}$  after  $v_{IN}$  is stepped from 0 to 1 volt.

Run the simulation and add V1 and  $v_{OUT}$  to the Plots window as follows:

**Edit -> Edit Plots -> Select Plots (LH panel) -> Right Arrow -> OK**

The graphs of the input and the output voltage should now be displayed in the same plot window.

Check that the calculations carried out above match with the graphical output of the simulator.

### **2.1.1.3 Sinusoidal Response of low Pass RC Network:**

Change the independent voltage source line to:

**V1 1 0 SIN(0 1 20 0 0)**

**Note:** This line specifies a sinusoidal input source of 1 volt amplitude with a frequency repetition of 20Hz .  
See the additional handouts for a more detailed description.

Change the analysis directive line to:

**.TRAN 10u 50m 0 1u uic**

**Note:** this directive informs the simulator to perform a simulation from 0 to 50ms  
See the additional handout for a more detailed description of the transient analysis statement.  
Note : The 50ms duration is to allow one cycle of the input and output waveforms to be observed.

Run the simulation and view the graphs of the input and the output on the same plot.

Investigate the output graphs.

Note any phase and magnitude differences between  $v_{OUT}$  and  $v_{IN}$ .



Now calculate the 3dB frequency of the RC network and the phase lag that  $v_{OUT}$  should have at this frequency.

Set up the sinusoidal input to have this frequency.

Note for these purposes you only have to be accurate to within 1Hz

Now change the transient analysis statement so that you can view the output waveforms over 1 cycle

Run the simulation and view the output waveforms.

Note any phase and magnitude differences between  $v_{OUT}$  and  $v_{IN}$  and check if they match up with your calculations.

#### **2.1.1.4 AC (Frequency analysis) of Simple Low Pass RC Network:**

Change the independent voltage source line to:

**V1 1 0 AC 1**

**Note:** This specifies an AC voltage source of 1 volt magnitude connected between nodes 1 and 0.

Change the analysis directive line to:

**.AC dec 20 100 500**

**Note:** This directive informs the simulator to run a frequency sweep analysis. The AC source is swept from 100Hz to 500Hz with 20 points being calculated per decade increase in frequency.  
See the additional handouts for more information on the .AC statement.

#### **Magnitude verses frequency plot:**

Run the simulation and add the magnitude plot of V(vout) as follows:

**Edit -> Edit Plots ->** Make sure V(vout) is selected in the left hand panel -> **Click the "MAG" button ->** VM(vout) appears in the right panel -> **Click OK.**

The plot of  $v_{OUT}$  magnitude against frequency should appear in the plot window.

Observe the plot and check it against your calculations in the previous section.

#### **Phase verses frequency plot:**

Remove the previous trace as follows:

**Edit -> Edit Plots ->** Select VM(vout) in the right panel -> **Click on the Left arrow button .**

Now add the phase trace as follows:

Make sure  $V(v_{out})$  is selected in the left hand panel -> **Click on (PH\_deg) -> Click on OK.**

The plot of  $v_{OUT}$  phase against frequency should appear in the plot window.

Observe the plot and check it against your calculations in the previous section.

### **2.1.2 Schematic Entry:**

Open a new schematic window and enter the circuit from the previous section.

Set up the values for the passive components.

**Note:** The diamond shaped symbol ( $V_{out}$ ) in the circuit diagram is a marker. This can be found in:

**Devices -> Marker.**

Set the parameters as follows:

**Double click on the symbol-> Change the part title to  $V_{out}$ -> Select the Checkbox Voltage probe -> Click on OK.**

#### **2.1.2.1 Step response of low pass RC network**

**Double click on the voltage source**

**Select the radio button Piecewise Linear** (Ensure all other options are de selected and all fields are clear)

**Click the Transient properties tab**

**Click the EDIT TABLE button**

**Enter the following values:**

0 0

1m 0

1.01m 1

10m 1

**Then click on OK**

To setup the analysis the analysis:

From the toolbar:

**Select simulation -> Set up simulations ->** Ensure the Transient and Fourier analysis is enabled -> **Click on the Transient and Fourier analyses button**

A dialogue box should appear.

Change the values in the text fields as follows:

**Results delay start time = 0**

**Stop time = 10m**

**Linearisation step = 10u**

**Step ceiling = 1u**

After entering these values **Click on OK**

To run the simulation

From the toolbar:

**Select simulation -> Run simulation**

The same results should appear as in the previous exercise.

Produce a net list for the circuit:

**File -> Create Net list Document**

A window should appear with a similar net list to that which was produced manually in the previous exercise.

### **2.1.2.2      Sinusoidal response of a low pass RC network:**

**Double click on the Voltage source symbol.**

**Select the Sinusoidal Radio Button.**

**Click on Transient properties.**

**Change the frequency setting to 20**

**Note:** To change the frequency-setting highlight the Freq parameter and enter the value 20 in the value box above.

Set up a transient analysis with the following parameters.

**Stop time = 50m**

Leave all other parameters the same.

Now try to set up a transient analysis for a sine wave stimulus at the 3dB frequency.

### **2.1.2.3      AC (Frequency response):**

**Double click on the Voltage source symbol.**

**Click on the use AC checkbox**

**Change the magnitude to 1V**

**Note:** All the other parameters can be left as they are.

Set up the analysis as follows:

From the toolbar:

**Select -> Simulations -> Set up simulations -> Select the checkbox .AC (AC frequency sweep) make sure all other check boxes are disabled -> Click the button .AC (AC frequency sweep) -> Set up the parameters as below:**

<b>Start Value</b>	<b>= 10</b>
<b>Stop Value</b>	<b>= 500</b>
<b>Number of steps per interval</b>	<b>= 20</b>
<b>Stepping interval</b>	<b>= Decade</b>

**Click the radio button “Use Magnitude Plots”.**

Now run the simulation and view the magnitude and phase plots of  $v_{OUT}$  separately. The results should correspond to the exercise you carried out for net list entry.

**Create a net list document and investigate it.**

**Note:** You will notice on the voltage source line (V1) that various parameters for the source exist. The parameters that are used during simulation depend on the type of analysis being carried out.