Multi-instrument – discount *and* free dmm

ELECTRONICS WORLD

INCORPORATING WIRELESS WORLD

September 1996 £2.25

Austria Asch. 65.00
Denmark DKr. 66.00
Germany DM 15.00
Greece Dra.1000.00
Holland Dfl. 11.50
Italy L. 8500.00
Malta Lm. 1.45
IR £3.30
Singapore S$12.60
Spain Pts. 800
USA $5.50

A REED BUSINESS PUBLICATION
SOR DISTRIBUTION

Linsley-Hood Class-A power

Gigahertz prescaler

Intelligent opto

Video inserter

Tina CAD – UK launch, review and reader offer

9 770959 833028 09>

Design lab bytes

Tina is a pc-based circuit design lab with virtual instrumentation including scope and spectrum analyser. Until now it has bubbled under as a DOS package, but Clive Ousbey believes that the new Windows version poses a serious threat to the competition.

Recent years have seen a greater use of simulators as an aid to electronic design. This has been due largely to easier to use graphically driven systems. While being powerful in experienced hands, older style netlist entry types of simulator where not user friendly. On the other hand, the early graphics-based programs that *were* easier to use were either expensive 'professional' systems or lacked much power or flexibility.

TINA for Windows, version 4, from DesignSoft, is the latest simulator package attempting to close this gap. An acronym for Toolkit for Interactive Network Analysis, *TINA* is a Hungarian product that has existed in a DOS form for several years. Until now, it has had quite a low profile in the UK. In order to address these problems, the makers have appointed a new distributor to market the significantly enhanced new Windows version.

TINA's graphical interface is similar in style to its main competitor *Electronics Workbench*, but offers a wider variety of output presentation, greater flexibility, optional instrumentation hardware and more extensive analysis options. The latter gives it a functionality more in line with something like PSpice.

Circuit capture facilities

The schematic editor is normally used for working on a circuit for analysis. The alternative is to import a PSpice type netlist: exporting is also a possibility. *TINA* has an editor which is easy to use and operates as you would expect a Windows program to work.

Component symbols are selected from tabbed groups on the tool bar. In some cases, a generic symbol is not appropriate. Clicking the symbol icon instead opens a list of related parts to choose from. The symbol can be rotated or mirrored and its properties modified, either while placing or at any time thereafter. Various parameters relating to the component can be edited – value or tolerance for example – via the properties window.

In addition to the usual components, there are various others to aid building and simulating a circuit. These include voltage sources, jumpers allowing separate parts of the circuit to be connected by a signal name and the ground symbol that must always be present.

Input and output can be slightly confusing due to the variety of ways it can be achieved, depending on the type of analysis required. For measuring basic signal input and output, there are voltage or current generators, various meters and voltage test pins available. A selection of signal types can be applied to the input – including the option of a user-defined waveform.

Virtual function and digital signal generators are available to provide stimulus. For more complex analysis, a

Tina's vital statistics

Editors
- Schematic and netlist editors
- Text and equation editors with interpreter

Presentation
- Bode plots and Nyquist diagrams
- Transient response plots and digital waveforms
- Linear or log scales

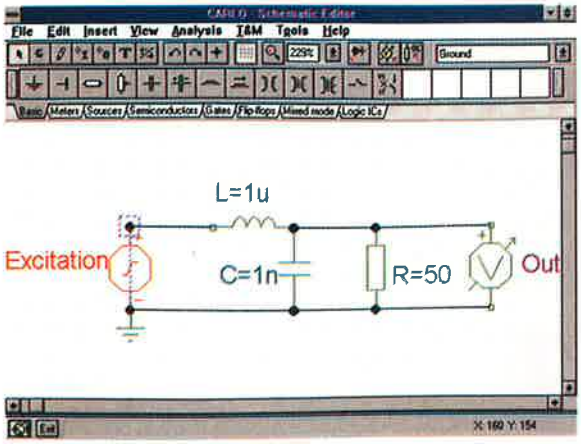
Analysis
- Fourier
- DC, with automatic circuit optimisation
- AC, with complex *V, I, Z* and power analysis
- Transient analysis for analogue *and* mixed mode
- Symbolic analysis gives closed form expressions

Simulation
- Analogue, digital and mixed-mode

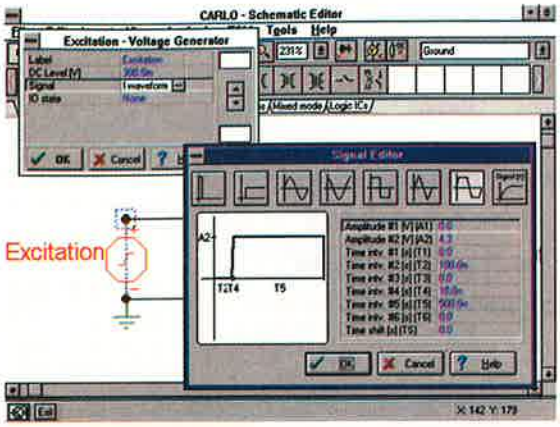
Stimulation
- Standard and user-defined circuit excitation

Virtual instruments
- Signal analyser
- Storage oscilloscope
- Function generator
- Logic analyser
- Digital multimeter
- Digital signal generator

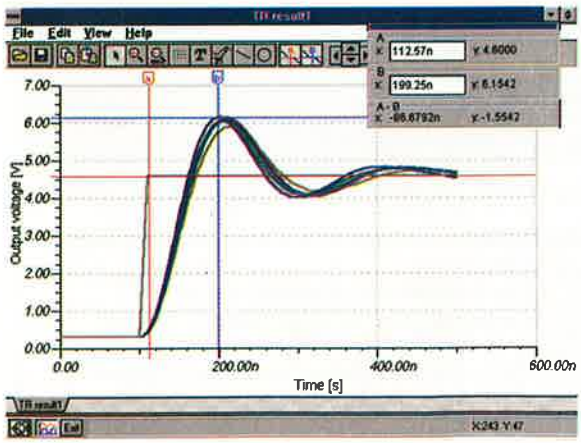
Hardware options
- PC measurement expansion card
- Experimenter box with plug-ins



Main editing window with example RLC circuit.



Setting up non-ideal step excitation.



Output graph showing result of transient analysis with Monte Carlo tolerance sweeping (5 runs only), the step input and the cursors can also be seen.

special input, and or output, need to be added.

To enhance presentations, teaching materials or other literature, text – including mathematical expressions – may be added to the circuit diagram. Pressing F9 automatically inserts the component value as a label on the diagram. Alternatively, the component label can be entered manually and customised.

Symbols are connected with the wiring tool and this is one of TINA's weaker features. While better than some packages, when first wiring up a circuit in the usual way, care is needed as it is reasonably easy to fail to connect things up.

DesignSoft is currently working on improvements in this area which I am told will be ready by the time the production version becomes available. Even so, using the mouse in conjunction with 'hot keys' makes wiring much easier. Also, if a circuit element is not connected, it is highlighted before analysis, and pressing DEL gets rid of any extra wires.

Wiring is made up of vertical or horizontal segments. These can start or end on any grid point – not just at pins. Segment lengths can be changed but if a mistake is made it is probably easier to delete the original and add a new segment of wire. This also means that if a component is moved the wiring stays put, unless all the wiring segments are selected as well. Moved components then have to be reconnected and the original wires deleted.

The above problem is only relevant when one component is moved. You can move whole circuits or circuit segments and the wires stay connected. DesignSoft says it is currently working on an upgrade involving rubber banding, which will solve this inelegance.

With the exception of the virtual instruments, almost all analysis is controlled from the menu. The usual Windows type dialogue/control boxes make it reasonably easy.

Analyses include mixed-mode

There are three basic types of analysis – namely digital, analogue and mixed mode. For a purely digital circuit you can choose whether or not to look at delays, but all high/low transitions are considered ideal – i.e. almost instantaneous. In analogue mode, a full simulation takes place and in mixed mode, propagation delays of the digital parts is also taken into account.

A digital-only circuit can optionally be run in mixed mode. In mixed-mode analysis, again you can choose to include delays, but in this case, rise times and slopes are calculated. In analogue modes, options include dc, ac, transient and noise. Having run a transient analysis, a Fourier series or spectrum can be obtained via the process menu.

Particular component parameters can be swept to determine their effect on a circuit. The component can also be optimised to a target or maximum/minimum value. Component tolerances can also be varied using various distributions or worst case. This is useful for seeing how sensitive a circuit will be to real component variations.

The analysis can be run at any desired temperature or swept over a range of temperatures.There is also the option of running a PSpice analysis by first generating a netlist.

Output facilities

Standard analysis output is a graph, or set of graphs, in a new window. There is also the option of using a virtual multimeter, oscilloscope, signal and logic analyser.

The graphs windows allows the placing of a moveable cursor that can track any curve to obtain the x and y values. To aid presentation, etc lines and circles can be drawn on the graph as well as text and labelling. The graphs can be rescaled and the annotation of axes can be changed before printing.

Libraries

A library catalogue compiler – running in DOS only – is available, allowing users to add their own parts into the library. This was not included in the review release.

The method as described in the manual uses a text file that contains the various component modelling details as well as symbol drawing directives. This is compiled into the binary catalogue that *TINA* uses.

Additional features include matching hardware

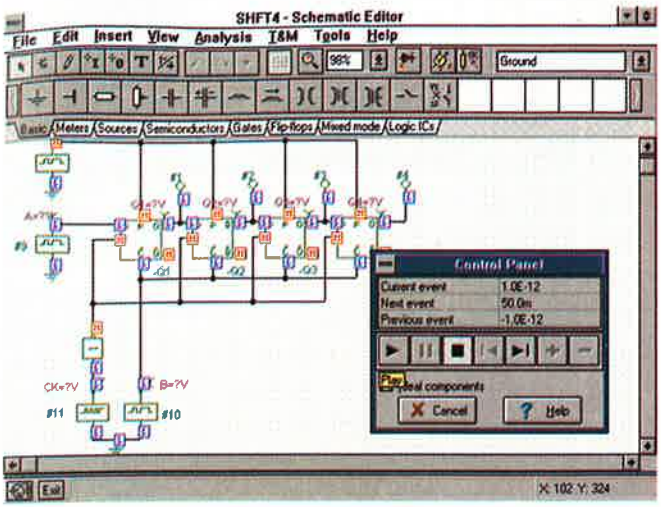
A mathematical interpreter is provided to allow the entry and evaluation of expressions and equations. Other uses include plotting results and defining arbitrary signals for circuit stimulation.

As already mentioned there are also features included that can assist in the teaching of electronics. There are modes for training and examination that provide for students' exercises preassigned by a lecturer. Fault simulation is also a useful feature for the teaching environment.

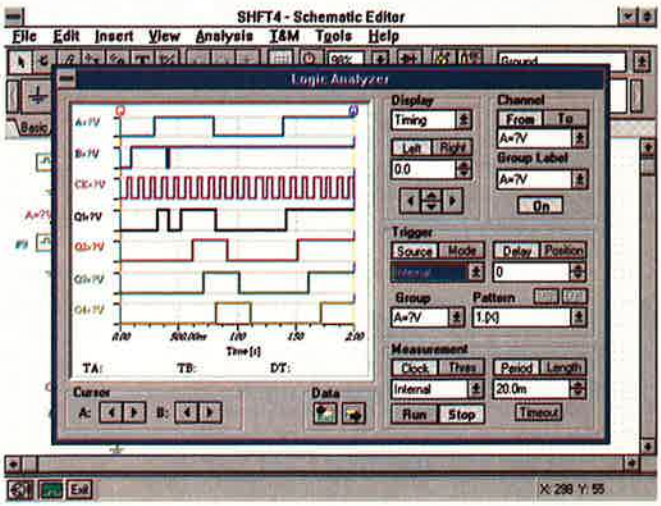
Matching hardware, for use with *TINA*, is available but was not reviewed. A plug-in instrumentation *PC* card – known as *TINAlab* – provides a multimeter, oscilloscope and signal generator under *TINA*'s control. User interfacing for these features is provided by the virtual instruments. This allows a real circuit to be built and directly compared with the simulated version.

Further available hardware includes an experimenter box with a breadboarding area that connects to *TINAlab*. This also has a slot for plug-in modules such as a fault insertion card or a digital measurement card.

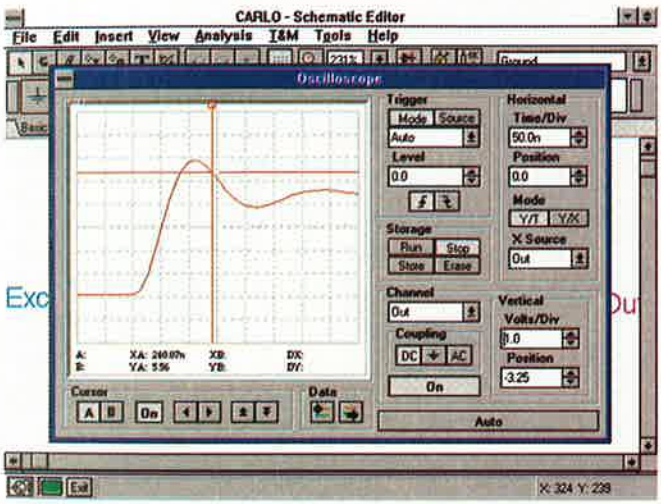
This feature of matching hardware for real world interfacing is something quite rare in simulators and provides new scope to their use.



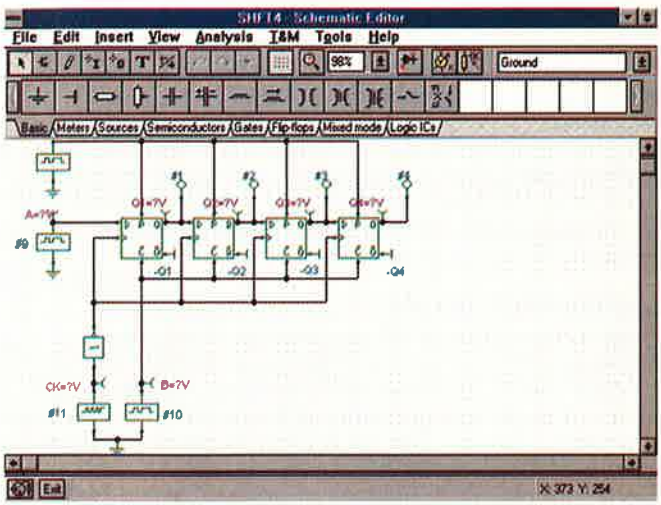
Four-stage shift register showing logic states of nodes and step-by-step analysis control control.



Virtual logic analyser showing resulting states of shift register circuit.



Virtual oscilloscope showing response of RLC circuit.



4 stage shift register.

Capabilities, requirements and manuals

TINA's capacity is stated as being 1000 components and 2000 nodes, dependent on memory. But the actual relationship between capacity and memory size is not given. The minimum specification for the package is a 386SX running, Windows 3.1.

Generally, the manuals are good, but some areas are not explained very well. For example, the different methods of input and output are not made clear. On the other hand, good attention is paid to the various components and to the simulator itself. Appendices covering the library compiler, interpreter, additional hardware and the educational aspects are also good.

I found some inconsistency in the getting started section. The screen shots did not always show what had been described in the text. For example, a meter symbol that has been placed on a circuit as instructed in the text is different from that shown in the screen shot.

In summary

Overall this is an excellent package, versatile, value for money and easy to use. It poses a serious threat to the competition. TINA scores particularly well with the ability to take into account tolerances, do parameter sweeping and being able to define new symbols as well as simulation models.

The features to aid training and presentations are very good and the additional instrumentation hardware could be very useful.

The supplier has indicated various forthcoming features which – if they don't fall into the everlasting 'coming real soon' category of upgrades – will usefully enhance *TINA*.

Top of the list must be an interface to a pcb layout tool. This is a natural progression as what often happens is that a simulated circuit has to be re-entered into the pcb system before layout can commence. Improvement to the schematic editor giving 'rubber banding' of the connections is also promised.

Macros or the ability to make sub-circuit blocks from a circuit to give a form of hierarchical design could make the handling of larger circuits easier.

Although not mentioned, the ability to have component parameters visible or not would be very useful instead of having to enter them separately as a text label or automatically via the F9 function key.

Beyond these features, what would make a very interesting application would be the ability to use the simulator itself as a sub-circuit. That is by using the hardware to take input from and provide output to the real world *TINA* would act as a virtual breadboard. ■