Avenue de l'Europe
ZI la Blanche
59270 BAILLEUL
FRANCE

Tél. +33 (0) 3 28 41 10 10
Fax. +33 (0) 3 28 42 22 00
Email. contact@cotel.fr
Website. www.cotel.fr

# HEXAN

# MANUSOFT SOFTWARE

## USER GUIDE

# Table of contents

# 1 Manual versions

| Manual version | Software version | Distribution pack | MODIFICATIONS |
|---|---|---|---|
| 7.00 | 9.10 | 2.04 | Installation of USB driver<br>Settings of time delays for for PC directly in MANUSOFT<br>DC components with harmonics option in MANUSOFT. |
| 7.04 | 9.27 | 2.23 | New USB driver |

# 2 Overview

COTEL thanks you for having bought a HEXAN. If you have any comments or problems in using this new product range, please do no hesitate to contact us.

Hexan is a test set for protection relays designed for electrical networks.

It contains the following parts:

The tester itself.
A PC running the control software.

MANUSOFT software allows you to test this kind of relays :

- Current (Maximum or minimum)
- Voltage (Maximum or minimum)
- Decoupling
- Recloser
- Directionnals
- Différentials
- Homopolar power protection
- Measurement converter
- Energy counter
- And many applications

This software is user-friendly and presents to you a manual interface like test case with potentiometers

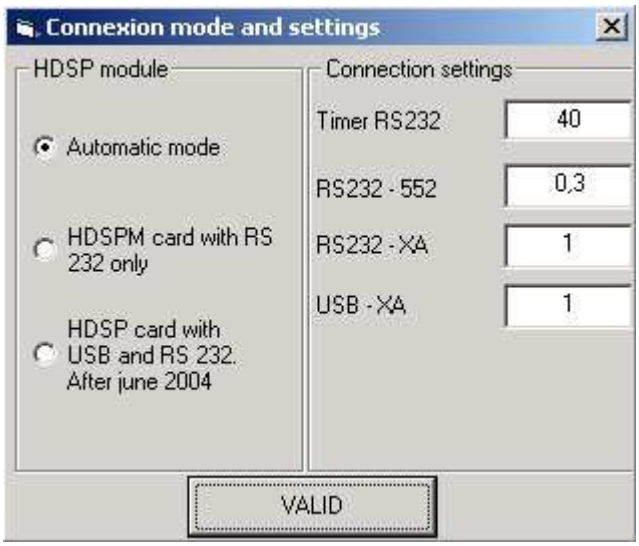
# 3 Installation

The software has been designed to work in a 32-bit environment and as such will work with Windows 9x/NT/2000/XP, but not Windows 3.x.
If you have a previous version of MANUSOFT or PROSOFT you should uninstall it before installing the new version. You can uninstall it by choosing Control Panel from the Settings submenu of the Start menu and then double-clicking the Add/Remove Programs icon. Then select MANUSOFT or PROSOFT and click the Add/Remove button.
The installation will not delete any saved test reports and settings. If these are saved in the same directory as the MANUSOFT or PROSOFT application, or in a subdirectory of these directories, the uninstall program will simply tell you that it could not remove all the components of the application.

## 3.1 For fast laptops PC with RS232

Since the processor have speed over 500 MHz many laptops have some problems to synchronize the RS232. To solve this problem, you can configure the file : **"compteurRS232.cfg"** for **Pack 1.14 distribution** in the application directory : **"c:\program files\cotel\hexan software".** You can open the file with notepad. The default value in the file is 0, this value is for the slowest PC (<P3 500 Mhz). For example, in our laboratory a value equal to 20 is useful for P4 1,7 Ghz.
If your PC doesn't communicate with the HEXAN you can put a greater value in the file. Don't forget to save the file before launch the program.

It's possible to change directly the value (Timer RS232) in MANUSOFT with the menu "Configuration\Connexion settings" :



## 3.2 Driver installation for USB connexion

**THE USB CONNEXION IS AVAILABLE WITH WINDOWS 98/2000/XP. If you have windows 95 ou NT, you should use RS232.**

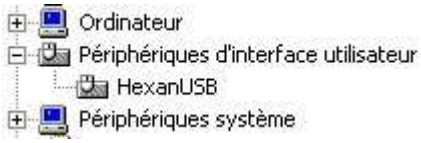
YOU MUST INSTALL THE DRIVER BEFORE USING MANUSOFT

IT'S NECESSARY TO FOLLOW THE INSTALLATION STEP IN ORDER

For a RS232 it's not necessary to install any driver

.

I'ts necessary to uninstall the old driver (used until pack 2.09). To delete it, you must be in USB connection with the HEXAN. The item to uninstall with is in **USER INTERFACE PERIPHERALS** and is called **HexanUSB**. To delete it, you can see the item with HEXAN plug on USB.




The new driver will be load in : **LibUSB-Win32 Devices** and is called **Hexan.**

**Installation of driver file « hexan.inf » :**

- The HEXAN must be out of voltage
- Plug the USB on your computer
- Plug the USB on the HEXAN
- Swith ON the HEXAN
- Windows® will launch the driver acknowledge
- Select the **«d:\driver\Hexan.inf » or C:\ProgramFiles\Cotel\HexanXA\Hexan.inf »**
- Let windows® install the driver. If windows® show you the driver is not registered click on « Continue ».
- When windows® have closed the installation, Switch OFF the HEXAN and after a few seconds switch ON the HEXAN.
- Now you can launch MANUSOFT software.

At the end of the installation driver you can see the item like below if the HEXAN is connect to the PC with USB.

## 3.3 Starting up

Before using the tester you are advised to thoroughly read the HEXAN/POWHEX User Guide.
To use the Hexan, connect it to a PC with USB or RS232. Then switch on the PC.
To start the manual interface choose either MANUSOFT from the Programs sub-menu of the Start menu.
This section of the manual covers the manual interface of the Hexan whether you are running MANUSOFT. When the application starts up it tries to detect the tester. The following can occur:
The application detects the tester and starts normally.

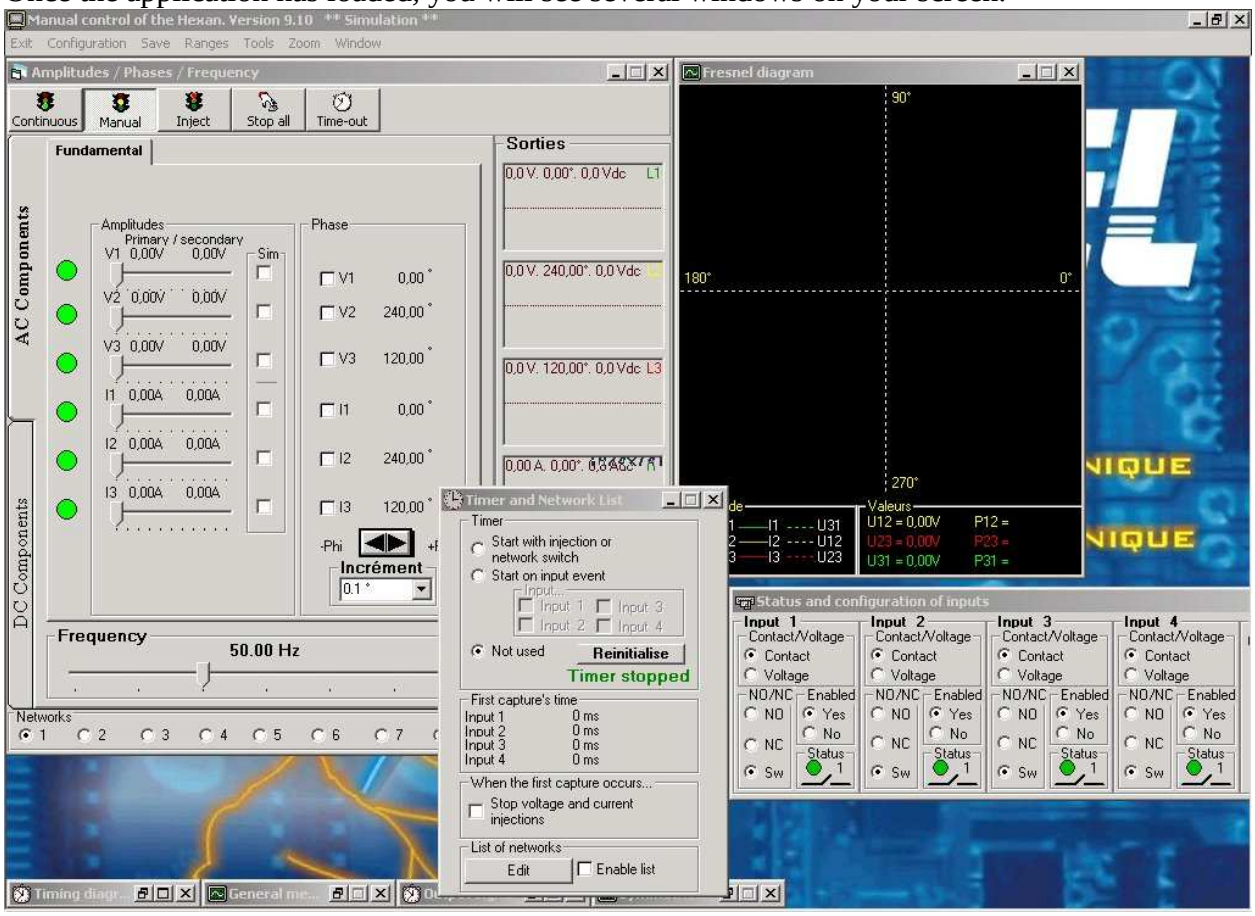
The application does not detect a tester and displays a dialog box.

You can choose to cancel, retry or ignore the error:

- Click the Cancel button to exit the application.
- Click the Retry button to run the detection procedure again.
- Click the Ignore button to run the application with or without the Hexan being connected. This is useful if you want to look at or print tests you have run, or program tests for future use. The software displays an indication that you are in simulation mode in the title bar next to the software version.

In simulation mode the configuration of the Hexan, particularly in respect of the optional amplifiers V4 and I4 is the same as with the Hexan connected. If you have already connected an external module, the software reacts as if it is connected to give you the largest number of options.
If you have the harmonics option you have the choice of working with or without harmonics at start-up.

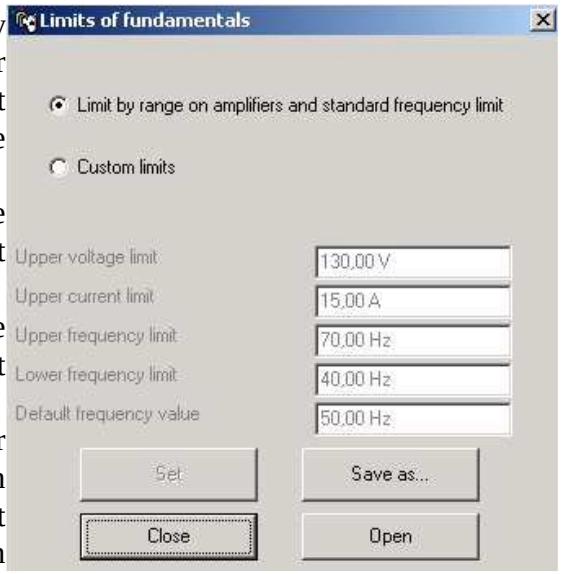
Once the application has loaded, you will see several windows on your screen.




## 3.4 Option harmonics and injection DC

At startup, if you have the harmonics option, the software asks to you if you want to inject Harmonics. To inject DC components, you must choose the harmonic option.

# 4 Injection limits window




The Injection limits window is important for safety since it allows you to set limits for the injections for both the current and voltage. It is for this reason that it is displayed on to of all others when you start the software.

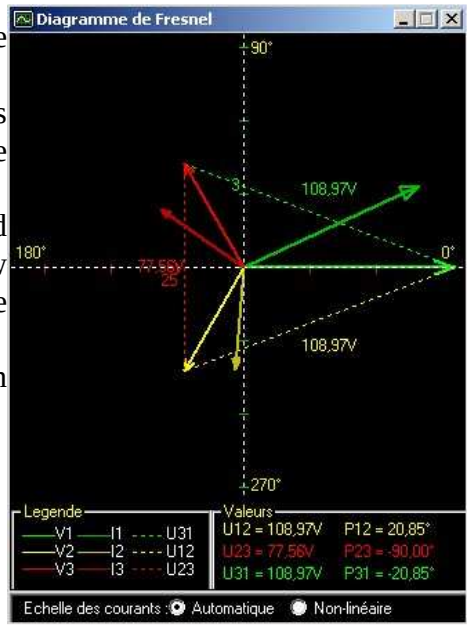
You can set the limits to be the maximum of the range of the amplifiers (the default configuration) or set custom limits.

You use the sliders to set the custom settings. Once you have made changes make sure you click the Set button.

The 'Save As…' button allows you to save your configuration in a file of your choice. The file with the name defaut.lim is reserved for the default configuration. These are the limits that are used when the software starts up.

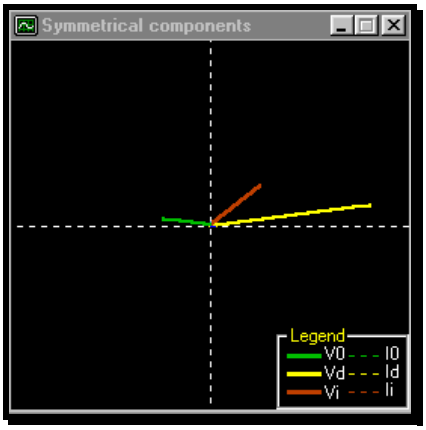
The ‘Close’ button closes the limit configuration window.
The ‘Open’ button allows you to open and load a limits file.

# 5 Fresnel diagram window




The Fresnel diagram shows the electrical values that you have set.
Solid lines represent voltages solid lines and dashed lines represent currents. The colours used are those in general use (green, yellow and red for phases 1, 2 and 3 respectively).
To modify a vector, position the mouse cursor on one end and press and hold down the left mouse button. The vector will now follow the movements of the mouse. To set the value, release the mouse button.
You can also use the amplitude and phase sliders as described in the relevant sections to achieve the same results.

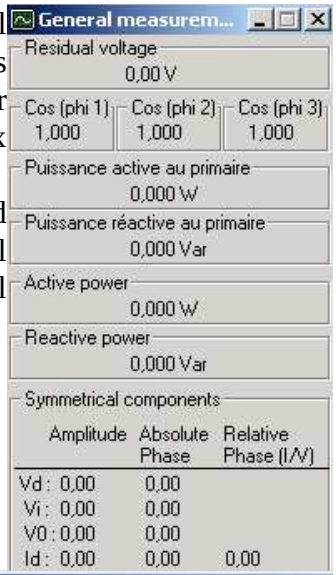
# 6 Symmetrical components window




The symmetrical components window displays the positive, negative and zero sequence components of the current network.
The Legend displays the colour correspondence.
As in the case of the Fresnel diagram solid lines represent the voltage values while dashed lines represent the current values.
You can change the size of this window and there is a zoom function to allow you to change the size of the contents of the window. The numerical values of the symmetrical components are displayed in the General measurements window.

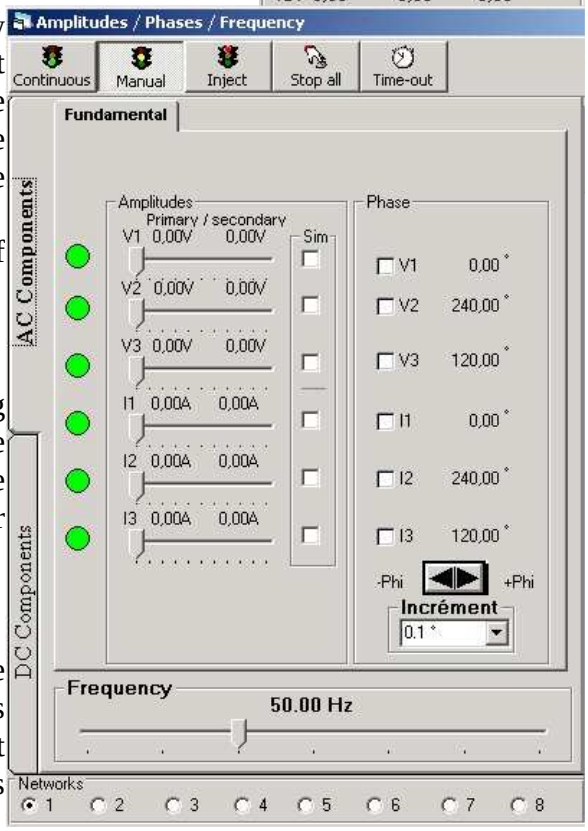
# 7 General measurements window

This window constantly displays the zero sequence and residual voltages, the active and reactive power and the cosines of the angles between V1 and I1 (for phi1), V2 and I2 (for phi2) and V3 and I3 (for phi3). It also displays the phase shift between I1 and the complex voltages.
The window displays the values of the symmetrical components and their phases. It also shows the phases of the current symmetrical components compared to the phases of the voltage symmetrical components.




# 8 Amplitudes/Phases/Frequencies window

The Amplitudes/Phases and Frequencies window contains several tabs that allow you to access eight different harmonics, including the fundamental. The order of the harmonic can be configured. You have a toolbar available that allows you to control the injection at the outputs of the Hexan.
You have the choice of the following modes of injection:




- **Continuous Injection**

You set the Continuous Injection mode by clicking the 'Continuous' button. The settings that you make on the screen are immediately available on the outputs. For example, as soon as you move a slider the output of the Hexan changes.

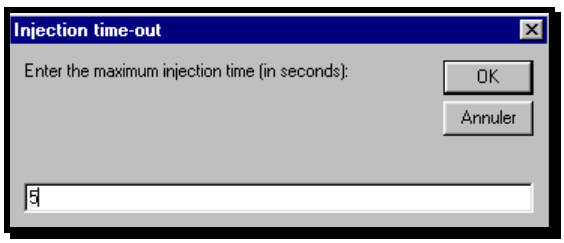
- **Injection when Inject button clicked**

When you choose the Manual injection button, the settings you have made only appear on the outputs when you click the 'Inject' button. You can adjust the settings without fear of creating a dangerous situation for the device being tested.

You have two other buttons available on the toolbar: The Time-out button and the Emergency stop button. The following sections describe these buttons.

- **Time-out button**

The Time-out command is for safety and not a way of controlling the duration of the injection. It allows you to set the time-out for the outputs of the Hexan.
The time-out starts at the same time as the injection (either when you click the 'Inject' button, or switch a network with automatic injection, or change a



network setting during automatic injection check box selected (see section ‘Network memories’).

- **Emergency stop button (Stop All)**

The Stop All button immediately removes power from the outputs and displays a message that you must acknowledge. The display of the settings does not change so that any modifications in the settings that you have made before stopping the Hexan are not lost. After an emergency stop the injection mode changes to “Manual” for safety.

## 8.1 Tab operations

### Selecting harmonic

To select the parameters for the harmonic you want to adjust click on the corresponding tab.

### Changing the order for a harmonic

To change the order for a harmonic, select the corresponding tab by clicking on it with right mouse button. A contextual menu allows you to choose the order of the harmonic. You also have the possibility of configuring all the tabs as odd or even order harmonics. Note that when you change the order of the harmonic all the phases and amplitudes are reset to zero.

### Adjusting amplitudes

There are several ways that you can adjust a setting depending on the accuracy you require.

- **Coarse adjustment:**

Move the cursor over the slider that you want to adjust then press the left mouse button, drag the slider to the desired value and release the mouse button.

- **Fine adjustment:**

Select the slider corresponding to the setting you want to change (click the left mouse button with the cursor over the slider).
Adjust the setting using the Page Up and Page Down keys on the keyboard.

- **Very fine adjustment:**

Select the slider corresponding to the setting you want to change and adjust the setting using the left and right arrow on the keyboard (← and →).

- **Manual setting**


Move the cursor over the value display at which point the cursor will change into a hand symbol indicating that you can manually enter a value.
Click the left mouse button. The software displays a dialogue box similar to that opposite. Type in the value you want and click the OK button, or click the Cancel button to close the dialogue box without making any changes. Note that the dialogue box contains the current setting for the slider.
You should note that the values that you have set here are not necessarily available on the outputs of the Hexan. For more information see the section covering the Control window.

### Simultaneous amplitude adjustment

To adjust several voltages or currents simultaneously click the corresponding 'Sim' check boxes to lock the sliders. In this situation adjusting one slider will also adjust all sliders that are locked to it. You have all the adjustment options described previously.
For all the settings you can select the required slider by pressing the Tab key on the keyboard after having selected the Amplitude window.
You can also manipulate this window in the same way as the Fresnel Diagram (see the sub-section "Closing the window" in the section "Fresnel Diagram").
The units are Volts and Amps for the fundamental harmonic, and percentages for the others.
You can reset all harmonics to zero at any time by clicking the "Disable harmonics" button.

### Phase adjustment

Select the vector or vectors that you want to change the phase for by clicking the corresponding check box and then click on the Dephaser (  )
Each click on the Dephaser adjusts the phase of the selected vectors by one increment. To adjust this increment, choose the increment you want from the Increment drop down menu.

When you change the phase of the fundamental, you can ask the software to change the phase of the corresponding harmonics by clicking the Change Harmonic Phases check box on the Fundamental tab.
You change the increment by choosing the increment you want from the Increment drop-down menu.
You can change which vectors you want to modify at any time.

- **Manual adjustment**

You can make a manual adjustment of the phase in the same way as for the amplitudes.
You can also manipulate this window in the same way as the Fresnel Diagram (see the sub-section “Closing the window” in the section “Fresnel Diagram”).

### Frequency adjustment

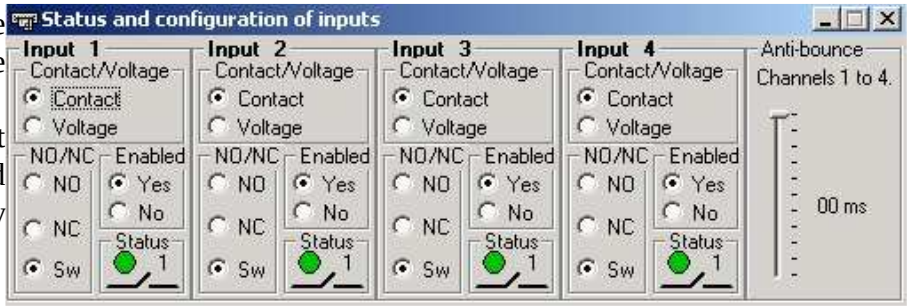
This slider works in the same way as the amplitude sliders. You can adjust the frequency between 0.01Hz and 100Hz.

### Choosing a network

To simplify the manipulations necessary when testing protection devices the Hexan contains eight user-defined networks. You simply select the network that you want to use (in the Network zone) then program your settings. To switch from one network to another, click on the number of the network.
If you want to copy the settings of one network to another, click the source network number and choose the destination network number from the contextual menu. You can easily switch between networks using these techniques.

**Note: a red or green circle next to a slider indicates whether there is an error or not on the corresponding generator (red = error, green = no error).**
**An error can occur if you exceed the maximum power rating of the generator or if the generator has overheated. If a generator overheats a red indicator on the front panel lights, although it has not been disabled by software (see the section ‘Hexan configuration’ for a description).**

# 9 Status and configuration of the inputs window




This window affects the four timer inputs of the Hexan.
Each input is independent and electrically isolated and can be individually configured.
You can do the following:
Configure the input to react to signals by contact or voltage.
Configure the input to react to changes in state for contacts that are Normally Open (NO), Normally Closed (NC) or are changing state (Sw).
Enable or disable each of the inputs.
Configure the anti-bounce filtering on inputs 1, 2 or 3.
The window shows the state of each input in the Status area.
When you configure the input to Voltage, ‘NO’ corresponds to the appearance of a voltage (or rising edge), NC’ to the disappearance of a voltage (or falling edge) and ‘Sw’ to the appearance or disappearance of a voltage.

These configurations allow the triggering of the timer.
The window displays the state of the inputs as a 1 or 0 along with green or red round indicator to improve visibility. The following table shows the correspondence.

| Status | Contact | Voltage |
|---|---|---|
| **1** | Contact open | Voltage absent |
| **0** | Contact closed | Voltage present |

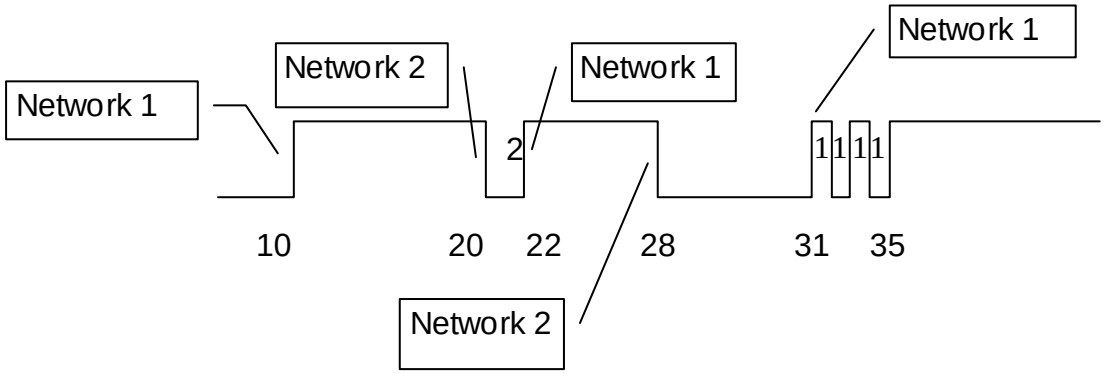
## Anti-bounce filtering

Anti-bounce filtering occurs at two levels:
Timing diagram display
Any pulses with a duration less than the selected value are ignored. You can alter the duration of the anti-bounce even after having taken the measurements. The displays are automatically updated.
Network lists (see the section 'Timer and Network List window') the list of networks can occur more rapidly than expected due to the presence of oscillations. To avoid this you can set the anti-bounce filtering to stop the change to the next network in the list if two consecutive events occur in an interval less than that programmed in the filter.

For example, if the anti-bounce filter is set to 2ms, the following network list is created:

Network 1
Network 2
Network 1
Network 2
Network 1
Network 2
Network 1
Network 2

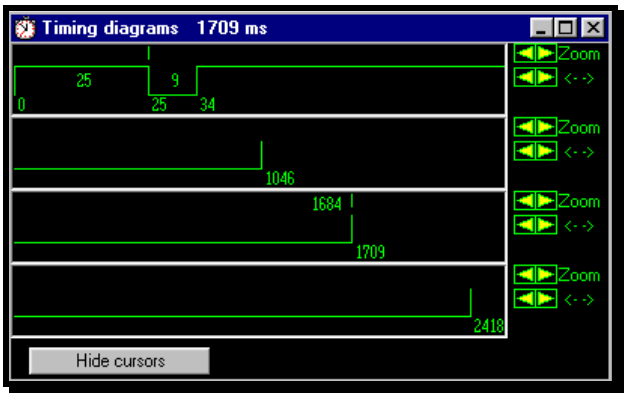
The following events arrive. The actions taken by the Hexan are indicated on the same timing diagram.




The pulses at the end are ignored if they are less than one millisecond apart.
Note: When anti-bounce is being used the last display state may not correspond to the actual state of the timer input. In this case you should modify the anti-bounce settings.

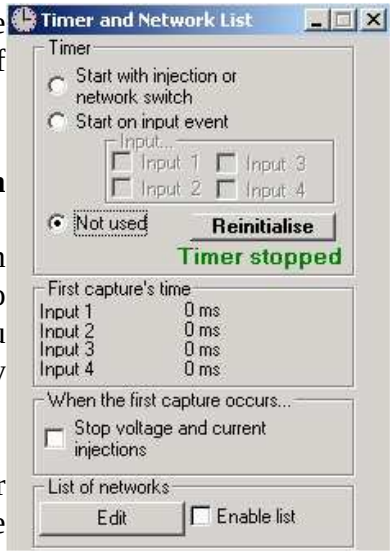
You can also manipulate this window in the same way as the Fresnel Diagram (see the sub-section "Closing the window" in the section "Fresnel Diagram").

# 10 Timing diagram window




The timing diagram has four zones that correspond to each of the inputs and are numbered 1 to 4 from top to bottom.
In each zone you can see:
The levels (Inputs 1 and 2) or edges (in the case of Inputs 3 and 4).
The times in milliseconds at which the levels have changed or an edge has been detected displayed at the bottom of the zone. (For example, 31 for Input 1)
The duration in milliseconds between two detected edges or level changes displayed in the middle of the zone. (For example, 81 for Input 1).
You can change the size of the timing diagram window in the same way as that for the Fresnel diagram. You can even expand it to full screen size by clicking the symbol.
You have the following tools to help you analyse the time measurements:
A zoom function on each input which allows you to increase or decrease the time scale. (Zoom button).
A time scroll using the left and right arrows (← and →). You can therefore easily check the duration of a pulse at any point in the timing diagram, or easily display the fast and slow cycles of a rearming device.
Two cursors that allow you to measure time intervals between points in the timing diagram window. For example, suppose you want to measure the time between two points A and B:
Position the mouse cursor over point A (the software displays the time at the cursor point in the title bar of the timing diagram window) and click that point. The software displays a message Time measurement in the same title bar and a cursor at the time A.
Now move the mouse cursor to point B and click the mouse again. The software displays the time between the two points next to point B, behind the corresponding cursor. In the example above a time measurement has been made between instant 31 of Input 1 and instant 58 of Input 3.
**Note : These tools are only available when the timer has stopped.**

# 11 Timer and Network List window




The Timer and Network List window allows you to control the various ways that an injection can start and also the list of networks.

## Timer section

- **Start with injection or network switch option button**

When you select the Start with injection or network switch option you tell the timer to start when you switch from one network to another during automatic injection. The timer will start when you switch the networks and status area of the window will display Timing… instead of Timer stopped.

- **Start on input event option button**

When you select the Start on input event option you tell the timer to start when an event occurs on one of the selected inputs. The timer will start when the event programmed in the Inputs window (i.e. Contact or Voltage) occurs on the relevant input. You still have access to the list of networks described previously.

- **Not used option button**

If you are not using the timer click the 'Not used' option button.

- **Stop/Reinitialise button**

The Stop option button stops the timer and allows you access to the timing diagram tools (zoom and scroll). The text on this button changes to Reinitialise when the timer is stopped. If you click this button again you reinitialise the timing diagrams. The software prompts you to confirm your action.

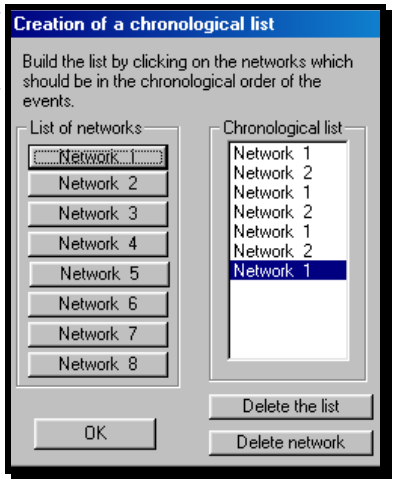
- **Status area**

The status area is at the bottom of the timer section and displays the status of the timer in green text.

## Network list section

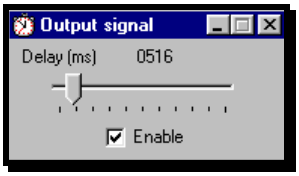
- **Edit button**

Let us suppose that you want to measure the reaction time for a relay.
You have programmed the 'Good' network into Network 1 and the 'Trip' network into Network 2. You then click the Network 1 button so that the 'Good' network is sent to the relay, and then you configure the timer to start on an injection by choosing the option Start with injection or switch network from the window. The test starts when you request the injection of Network 1.
If you want the network becomes 'Good' as soon as the timer has received the signal from the relay you will have to watch the state of the input and quickly click the Network 1 button. However, instead of performing this delicate manoeuvre you can tell the software to do it for you by creating a network list.
The list of networks represents the series of networks, which must automatically be invoked in chronological order when events are detected on the timer inputs.
In the case of the example given above it is necessary to create a list that only contains Network 1 so that the 'Good' network is sent to the relay once the signal has been received.




To create the list click the Network list button and then simply click the Network buttons in the order they should occur.
In the example above, you need to create a list containing only Network 1 so that the network returns to a pass state as soon as the relay sends a signal.
To create a list, click the Edit button in the List Network zone and then create your chronological list by clicking the network buttons in the order that you want them used.
You also have the following options:
Delete the entire list by clicking the Delete the list button.
Delete a network by clicking on the entry in the list and then clicking the Delete Network button
When your list is complete click the OK button.
The list of networks is controlled completely by the Hexan and ensures that the actions happen in real time. You can use these functions to measure the response time of the relay when returning to its quiescent state.
Once you have created a list of networks, do not forget to enable the list by clicking the Enable List check box.

# 12 Output signal window




In addition to the logic inputs associated with the timer, the Hexan has an output with a programmable delay. When the function is enabled it switches the contact n°1 after a programmed delay has elapsed after the timer starts.
This function is therefore easy to use since you only have to move the slider to set the delay that you require. When the timer starts (whether by the start of an injection or by a timer event on an input) the C/O contact will close after the programmed delay.

# 13 Main menu bar

The main menu is displayed at the top of the screen attached to the main application window.

## Configuration menu

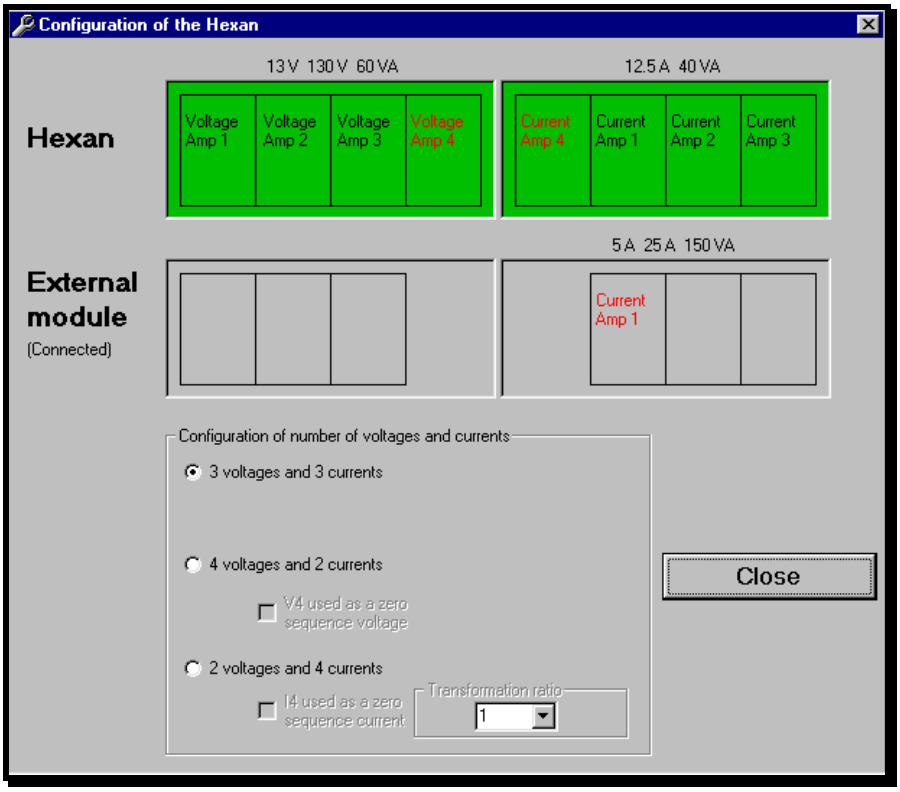
The Configuration menu controls the following:
The position of the windows
The configuration of the Hexan
The injection limits

- **Save window position**

Once you have saved the window position you can load any context that you want. When the software starts it loads the positions in the file defaut.pos. To save the position of the windows choose Save window positions from the Configuration menu and give a filename.
To load the window positions choose Load window positions from the Configuration menu and choose a file.

- **Configuration of the Hexan**

The configuration of the Hexan option allows you to select which generators that you want to use as a function of the options installed in your machine.




The Hexan has an internal generator capable of producing six sinusoidal signals. These signals can control six generators.

Depending on the configuration of your tester you can have up to eight internal amplifiers. The configuration option allows you to choose which of these amplifiers to use.

The unselected amplifiers will be disabled. An LED on the front of each generator shows if the generator is disabled or not.

The choices of configuration you have available are:

Three voltages and three currents

Four voltages and two currents

Two voltages and four currents

If you choose four voltages, you can request that the voltage V4 automatically changes to the zero sequence voltage of the network given by the voltages V1, V2 and V3. You can, of course, adjust this voltage.

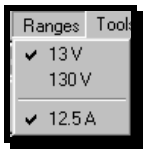
If you choose four currents, you can also tell the software to adjust the current I4 to the zero sequence current of the network I1, I2 and I3.

You can also take into account a conversion ratio, which represents the ratio between the value displayed by the slider for I4 and the actual current that the Hexan supplies. This is the only displayed value to be affected by this ratio. The zero sequence current in the General measurements window and that in the Outputs window always represent the actual current that the Hexan injects.

This ratio has a value of 1 by default

Note that the amplitude adjustment, the phase adjustment and the output display windows automatically change depending on the configuration you have selected. In these windows V4 takes the place of I1 or I4 takes the place of V3.

## Range menu

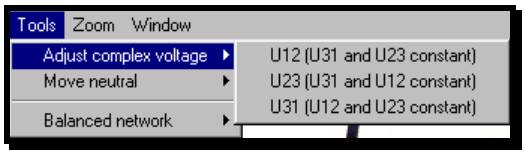



The Range menu allows you to select the range of the voltage generators. The ranges available in the menu can change depending on the configuration of your Hexan.
In the above example, the Hexan has two voltage ranges: one up to 13V and the other up to 130V and a single current range of 12.5A.

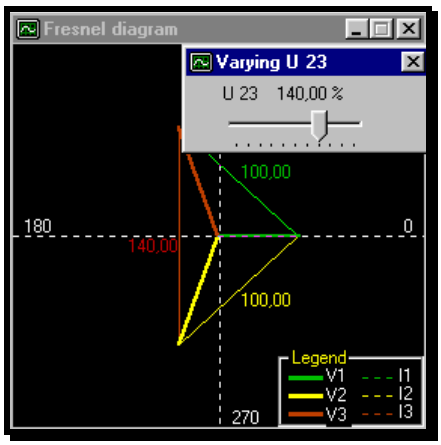
## Tools menu

The tools menu contains tools for the variation of the complex voltage and the movement of the neutral point. These allow you to make these changes easily, rather than doing them manually by adjusting the phases and voltages, which would prove very difficult.
The network must be in the balanced state before using either of these two tools.

- **Varying the complex voltage**

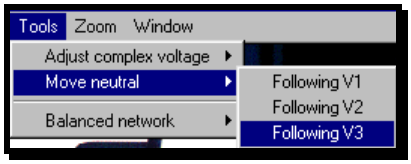



The first tool allows you to vary a chosen complex voltage without changing the amplitude of the other complex voltages and without moving the neutral point.
For example, you have a 'Good' network of 100V and you want to vary the U23 complex voltage without changing the other complex voltages and without shifting the neutral point.
Choose the U23 (U31 and U12 constant) option from the Vary complex voltage sub-menu of the tools menu.



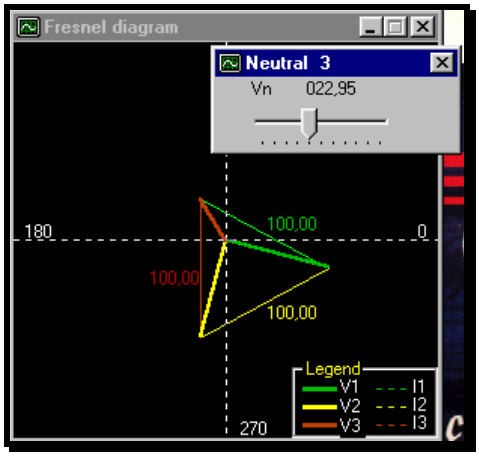

The software displays a customised slider with percentage graduations, which allows you to change the amplitude of the complex voltage that you have selected. In the current example, you will see that the voltages U12 and U31 will not change and that the neutral point will not move.
The phases and the amplitudes of the voltages are recalculated in order to maintain the settings that you want.
This function is useful for exciting just the protection device monitoring the complex voltage that you want to test without tripping any other protection devices.
The slider works in the same way as the other sliders.
To close the tool, click the ⊠ symbol situated in the upper right corner of the window containing the slider.
You may receive an error message if the voltage exceeds the selected range, or if you try to obtain a complex voltage, which is impossible with the network being used.

- **Moving the neutral point**



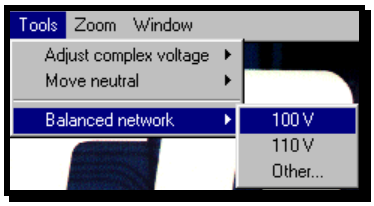

This tool allows you to move the neutral point inside an initially balanced network without changing the complex voltage values
For example, you have a balanced network at 100V and you want to move the neutral point along V3.
Choose Following V1 from the Move neutral point sub-menu of the Tools menu.

The software displays a slider, which allows you to move the neutral point from its initial position.
The software recalculates the voltage amplitudes and phases so as not to change the amplitude of the complex voltages.
The slider has the same functions as the other sliders in the software.
To close the tool, click the ✕ symbol situated in the upper right corner of the window containing the slider.
You may receive an error message if the voltage goes out of range.



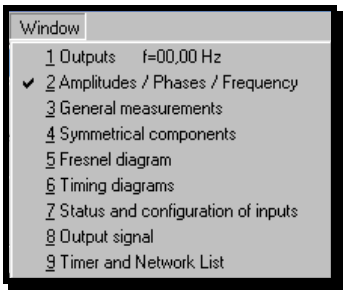

- **Balanced network**

When you need a balanced network use the Balanced network tool rather than calculating the voltages yourself.
You can choose a balanced network of 100V, 110V or any other value by choosing the Other… menu option. If you choose the Other… menu option, the software prompts you to enter a value. If you choose a value that does not correspond to the voltage range, the software displays an error message.




## Windows menu

The Windows menu contains a list of all the open windows. If a window is minimised and you cannot find it on the screen, you can maximise it from the Windows menu by choosing the name of the window from the menu.



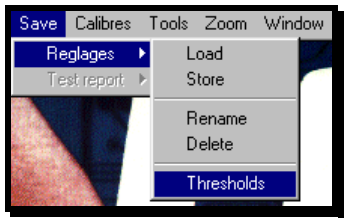

## Save menu

You can save your tests, which includes all the network settings, the threshold, the configurations, the window positions and the time measurements. Once saved, you can reload a test, run it again and print the results.

- **Thresholds**

**Saving a threshold**

When you have determined the threshold for a protection device you should save it to keep a trace. The threshold contains the settings for the threshold and also the measured value for the response time.
To save a threshold, choose the Thresholds option of the Settings submenu of the Save menu.
The software displays a window similar to that opposite containing a list of the thresholds that have previously been saved.
Click the Add threshold button and then enter the name for the threshold. The software will display the name in the list.

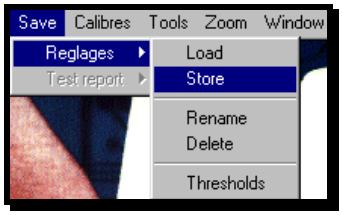



**Loading threshold settings**

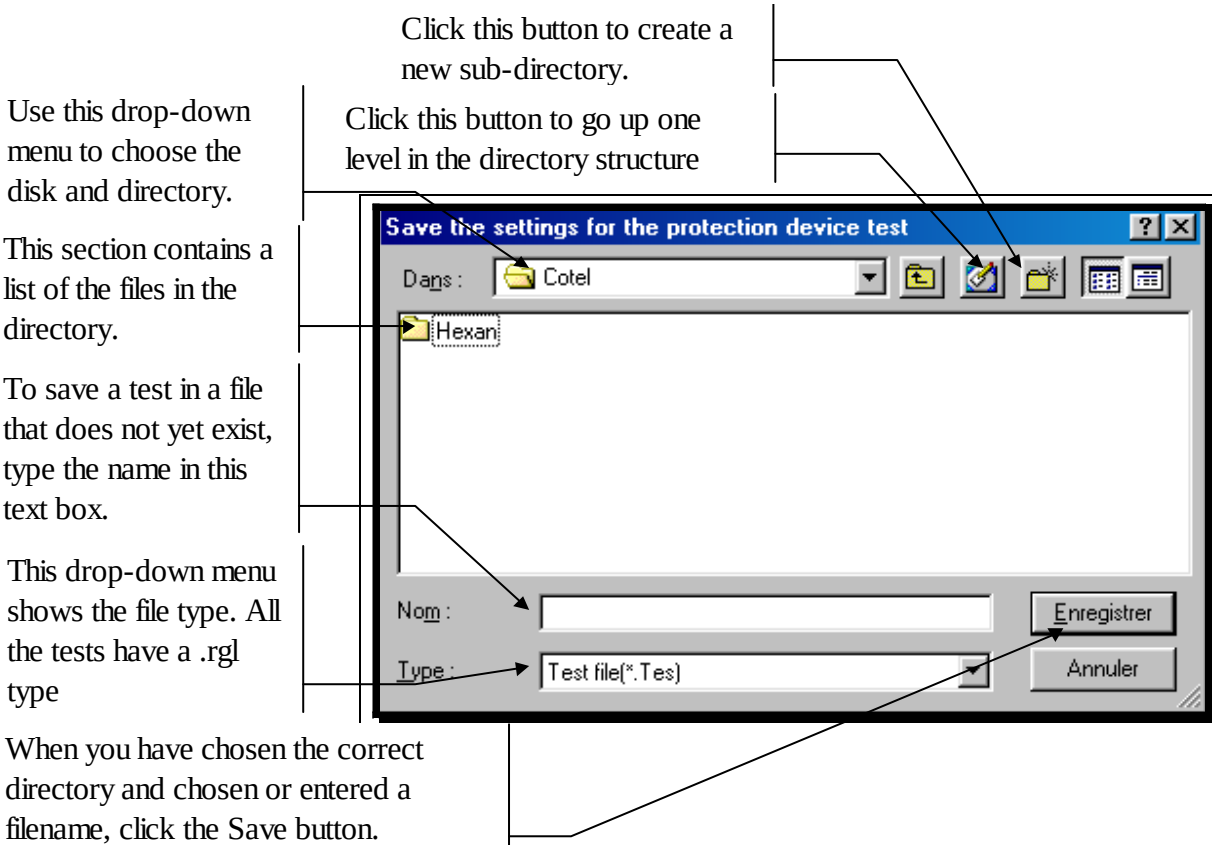
To load the threshold settings and the associated timing diagrams, choose the name of the threshold from the list.

- **Saving settings**

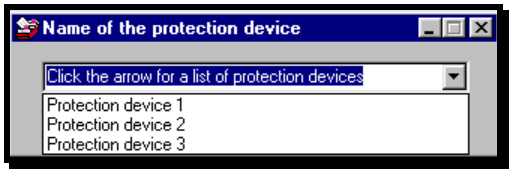
To save a test, choose Store from the Settings sub-menu of the Save menu.




The software displays the standard Windows Save dialogue box from which you can choose the disk and directory and the filename for the test you are saving. The following illustration describes the dialogue box.




Once you have entered all the information concerning the path and filename and clicked the Save button, the software prompts you to enter the name of the protection device with the following window.



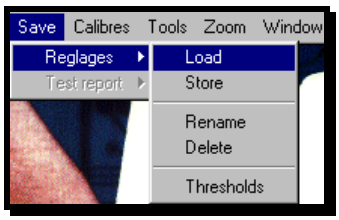

The list box contains the names of the protection devices that you have already tested to avoid you having to retype a name for a previously tested device.

Type a name in the text box or choose one from the drop-down menu and click the OK button.

The software now saves all the test parameters, i.e. the settings for the threshold (current and voltage and timing diagrams), configuration of the timer inputs, the selected ranges, the configuration of the amplifiers I4 and V4 (if installed), and the position of the windows.

- **Loading settings**

To load a test, choose Load from the Settings sub-menu of the Save menu.

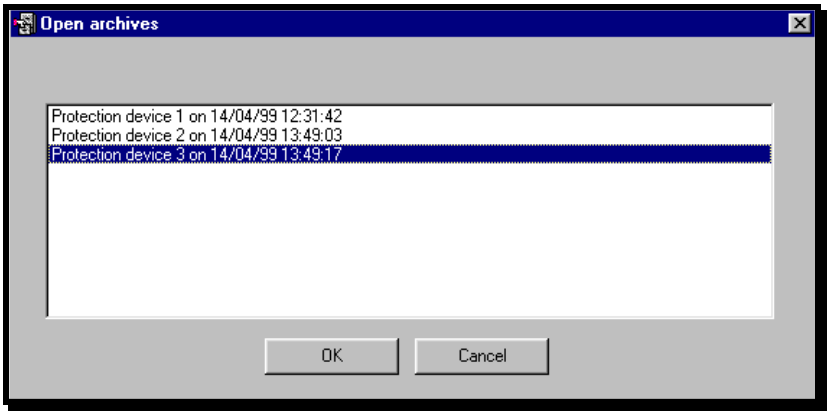



Use the window below to choose the disk and directory and the file containing the settings that you want to load.

This window has the same functions as the windows described previously for storing the tests.

Once you have selected the file click the Open button.

The system loads the file and displays a list of settings that it contains. The list contains the names of the protection devices, and the date and time that you saved the test.




Choose the settings for the test of the protection device that you want from the list and click the OK button. The software then loads all of the test parameters, i.e.: the threshold (settings and timing diagram), the contents of the various networks, the list of networks, the range, the configuration of the Hexan, the position of the windows, etc.

For safety reasons the Automatic injection option is not restored and is replaced by the Manual injection option. This prevents the Hexan making an injection by default and requires the user to explicitly choose an injection.

- **Changing the name of a setting**

The name of the setting also contains the name of the protection device since the software only understands a protection device by the settings programmed into the Hexan.

To change the name of a setting you have to load the settings file and then choose Rename from the Settings sub-menu of the Save menu. You then click on the name of the setting that you want to change in the list. The software displays a dialogue box for you to enter the new name.

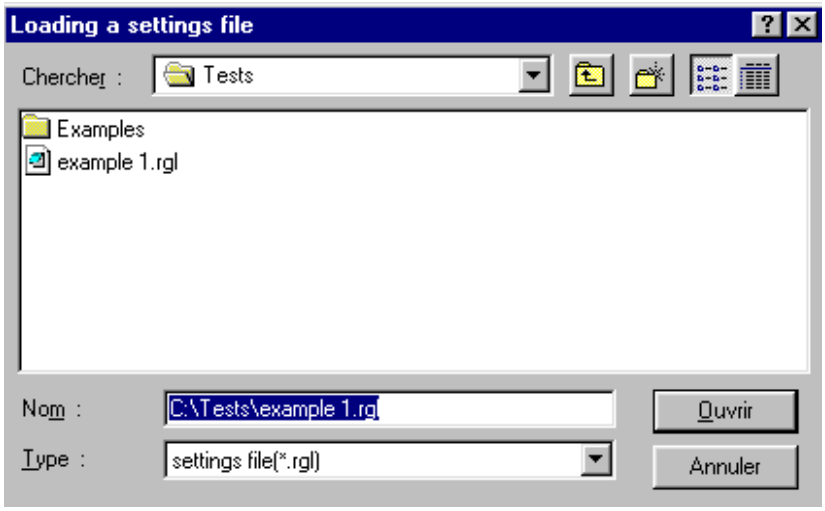
- **Deleting the name of a setting**

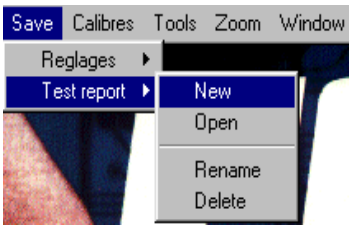
You delete the name for a setting in a identical way to that above, but in this case you do not have to enter a new name, simple click on the setting to delete.

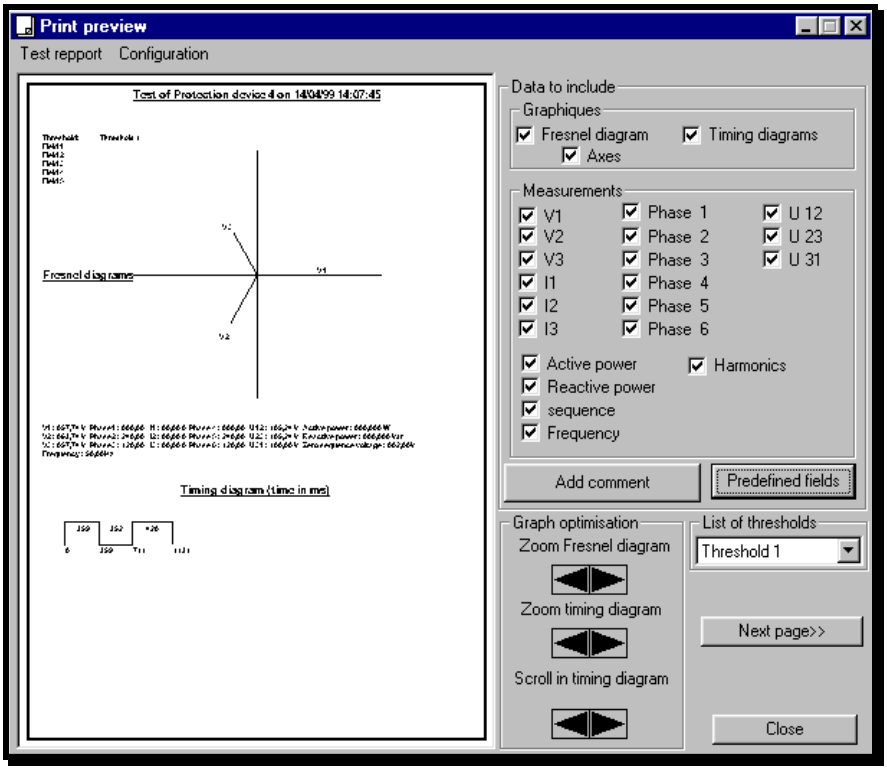
## Editing and printing a test report

To print a test report, you will have to have already loaded a set of saved settings. Then choose to print a test report as shown opposite.

The software displays a dialogue box from which you can print the report.






This window contains a print preview and has various tools to allow you to optimise it.
Two zoom functions, one for the Fresnel diagram and the other for the timing diagram.
A scroll function for the timing diagram to ensure that the measurements are readable.
Nearly all the options available so that you can display just what you want on the report.

You can, for example, choose whether to display the complex voltages, the zero sequence voltage, the axes of the Fresnel diagram, the phase, the powers…

The option for adding text by clicking the Add comment button.

You enter your text when prompted and then position it on the report with the mouse where you want it to appear. You have a maximum of 30 text zones. You can later move any text zone by dragging it with the mouse, or modify the text by double-clicking it.

User configurable fields

You configure these fields from the Configuration menu in the Print preview window. The configuration consists of giving a name to each of the ten fields. You can, for example, give the first the name “Substation” and the second the name “Section”. You fill in these fields by clicking the Predefined fields button. The field name is the same for all test reports.

For versions with the Harmonics option, a second page displays the settings for the harmonics. To display this page click the Next page button.
The window also displays a list of saved thresholds to allow you to easily choose between the various tests made on a protection device by simply selecting the required threshold.
Note that whenever you choose an option you must click the Apply button for it to appear on the report.

- **Printing the test report**

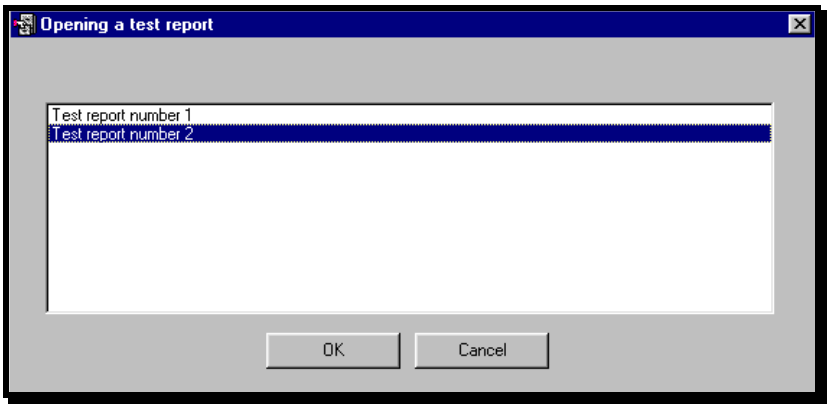
When the print preview is to your liking, click the Print button to start the print job on the default printer on your PC. Use the Windows Printers option in the Start menu to change the printer.

- **Saving a test report**

To save you test report, choose Save As… from the Test report sub-menu of the Save menu. The software will ask you to enter a title for your test report.
You can save several test reports for each test.

- **Opening an existing test report**

Before opening a test report you should load the associated settings and then choose Open from the Test report sub-menu of the Save menu. The software displays the following dialogue box.




The list in the dialogue box contains the reports for the currently loaded settings. You should choose which report you want to use.

- **Changing the name of a test report**

Before renaming a test report you should load the associated settings and then choose Rename from the Test report sub-menu of the Save menu.
The software displays a dialogue box containing a list of reports for the currently loaded settings. You should choose which report you want to rename by clicking on it in the list and then enter the new name.

- **Deleting a test report**

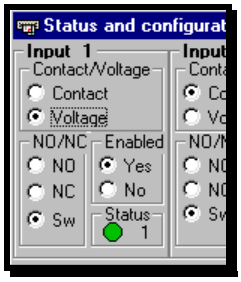
Before deleting a test report you should load the associated settings and then choose Rename from the Test report sub-menu of the Save menu.
The software displays a dialogue box containing a list of reports for the currently loaded settings. You should choose which report you want to delete by clicking on it in the list.

| | |
|---|---|
| Note: | The outputs of the Hexan remain at their programmed values during the whole time that you are viewing or creating a test report to ensure that the connected protection device does not change state. If you do not want to have the outputs operational for a long time, reset them to zero before consulting the reports. |

# 14 Example to test an over-current relay

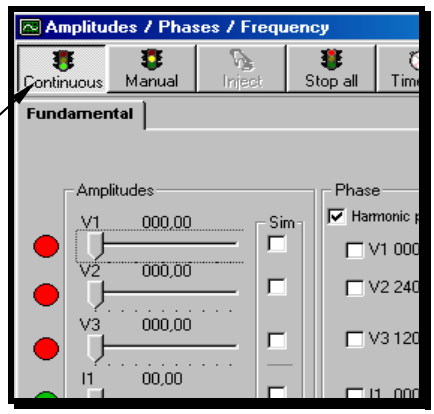
To test this relay, like most relays, requires two stages:
Detecting the tripping threshold
Measurement of the tripping time
Use the following sequence to run this test:
Connect the PC to the Hexan with the supplied cable (RS232 port of the Hexan to the COM port of the PC).
Connect the PC to the power socket on the front panel of the Hexan.
Press the On/Off button.
Start the software.
Connect the I1 current generator to the relay, and the relay trip signal to timer input 1.
We will assume that the relay returns a voltage trip signal so we have to configure timer input 1 to Voltage.

Choose the Voltage option button for Input 1 in the Inputs window.



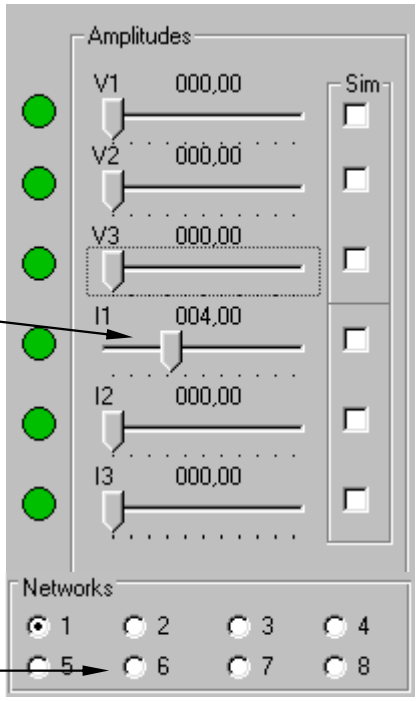

In general, you will not have to do any more configuration since the default values cover the most common requirements.
We will now look for the tripping current of the relay. It will be necessary to tell the Hexan that the outputs should immediately follow our settings.

Select Continuous injection in the control window




We now have to set the current I1 with its slider.

Use the mouse or the arrow keys to adjust the current until the relay trips.
You can listen for the relay to trip, or you can check the state of Input 1 in the Inputs window.

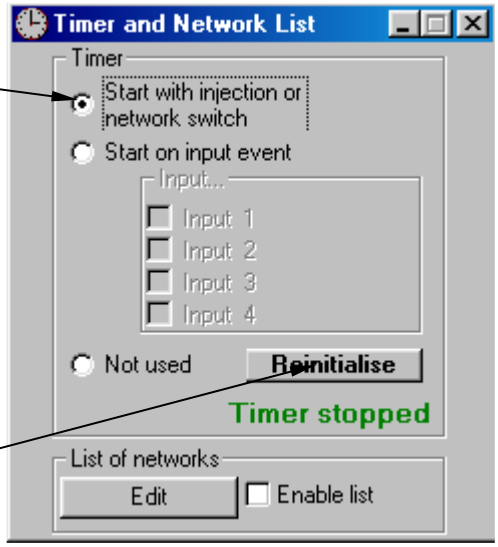



When you have found the threshold, click on the network 2. This network will be use for the network without current. The network 1 contains the threshold

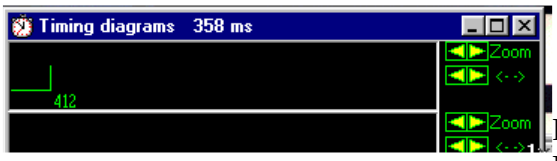
We now have to measure the tripping time of the relay.
Select network 3 and adjust the current to 120% of the threshold. You can also go back to network 1 and copy its settings to network 3 and then adjust I1 of network 3 to 120% of its value.
In summary, there are three networks used:
Network 1, corresponding to the threshold.
Network 2, the 'Good' network.
Network 3, the 'Fail' network (120% of threshold).

The tripping time of the relay is the response time when switching from network 2 to network 3. We will start by outputting network 2. Since we have selected continuous injection, the outputs will immediately be at zero current.
We now have to tell the timer to start timing.

Choose the start on injection or network switch option in the Control window. The timer will start at the moment of changeover between Network 2 (no current) and Network 3 (with a current slightly above the tripping current).




Stop the timer and zoom in the timing diagram window.




Return the relay to its quiescent state by clicking the Network 3 button and you will get a measurement of the response time of the relay on the timing diagram for input 1.

You can return the relay to its quiescent state by clicking the Network 2 button
It would also be possible to recall Network 2 automatically by creating a network list. This way once the signal had been received from the relay, network 2 would be sent to the Hexan's outputs.
To save the threshold, configure the injection to occur when the Injection button is clicked (option When Inject clicked). Click the Network 1 button and then choose Thresholds from the Save menu. Click the Add Threshold button and give your threshold a name. You can measure other thresholds (opening and closing thresholds, limits, etc…) and save these as well.
When all the tests are complete, save your settings by choosing Store from the Save menu. All your tests, your time measurements, the configurations and the window positions will be saved.


COTEL