Easy-Spice

A/D Mixed Mode Simulator

Evaluation Guide

Number One Systems

Welcome To Easy-Spice

This is a brief guided tour of the evaluation version of the Easy-Spice A/D mixed mode simulator. The evaluation version is a fully functional version of the program but is limited in its available design capacity. You will find the capacity to try some simple circuits of your own but nothing too elaborate.

There are no time limits on this evaluation and it can be removed at any time using the Windows [Add / Remove Programs] mechanism in the [Control Panel].

Hardware Requirements

As with Easy-PC For Windows, Easy-Spice doesn't require special or 'high powered' hardware to run on. A fast 486 or a Pentium processor machine is suitable provided is has enough spare disk space to load Easy-PC and Easy-Spice (about 50Mb is required), and enough RAM to ensure that the program isn't being paged in and out of memory, so about 32Mb minimum for the evaluation is OK. For the fully functional Easy-PC product you would probably need more than 32Mb of RAM to get the best performance out of the application. The evaluation runs happily on Windows 95/98/ME/NT4.0 and Windows 2000 but cannot run on Windows 3.1x - sorry! A CD-ROM drive and mouse are also required to load and operate it.

Software Requirements

You can run Easy-Spice with Easy-PC For Windows version 4.0 or later. We have supplied you with demonstration versions of both Easy-PC and Easy-Spice. You can however, just load the demo of Easy-Spice to run with your existing ‘purchased’ copy of Easy-PC. The Easy-Spice product will still run in demo mode with its demo limits but it does mean you can save different circuits.

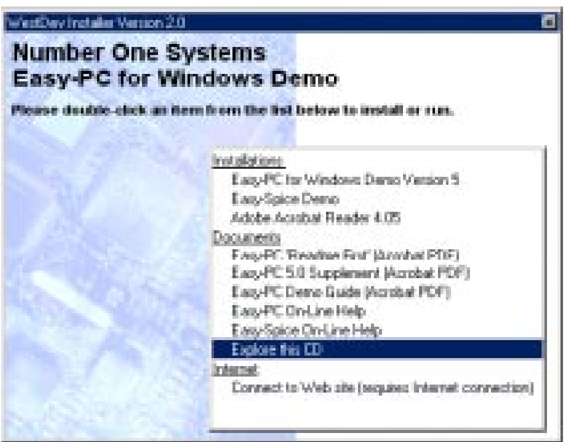
Installation Of The Evaluation Software

System Setup / Installation from CD ROM

You will need to install both the Easy-PC For Windows and Easy-Spice products before the sample designs can be loaded and the simulation run.

To install the evaluation version of Easy-Spice, first ensure that all other applications have been closed, and relevant data saved. Insert the CD-ROM into the CD drive and wait for a moment. The default setup of Windows enables the *autorun* facility to be automatically used. If your system has the autorun facility disabled or it simply hasn't worked, select [Run] from the [Start] menu, and type D:\Easy-Spice\setup.exe (replacing the D: with the actual CD drive letter, if necessary, then <Enter>.

The Easy-Spice Setup screen will now provide you with a number of choices for installation of products and for various utilities. A Setup screen is now displayed, select the 'Install Easy-Spice Demonstration' option and follow the instructions on the preceding screens.



If at any time you click on [Cancel], the window that appears gives you the chance to Continue or Exit the installation.

When the CD has been copied, messages will briefly display indicating that registry entries have been created, and icons copied. The selected folder will appear briefly as the icons are added, then a 'Setup Complete' window appears. You can choose whether to immediately launch the program, but in any case, the final step is to click on [Finish] to complete the installation.

What Should You Expect From A Simulator ?

Simulation should be seen as a complement to the traditional breadboarding phase of circuit design, rather than a complete replacement. With simulation, you can gain an insight into the workings of your circuit that are difficult or impossible to do in "real life". Here are some examples of things you can do with a simulator that can't be done on the bench:

- Make "non-invasive" measurements. For example, current is always difficult to measure without adding components to the circuit. Even magnetic current probes can alter a circuit's behaviour. Measuring current with a simulator, however, is straightforward and doesn't affect its characteristics.
- Remove parasitic elements. For example, you can remove the capacitance that cause "Miller Effect" in an amplifier to see if that is the cause of a misbehaving circuit.
- Operate components outside their safe operating area. Sometimes it is useful to see what will happen when you run - say - a power supply at beyond its ratings. In real life you risk damaging some components in the circuit, but a simulator does not suffer such limitations.

It is often not viable to simulate a complete circuit either because the simulation would take too long or because models are not available for all the components you are using. But, it is nearly always possible to simulate a part of your circuit and learn a great deal about the design by doing so.

You can also use a simulator to try out ideas at a system level. Easy-Spice has a number of components that can be defined by an equation either to describe a non-linear characteristic or to implement an arbitrary frequency response. You can use these blocks to implement functionality without having to think about the underlying implementation.

What's In The Product ?

Easy-Spice has two main elements:

1. The simulator. This is a "number cruncher" that takes a netlist as it's input and generates a - sometimes very large - data file containing the results. The netlist contains a description of your circuit along with commands to instruct the simulator exactly what analysis to perform. The data file contains all the voltages and currents in your circuit for later plotting or analysis.
   The simulator is based on SPICE3 developed by the University of California at Berkeley and XSPICE developed by the Georgia Technical Research Institute. Although based on these programs, Easy-Spice has received extensive further development to improve its overall performance.
2. The waveform viewer. This part of the program plots the results of the simulation using the data file created by the simulator. It can do this while the simulator is running or on demand after the simulation has completed. As well as plotting facilities, the waveform viewer can also perform measurements on the results such as RMS, peak-peak, frequency etc.

To run Easy-Spice you will also require the Easy-PC For Windows Schematics package. This is used to hold the model values, create the circuit and to generate the netlist which is simulated.

Easy-Spice - Evaluation Guide

Getting Started

This evaluation guide is intended to take you on a short journey through the main features of Easy-Spice. It is not intended as a training course but is provided to give you a 'taste' of some of the powerful features in the product. We recommend that you print out this manual before starting to run the software. You will then be able to compare the printed instructions with the information on the screen without having to keep switching tasks all the time.

As this evaluation version does not allow you to save your work as you proceed, we have provided some example files. These can be loaded at any point and the simulation run.

Easy-Spice requires Easy-PC For Windows Schematics in order to generate a netlist which can be simulated. This guide will start with the Schematic design and move quickly onto the simulator. If you wish to know more about Easy-PC For Windows, please read the demo guide for the Easy-PC product which is also supplied.

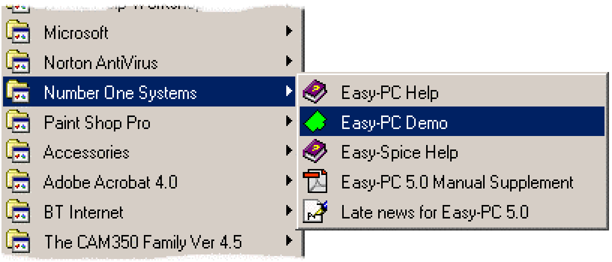
Standard Installation Folder

As we explained at the beginning of this guide, and during the installation, the evaluation version of Easy-Spice will install by default into C:\Program Files\Number One Systems\Easy-Spice. Other folders are created within this, for example, Spice example files under the main Easy-PC folder and libraries, '**Examples\Spice**' and '**Libs**' respectively.

Running The Evaluation Software

Starting Easy-PC

To start this evaluation you first need to run up Easy-PC. Click on the [Start] button on the taskbar (along the bottom of your screen). Select [Programs] [Number One Systems] and [Easy-PC Demo].



The application will now start running, Easy-PC is now ready to use, as is the Easy-Spice simulator (it is assumed that you have installed the Easy-Spice product as well !).

It is assumed that you understand how to run Easy-PC, but where specific actions are required to run the simulator, these will be documented as instructions here.

Load An Example Design

There are sample designs supplied in the Easy-PC folder under Examples, **…\Easy-PC\Examples\Spice** These are sorted in circuits of similar functionality. There is also a **Tutorial** folder (under **…\Examples\Spice)** which contains circuits used in this guide. This is the full set of circuits as supplied with the 'real' product but not all of them will run with the Easy-Spice demonstration mode software, some will be too large in size. All of the circuits supplied in the **Tutorials** folder will run.

A Ready To Run Circuit

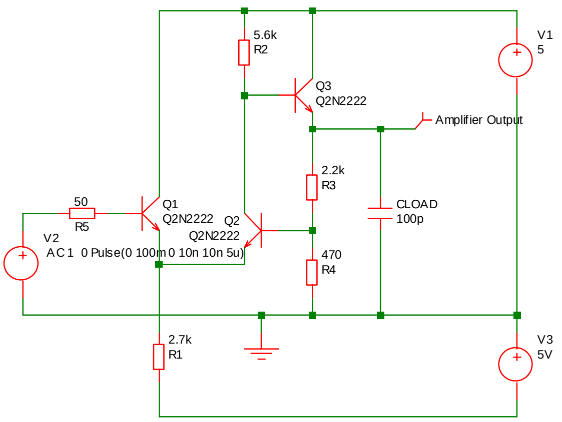
This tutorial demonstrates a basic simulation on a ready to run to circuit. All you need to do is load the circuit and then run it. We will then make a few changes to the circuit and make some extra plots.

Easy-Spice - Evaluation Guide

This tutorial demonstrates the basic features without having to get into the details of setting up a simulation. Proceed as follows:

Select the [File] menu and [Open]. Select the schematic file **Tutorial 1.sch** which you should find in the folder **Examples\Tutorials**.

Select [Open] to open this file. A Schematic window will open with the following circuit:

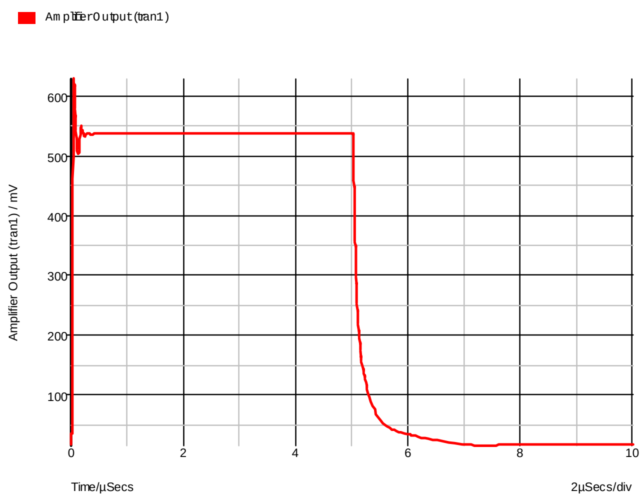


This is a simple feedback amplifier designed to amplify a 100mV pulse up to around 500mV. The basic requirement of the design is that the pulse shape should be preserved, DC precision is important but is not critical. The above is our first attempt at a design but has not yet been optimised.

This example circuit has been setup to be "ready to run".

To start the simulation, select from the Schematic window [Tools] [Easy-Spice Simulation…]. You will be presented with the Simulation dialog. Press [Run] to start the simulation. If not already running, the simulation program will be launched and the circuit's SPICE netlist sent to it for simulation. Simulation will not take long, on most machines less than a second.

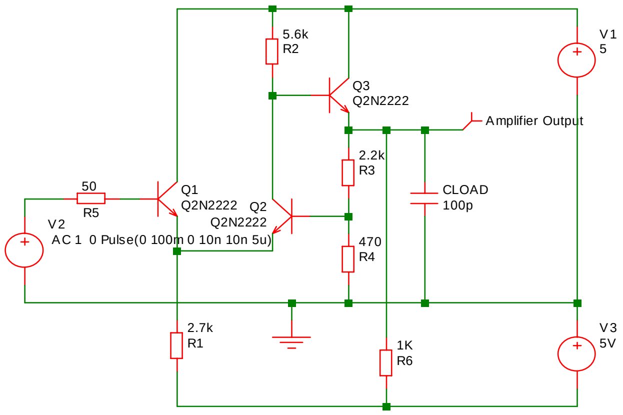
A window will open showing a graph of the output voltage:



As can be seen, our amplifier doesn't work all that well. There are two problems.

1. There is substantial ringing on the rising edge, probably caused by the capacitative load.
2. The falling edge is somewhat sluggish

The sluggish falling edge is caused by the absence of any standing current in the output emitter follower, Q3. To rectify this, we will place a resistor from the emitter to the -5V rail. The resulting schematic is shown below:



To make this modification, proceed as follows:

Select the menu [Add] [Add Component]. The Add Component dialog will appear. Use the Library **SPICE.cml** and choose the component **Resistor (Box Shape).**, or alternatively **Resistor (Z Shape)**. Press [Add] and a resistor symbol will appear. Place this in the location shown in the diagram above. Click the left mouse button to fix it to the schematic. You will now see another resistor symbol appear . Cancel this by pressing the [Esc] key.

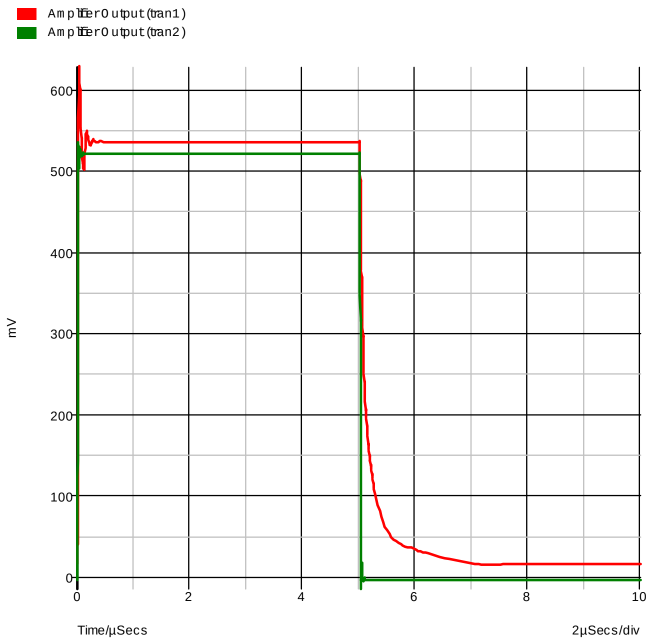
Remove the display of the Component name and Pin names on the new resistor using the [Properties] option.

Now wire up the resistor. There are a number of ways of doing this. Double click the left mouse button over one of the resistor's pins to start the connection. Click again to finish the connection on the required item.

You can also enter connection mode by selecting the toolbar connection button This puts schematic into a permanent wiring mode where the left key is always used for adding connections. Revert to normal mode by pressing the select button .

Re run the simulation using [Tools] [Easy-Spice Simulation…].

The graph will now be updated to:



The green trace is the output after the modification. As you can see, The problem with the trailing edge has been fixed and the ringing is much improved.

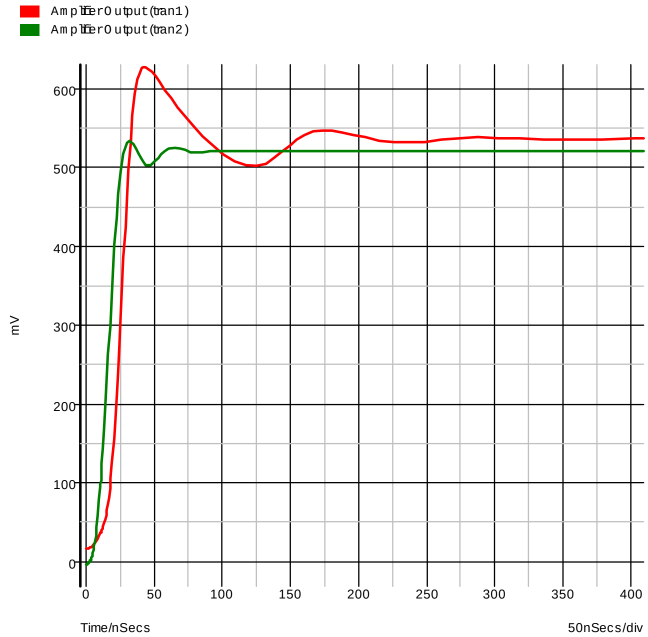
Now let's have a look at the ringing in more detail. To do this, we need to zoom in the graph by adjusting the limits of the axes. There are two ways of doing this. The quickest is to simply drag the mouse over the region of interest. The other method is to manually enter the limits using the "edit axis dialog box". To zoom with the mouse, proceed as follows:

Make sure that the graph window is selected by clicking in its title bar.

Place the cursor at the top left of the region of interest i.e. to the left of the y-axis and above the top of the red curve.

Press the left mouse key and while holding it down, drag the mouse to the bottom right of the area you wish to zoom in. You should see a rectangle appear as you drag the mouse.

Release the mouse key, you will see something like this:

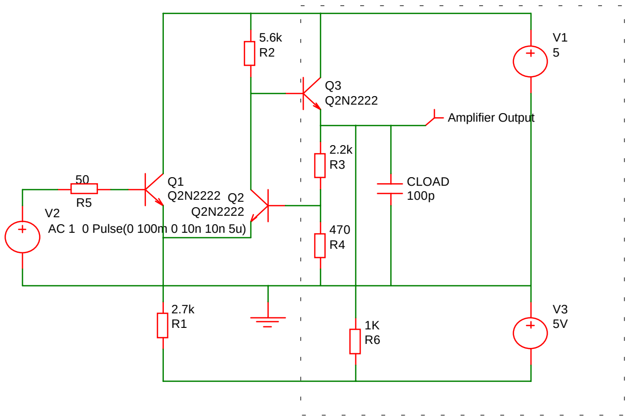


If you don't get it quite right, press the Undo Zoom button:  to return to the previous view.

We can probably improve the ringing by adding a small phase lead in the feed back loop. This can be done by connecting a small capacitor between the emitter of Q3 and the base of Q2. There isn't room to add this tidily at present,, so first, we will move a few components to make some space. Proceed as follows:

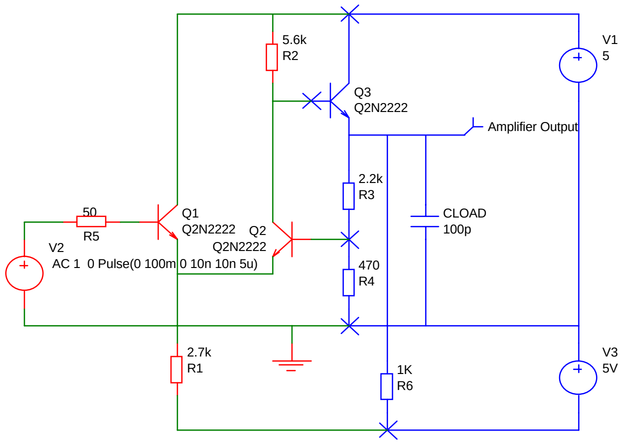
In the Schematic window, drag the mouse with the left key pressed over the region shown by the dotted lines below:

Easy-Spice - Evaluation Guide



As you drag the mouse, a rectangle should appear.

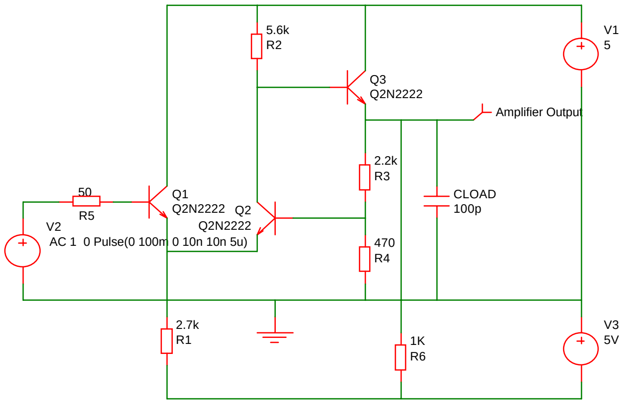
Release the mouse. The area enclosed will turn blue:



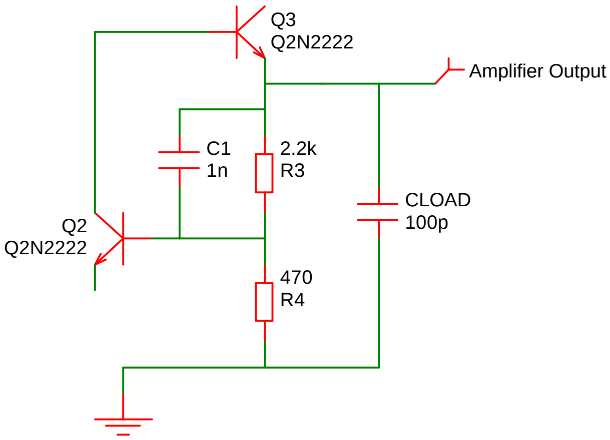
The blue wires and components are said to be "selected". To move them, place the cursor within one of the selected components - V1 say - then press and hold the left mouse key.

Move the mouse to the right by two grid squares then release the left key.

Unselect by left clicking in an empty area of the schematic. This is what you should now have:

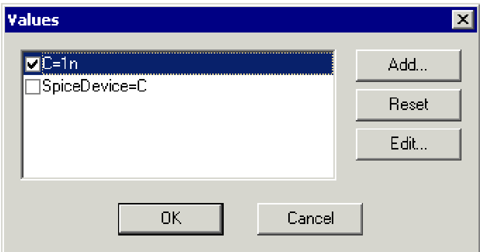


Wire in the capacitor C1 as shown below using a similar procedure as for the resistor R6.



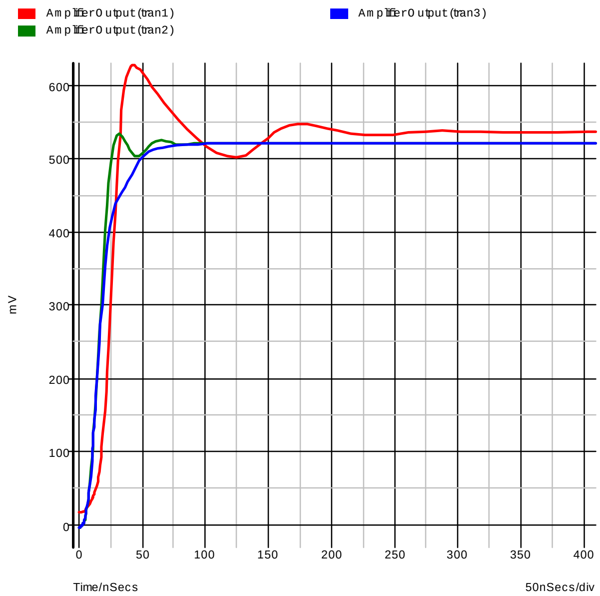
1nF is obviously far too high a value so we will try 2.2pF. To change the component's value proceed as follows:

Right click over C1 and select [Values] from the shortcut menu You should see the following dialog box appear:



Double click on the “**C=1n**” value, or single click on it and press the [Edit] button. Use the [Value] dialog that is then displayed to change it’s value to **2.2p**

Now re-run the simulation, this is the result you should see:



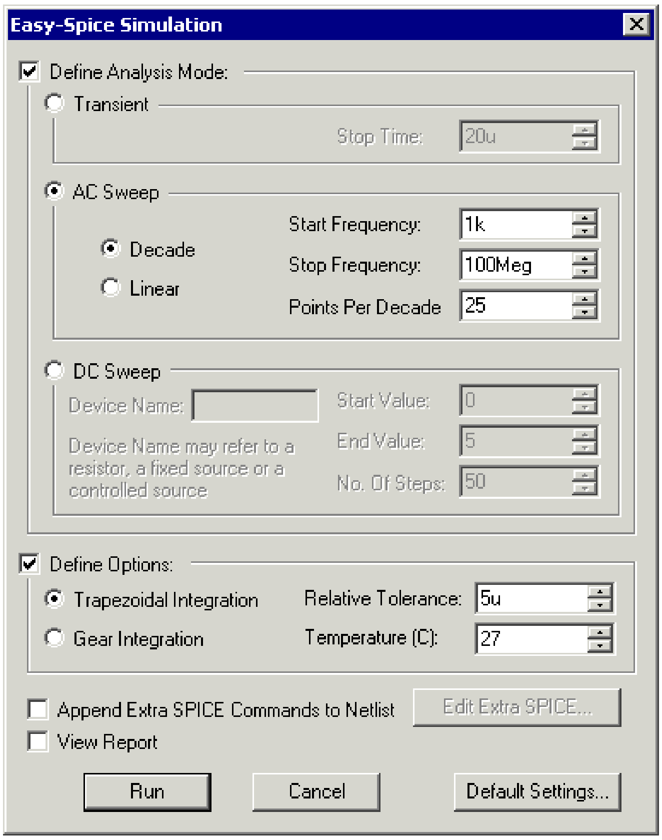
The blue curve is the latest result. This is now a big improvement on our first attempt.

We will now round off tutorial 1 by introducing AC analysis.

AC analysis performs a frequency sweep over a specified frequency range. To set one up, follow these instructions:

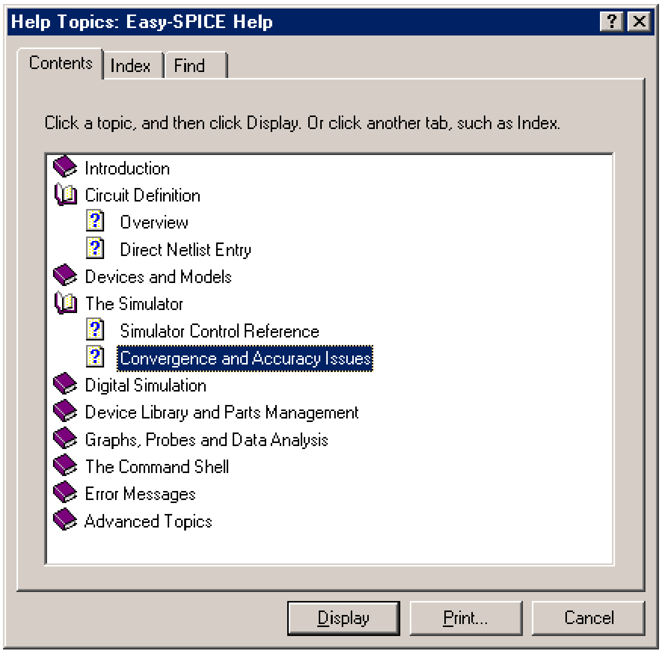
Easy-Spice - Evaluation Guide

In the schematic window, select the menu [Tools] [Easy-Spice Simulation], this is what you will see:



Click the radio button titled **AC Sweep**. The details of the AC sweep should be set up as shown in the dialog above. Click [Run] to start the simulation. A new graph sheet will open within the same window as the existing ones.

Easy-Spice Help



If you need more detailed information on any of the functions offered by Easy-Spice, simply press the <F1> key on the keyboard. You can either select a topic by double clicking it in the [Contents] tab, or search for a topic by typing the first few letters of a keyword in the [Index] tab.

Explore for Yourself

Now that we have taken you through the design process from Schematic to simulation, it is time for you to carry on exploring the facilities of this program on your own. As save is inhibited in Easy-PC, you will not change permanently any of the sample files provided or your own designs. If you have used the Easy-Spice demo with a working copy of Easy-PC, you will be able to explore more and try different iterations of the design, but don't forget there are demo limits which come into play.

Technical Support

For more detailed reference information we recommend that you use the on-line Help system accessible via the [Help] menu on the main menu bar, or by pressing the <F1> key on the keyboard.
If you have any problems installing or running this program, you can contact your dealer or our Technical Support department by telephone on +44 (0)1480 382538, by fax on +44 (0)1684 773 664, or by e-mail to support@numberone.com

How To Buy The Program

Now that you've decided you like the program and want to buy it, here's how to do this.

To obtain one of the fully enabled versions of Easy-PC For Windows and Easy-Spice, complete with a printed manual, full libraries and the ability to generate industry-standard manufacturing files, contact us at our International Office on +44 (0)1684 773 662 or by fax on +44 (0)1684 773 664

We can provide you with a price quote or Proforma invoice if one is required. Check the prices on the Number One Systems web site www.numberone.com, prices are included for upgrades and competitive cross-grades. These prices combined with the latest features make the Easy-PC For Windows product range an affordable professional choice.

Web Site Access

www.numberone.com

Our web site www.numberone.com keeps you fully up-to-date with current developments and new product information. The site is fully accessible and not restricted in any way, please feel free to visit and browse at any time.

Easy-Spice - Evaluation Guide

Evaluation versions of our simulators and other products can be downloaded from our Web site at the address above or obtained direct from our sales offices or dealers.

Contacting Number One Systems

We can be reached at the following address:

Number One Systems
Oak Lane
Bredon
Tewkesbury
Glos GL20 7LR
UK

Tel: +44 (0)1684 773 662
Fax: +44 (0)1684 773 664
Email: info@numberone.com
web site: www.numberone.com