INTRODUCTION TO LTSPICE

This section gives a quick overview of one of the free simulation programmes, LTspice. Linear Technology provides the programme for free, with no restrictions on its use. It should be noted that the Graphical User Interface (GUI) does not provide access to the entire set of LTspice functions. Despite this, LTspice provides the whole set of SPICE capabilities. The only thing to keep in mind is that some functions can only be accessed via the 'SPICE directives' option.

The following www location has updated information on where to find the software and how to install it:

http://kom.aau.dk/hmi/Teaching/LTspice/LTspice_Intro.html

A 'SwCAD III' icon appears on the desktop after installing LTspice according to the site instructions. By double-clicking this, LTspice is launched, and the window shown in Figure 1 appears.

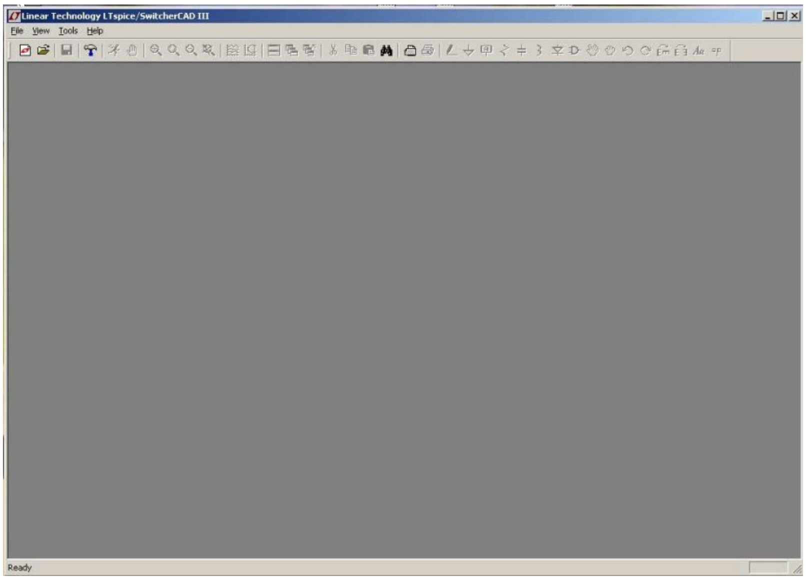

Fig. 1 *LTspice's initialization window*

The main window of LTspice is shown in Figure 1, and it contains practically all of the software's features. The majority of the buttons are initially non-selectable, but this changes when the 'New Schematic' button is pressed. All of the functions that are available through the buttons are also available through the various menus.

To begin a schematic design, all of the various components must be added. To insert a capacitor, for example, go to 'Edit -> Capacitor' and drag the component to any location in the schematic window. In this manner, all of the various components can be arranged to make the desired schemantic. The connections are established and the final circuit is constructed using 'Edit -> Draw Wire.' It's always a good idea to label the different connections and nodes that are of special importance after the circuit is complete. 'Edit -> Label Net' is the command to use. Because the labels serve as node identifiers, it's much easier to plot the right voltage and currents once the nodes have been labelled. If no labels are used, all signals are simply numbered, making it difficult to determine which node has which number. It's vital to remember that for the simulator to work, a ground node must always be present in any schematic.

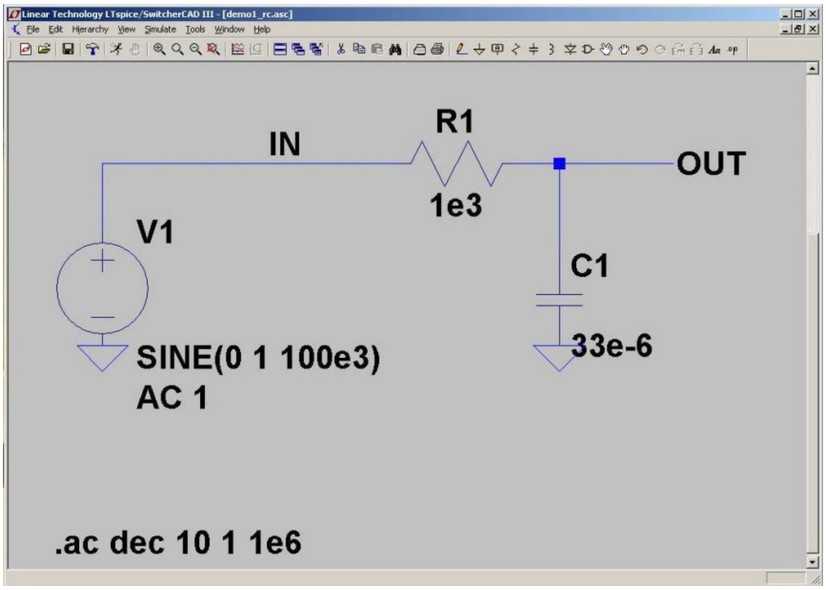

Fig. 2 *A simple LTspice schematic (demo1 rc.asc).*

The LTspice installation comes with a variety of components and models. The user only needs to use the 'Edit -> Component' menu option to gain access. This opens a new window in which you can access all of the libraries. Choose the 'voltage' component and position it in the schematic to add the voltage source shown in Figure 2. To edit the properties of a component, right-click on it to open the properties form. Set parameters using these forms, as shown in Figure 2. It's worth noting that the component properties form for the 'voltage' component contains several places where parameters can be entered. Different locations must be provided depending on the type of simulation the user wants to perform. Only the 'Small signal AC analysis' (.AC) field needs to be filled out for AC simulations. The settings under 'Functions' must be given to enable a temporary analysis.

Furthermore, when entering data in the various simulation tools, users should always take caution when utilising SI units. 'M' is frequently misinterpreted as equal to 'm,' resulting in a resistor value in milli- instead of Mega-.

AC Analysis

After you've established all of the settings, you'll need to perform a simulation on the circuit. There are a variety of specialised simulation types accessible in LTspice, and any other simulation programme for that matter. If no specific analysis is provided, the user will be prompted once the simulation has begun. As a result, the initial step is to use 'Simulate -> Run' to start the simulation. This opens a new window where you may choose amongst the several simulations. A simple ac simulation is utilised in this example. Set the sweep type to 'Decade,' the number of points each decade to 10, the start frequency to 1Hz, and the stop frequency to 100MHz in the 'AC analysis' window. The simulation continues with the simulation data entered after pressing 'OK.' After that, the user is asked to choose which waveforms should be plotted. Currents are reported as the current through a certain component, whereas voltages are supplied as the voltage in a single spot.

It's vital to note that practically all simulation programmes use a syntax in which positive currents flow INTO a component. Because a (positive) current flowing into a resistor must eventually exit the resistance, the output current must be negative, which causes confusion. Many simulation software have a way of indicating which component terminal is the positive one to avoid this. This is not the case in LTspice, making determining the direction of a current extremely difficult. This has no effect on the accuracy of LTspice; the phase may change by 180 degrees depending on the direction of the component when a current curve is drawn.

It's simple to see which waveform corresponds to the output thanks to the usage of labels, and to plot it, select 'V(OUT)' and then press 'OK.' This opens a new window, as shown in Figure 3, in which the output signal's amplitude and phase are displayed.

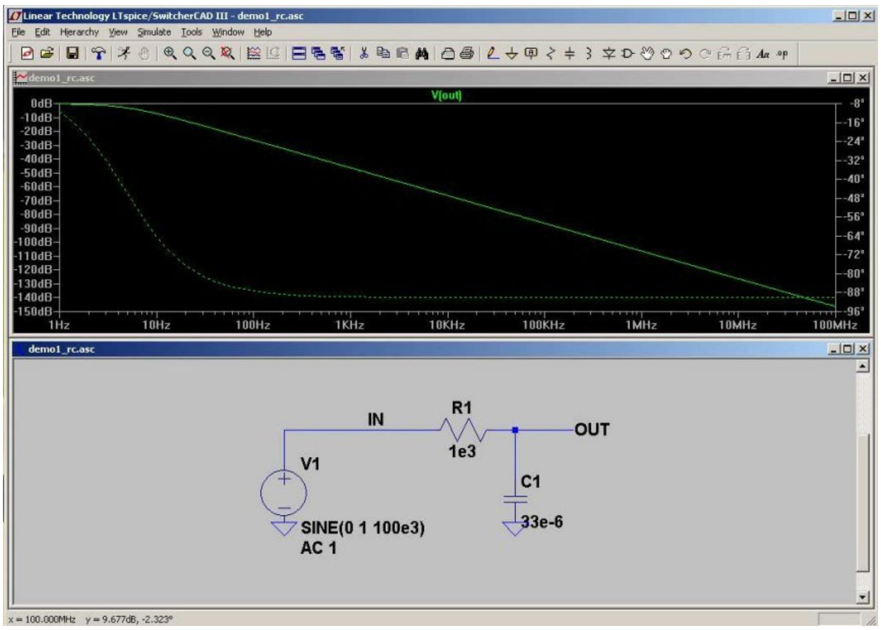

Fig. 3 *In LTspice, the result of a simple AC analysis (demo1 rc.asc).*

The output of an AC simulation, as shown in Figure 3, is a frequency response plot that looks a lot like the well-known Bode plot. The circuit implements a lowpass function, as can be seen by glancing at the amplitude plot. Waveforms can be added and removed at will, and because LTspice saves all node voltages and currents by default, adding a curve does not necessitate a re-simulation. When a user single-clicks on a node or wire, the curve is added to the existing curves; however, if the user double-clicks, the plot window is reset, and just the curve that was just selected is plotted.

Double-click on the axis to the right of the graph to prevent the phase from being plotted. This allows the user to provide axis information, and it is possible to only see the magnitude plot by ticking the 'Don't plot phase' field.

Double-click the 'V(out)' text to find the filter's 3dB corner frequency. This creates a marker window, which may be used to get an accurate reading of the performance by dragging the marker. At a frequency of 4.82Hz, the magnitude curve equals -3dB in this case. It is, of course, possible to scale the plot using the zoom feature.

A simple small-signal analysis is an AC simulation. This essentially means that the simulator creates a linear model for all non-linear components before analysing them. Consider the scenario of a 12V amplifier to better see why this is crucial. When a circuit is powered by 12 volts, it can't handle output voltage swings that exceed the supply voltages. If the amplifier's gain is 10, any input signal greater than 1.2V will result in signal clipping at the output. This is impossible to discern from an AC simulation due to the small-signal linearization. A 100V input signal will result in a 1kV output signal in this case. This is plainly not conceivable, and a transitory study is required to uncover such impacts.

Transient Analysis

A transient analysis is required to identify if signal clipping occurs and to evaluate any non-linearities. The transient analysis is a large-signal analysis that can take into account non-linearities. The transient analysis is quite simple to set up, however it makes no sense for the fully passive RC circuit example. Take the RC example from before and replace the resistor with a diode as illustrated in Figure 4 to demonstrate how transient analysis works in practise. It's worth noting that the capacitor has been shrunk in size.

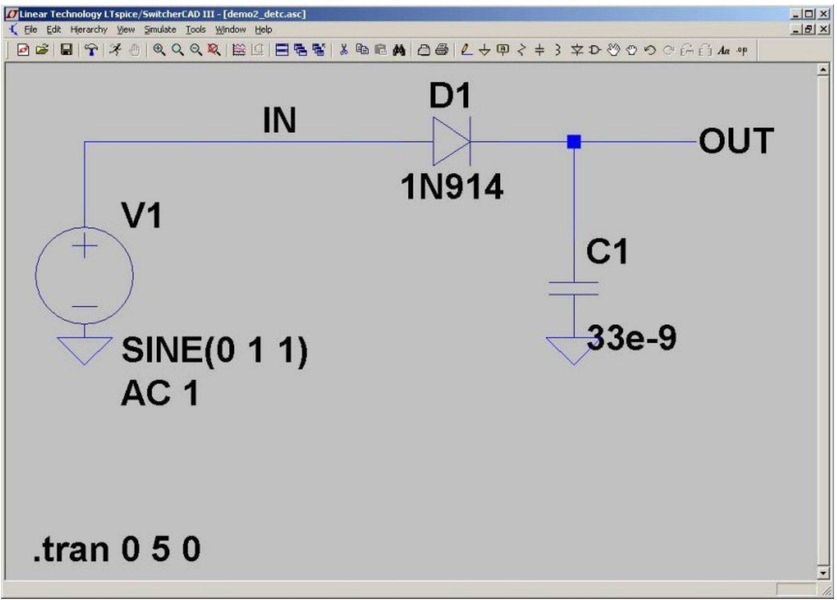

Fig. 4 *A simple LTspice schematic entry (demo2 detc.asc).*

When you right-click on the diode symbol, you may set some basic diode parameters or use the 'Pick New Diode' option to select a specific diode. When you select a new diode, a window appears with all of the diode models available in the LTspice component library. The circuit and the consequent AC simulation result are displayed in Figure 5 after replacing the resistor with a 1N914 diode and lowering the size of the capacitor.

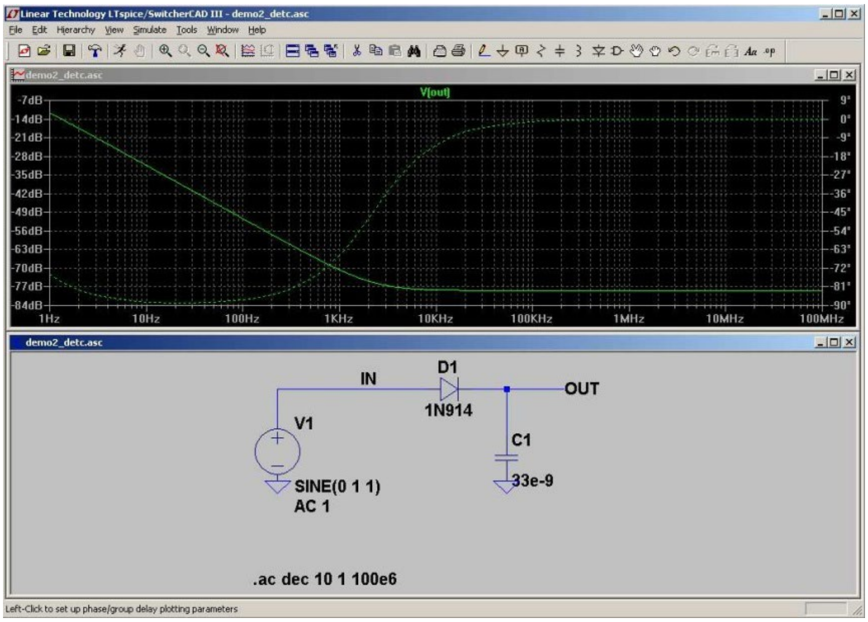

Fig. 5 *LTspice AC analysis result (demo2 detc.asc)*

Obviously, an AC analysis provides no useful information for the new circuit. A transient analysis is necessary to obtain some helpful information. This necessitates the configuration of the voltage source. The source has been set to 1V (peak voltage) and 1Hz frequency in this example. When specifying the transient analysis, keep in mind the number of signal periods that the simulation will span. Five periods have been chosen in this case, which amounts to 5s. The user can change the analysis type by right-clicking on the '.ac' text in the schematic window. As a result, the 5s is added as a transient analysis stop time. The simulation's outcome is depicted in Figure 6, which includes both input and output signals.

The plots show that the circuit has a rectifying impact on the input signal. When comparing Figures 5 and 6, it is evident that the two forms of analysis provide distinct information.

If a specific diode is missing from the component library, it must be added. This is the case if a user wishes to emulate the circuit's performance using, for example, the 1N4148 diode.

Adding Components to the LTspice Library

If a certain diode is required, such as the 1N4148, and it is not included in the component library, the user must add it. LTspice's file structure is fairly simple, making it simple to add any components to its library structure. If you install the software in the default place, you'll find it at:

C:/Program Files/LTC/SwCADIII

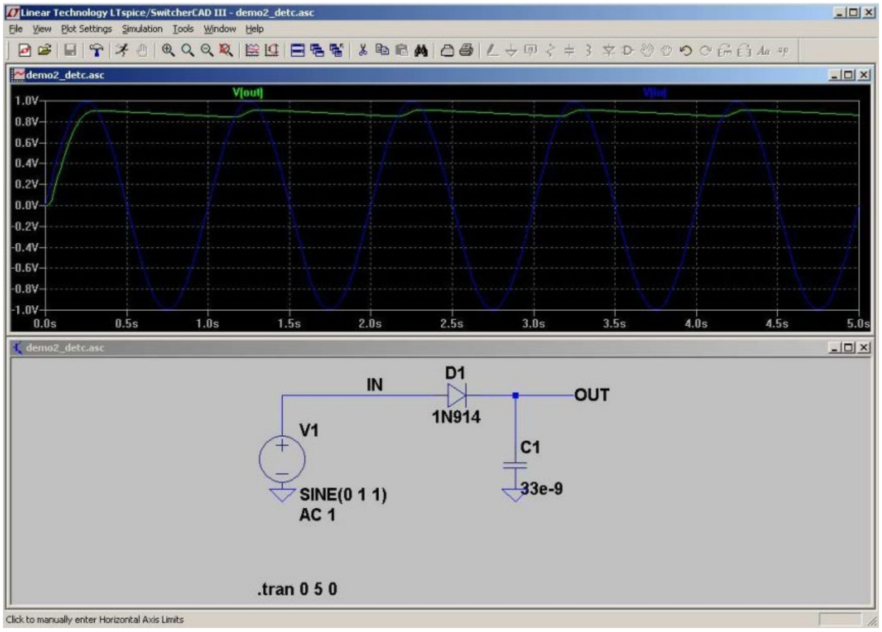

Fig. 6 *Result of a transient analysis in LTspice (demo2_detc.asc)*

The 'lib' sub-folder contains LTspice's entire component library. This library is organised into four folders: 'cmp,"sch,"sub,' and 'sym.' The component library is organised as the names suggest.

- cmp - Basic components and their models
- sch - Larger structures consisting of schematics
- sub - Sub-circuit descriptions to model the structures in the 'sch' folder
- sym - Symbol views of the different components.

The user really uses the symbols in the 'sym' folder when drawing the circuit schematic. Multiple models can be applied to any symbol in LTspice, which is a highly useful tool. This means that a symbolic view does not have to be included for every model in the library. This makes it simple to add new device models to the component library, as well as to change model parameters as necessary.

The initial step is to gather information about the diode in question, as well as a spice model for it. One approach to do this is to conduct an online search using Google or, better yet, to visit the websites of several component manufacturers. A Google search for 1N4148 and diode yields a slew of results, some of which include data sheets and even spice models. It is critical to always validate models downloaded from non-manufacture pages by comparing them to data sheets. It's a straightforward operation to add the model to the library once it's been verified. The diode should be placed in the 'cmp' category because it is a basic component. A lot of files can be found here:

- standard.bjt - BJT transistor models
- standard.cap - Capacitor models
- standard.dio - Diode models
- standard.ind - Inductor models
- standard.jfet - JFET transistor models

- standard.mos - MOSFET transistor models
- standard.res - Resistor models

Because these are all simple ASCII files, all that is required of the user is to open the ‘standard.dio’ file in any editor and cut and paste the 1N4148 spice model into it. When the user tries to select a new diode after this has been done, the new model should appear.

It’s worth noting that LTspice reads all of the information contained in the files in the ‘cmp’ folder during startup. This means that if a new component model is added to one of these seven files, the application must be restarted.