
CONTENTS

1	General	3
1.1	Software Conditions of Use	3
1.2	Warranty	3
1.3	System Requirements	4
1.4	Typographic Conventions	4
1.1	Specification	4
2	Introduction	5
1.5	Program Contents	5
1.6	Installing and Starting the Program	5
1.7	Task Bar	6
1.8	Mouse Procedure	6
1.9	Command Selection	7
2	Registering the digimess® Device	8
2.1	Enabling the Device Drivers	8
2.2	Configuring the digimess® Devices	9
2.2.1	Device Settings	10
2.2.2	Controller Settings	11
3	Starting the digimess® Devices	13
3.1	Operation of the digimess® Devices	13
3.2	Extended Menu – Extended Functionality	14
4	digimess® Display – Visualising and Saving the Measuring Data	15
4.1	User Interface of the digimess® Display	15
4.1.1	Entry Field	16
4.1.2	Display Window	17
4.1.2.1	Scanner (Multiplot Charts)	17
4.1.2.2	Logger (Multiplot Graph)	20
4.1.2.3	Scanner & Logger (Table)	22
4.1.3	Status Line	22
4.1.4	SETUP – Parameterisation of the digimess® Display Settings	23
4.1.4.1	Data to Logfile – Measuring Data Saving	24
4.1.5	START – Measuring Values Display	26
5	Smart Sequencer – Creating the Test Program	28
5.1	General	28
5.2	User Interface of the Smart Sequencer	28
5.2.1	Menu List	29
5.2.2	Action Window	29
5.2.3	Status Line	29

5.3 Macro Library.....	30
5.3.1 Macros.....	30
5.3.1.1 <i>System Reset</i>	30
5.3.1.2 <i>Device On/Off</i>	31
5.3.1.3 <i>Device Function</i>	31
5.3.1.4 <i>Show Display</i>	32
5.3.1.5 <i>Compare Measure Break</i>	33
5.3.2 Special Macros.....	34
5.3.2.1 <i>DELAY Function</i>	34
5.3.2.2 <i>LOOP Function</i>	35
5.3.2.3 <i>CALL Function</i>	35
5.4 Beginning Work.....	36
5.5 Creation of the Test Procedure.....	37
5.5.1 Edit Functions.....	37
5.5.2 Marking the Test Steps.....	38
5.5.3 Edit Test Procedure.....	38
5.5.4 Test File.....	39
5.6 Parameterisation of the Test Steps (Macros).....	39
5.6.1 General.....	39
5.6.2 Parameterisation Interface.....	40
5.6.2.1 <i>Menu Line</i>	40
2.1.1.1 <i>Macro Settings</i>	40
2.1.1.2 <i>Status Line</i>	41
5.6.3 Parameterisation of the Test Steps.....	41
5.6.4 Comment Display.....	42
5.7 Debugger – Implementing the Test Program.....	42
5.7.1 General.....	42
5.7.2 Debugger Interface.....	43
5.7.2.1 <i>Display Range</i>	43
2.1.1.3 <i>Control Panel</i>	43
2.1.1.4 <i>Status Line</i>	44
5.7.3 Marking the Test Steps.....	44
5.7.4 Calling up the Test Steps.....	45
5.7.5 Inserting the Break Points.....	45
5.7.6 Step Mode of the Test Program.....	45
5.7.7 Test Run of the Test Program.....	46
5.7.8 Resetting the Test Program.....	47
5.8 Testing the Test Procedure.....	48
5.8.1 Continuous Run of the Test Program.....	48
5.8.2 Loading the Protocol.....	49
5.8.3 Printing the Test Steps.....	49

1 General

1.1 Software Conditions of Use

- Contents
- All rights to the software (including documentation) in particular reproduction, distribution and translation rights, remain with GRUNDIG.
 - Copies may be made for archive or replacement purposes only.
 - The client may transfer the user rights to a third party with prior consent from GRUNDIG, if the latter recognises and agrees to the user conditions. A transfer forfeits all user rights of the client.

1.2 Warranty

- Warranty Conditions
- GRUNDIG guarantees the functioning of this software when expertly installed according to this operating instructions.
 - The warranty is valid for three months after delivery.
 - This warranty is void if errors occur as a result of incorrect installation, changes to the program files or improper use of the diskettes.
 - GRUNDIG disclaims warranty regarding software, accompanying written materials and accompanying hardware.

- Product Support
- For assistance, please contact:

GRUNDIG

GRUNDIG AG

Division Instruments

Test and Measuring Systems

ZENTRAL SERVICE

Würzburger Str. 150

D-90766 Fürth

Tel.: +49-911-703-4165

Fax: +49-911-703-4465

Please read the operating instructions carefully and proceed according to the instructions.

1.3 System Requirements

IBM-compatible PC To install the Soft Workshop a PC with the following configuration is required:

Operating System:	Windows 95/98
Processor:	486DX (minimum), Pentium 100 and higher (recommended)
Hard Disk:	15 MB free memory
Interfaces:	minimum one free RS-232C interface
Drives:	3 ½-inch floppy drive, CD-ROM drive
Display:	min. 800 × 600 Super-VGA graphics or higher use with laptops possible optimum display with screen resolution of 1024 × 786

Software Sets Appropriate device drivers are required in order to use the Soft Workshop for creating PC-based measuring technique applications. So-called Software Sets with the following contents are available for every **digimes**® device:

- Labview-runtime-environment
- specific Labview device drivers
- 25-pole RS-232C connecting cable with additional 25 to 9-pole adapter plug.

1.4 Typographic Conventions

Arial This script represents text which the user can enter or that which is displayed on the screen of the PC or the display of the device.

[Command Button] This type indicates a command button in a program window which should be pressed, e. g., **[OK]**.

[Key] This type indicates a key on the keyboard which should be pressed, e. g., **[Esc]**.

[Key 1][Key 2] This type indicates a keystroke for a command. It means that you should push **[key 1]** and **[key 2]** at the same time, e. g., **[Alt][X]**.

Menu/Selection This type represents menu commands. Instead of “select the command **Save** from the **File** menu” the formulation “select **File/Save**” is used.

Name This type represents the name of a window or box.

1.1 Specification

Contents	1	CD-ROM
	1	Operating instructions

2 Introduction

1.5 Program Contents

- General
- Soft Workshop is a software package for creating PC-based measuring technique applications, preferably with measuring devices from the Grundig **digimess**[®] series. Soft Workshop uses the software sets provided with each **digimess**[®] device and expands these with functions for:
- Data storage
 - Creating automatic measuring procedures
 - Presenting and recording automatic measuring procedures

1.6 Installing and Starting the Program

- Attention
- The Soft Workshop can only be installed after Microsoft Windows 95 has been installed on your computer. For further information about installing Windows refer to your Windows documentation.

- Installing the Soft WS
1. Start up Windows.
 2. Insert the CD-ROM in the drive.
 3. In Windows 95 select the command **Start\Run** and enter the following command line:
 - **disk1/deutsch/setup**, to start the German version
 - **disk1/englisch/setup**, to start the English version
 4. Click on **[OK]** and follow the set-up instructions on the screen.
 - The program executes the complete installation in a folder specified by the user (default **c:\program\digimess**).

- Starting the Soft WS
- After successful installation, the folder **digimess** with the program **digimess** is located in the menu **Start/Programs**.
5. Start up the program **digimess**.
 - The task bar, from which all actions in Soft WS are started, is opened.

1.7 Task Bar

Set-up After starting the program the task bar appears and displays the following elements:



List box Displays the activated and configured **digimess**[®] devices.

[System] Activates reading in and enabling of the selected device driver (see 2.1).

[Devices] According to the measuring task, a software device pool can be compiled.
The device drivers implemented with **[System]** can be configured according to the wishes of the client (see 2.2).

[Start Device] Starts the marked device, with all its known functions, from the Software Set. In addition, visualisation of the measuring data can be activated via the *Extended Menu* (see 3.2).

[SQZ] Opens the Smart Sequencer. The functions of this integrated software package enable automation of the measuring tasks, in particular procedure control, data visualisation and measuring value logging (see 5).

[?] Is not supported at present.

[↖?] Context-sensitive help
Opens a help text window. Choosing any element on the program window with the mouse displays a brief description of that element.

[Exit] Soft WS is closed.

1.8 Mouse Procedure

Introduction The following actions may be carried out on-screen with the mouse:

- Select commands
- Click on command buttons
- Mark text

Left Mouse Button Unless otherwise instructed, use the left mouse button.

Right Mouse Button Using the right mouse button displays some special menus. You will be instructed when you need to use the right mouse button.

Mouse Functions	Instruction	Action
	point	point to an object with the mouse pointer
	click	point to an object and press and quickly release the left mouse button once .
	double click	point to an object and press and release the left mouse button twice in rapid succession.

1.9 Command Selection

Introduction	Commands are instructions to the Soft Workshop. Commands can be activated using the keyboard or the mouse via menu options, or command buttons.
Keyboard Operation	<ol style="list-style-type: none"> 1. Push [Tab] a number of times until the required object (menu list or command buttons) is marked by a transparent frame. 2. Push [Enter]. <ul style="list-style-type: none"> – A pull-down menu is opened and you can call up a menu point by entering its initial. – If a dialogue window is opened by a menu option or a command buttons, select the corresponding fields with [Tab] and confirm the selection with [Enter].
Mouse Operation	<ol style="list-style-type: none"> 1. Click on the required object (menu list or command button) <ul style="list-style-type: none"> – The command button results in an immediate action. – In the menu list, the corresponding pull-down menu is opened. 2. Click on the required menu option in the menu list. <ul style="list-style-type: none"> – If a dialogue window is opened by a menu option or a command button, select the corresponding fields with a mouse click.

2 Registering the *digimess*[®] Device

Introduction In order to use the Soft WS to create PC-based measuring technique applications, the *digimess*[®] device must be enabled and configured. For this, you will need the device driver from the Software Set.

2.1 Enabling the Device Drivers

Introduction

1. Insert the driver diskette from the Software Set of your *digimess*[®] device in drive A (B).
2. In the task bar click on **[System]**.
 - The window *Enable Device Types* is opened:



- In the list box *Allowed Device Types* the current *digimess*[®] devices which are supported by Soft WS appear.

3. Click on **[Add]**.
 - The window *Enable Device Types Dialog* is opened:



4. Enter the file name, with complete path, of the driver in the *Drive/Directory* list box or search for the required file from your directory using **[Browse]**.

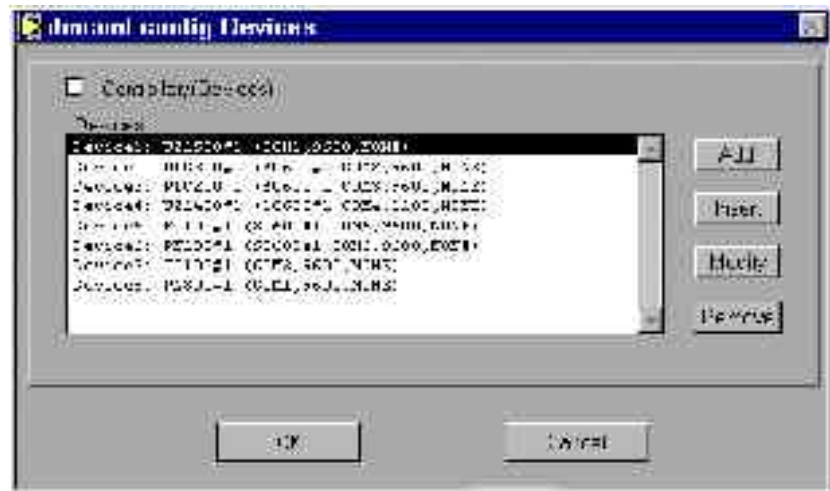
5. Click on **[OK]**.
 - The window *Enable Device Types Dialog* is closed.

6. In the window *Enable Device Types* click on **[OK]**.
 - The new driver is read in and activated.

2.2 Configuring the *digimess*[®] Devices

Starting the
Configuration

1. On the task bar click on **[Devices]**.
 - The window *config Devices* is opened:



- In the list box *Devices* the *digimess*[®] devices appear with the following interface parameters:
 - Control with controller (e. g.: **SC600#1**)
 - Interface address (e. g.: **COM1**)
 - Baud rate (e. g.: **9600**)
 - Transmission protocol (e. g.: **NONE**)
2. Click in the small box *Controller/(Devices)*, to change to the list box *Controllers*.
 - The small box *Controller/(Devices)* is marked with a cross.
 - In the list box *Controllers* the registered *digimess*[®] controllers SC 600 appear with the following interface parameters:
 - Interface address (e. g.: **COM1**)
 - Baud rate (e. g.: **9600**)
 - Transmission protocol (e. g.: **NONE**)

2.2.1 Device Settings

Adding the device

1. Click in the small box *Controller/(Devices)*, to change to the list box *Devices*.
2. Click on **[Add]**.
 - The window *Config Device Dialog* is opened:



3. In the list box *Device Type* click on the device you wish to add.
 - The list line is marked black.
4. Click on **[▲]** or **[▼]** on the left of the entry field:
 - *Address*, to change the interface address (default **0**).
 - *Baud Rate*, to change the transmission rate (default **9600**).
 - *Protocol*, to change the transmission protocol (default **NONE**).
5. In the list box *Controllers* click on:
 - **NONE**, to control the device **without** controller
 - **SC600**, to control the device **with** controller
 - The list line is marked black.
6. Click on **[OK]**.
 - The window *Config Device Dialog* is closed. The newly configured device is added to the device list.

Inserting the Device

1. In the list box *Devices* click on the device before which the newly configured device is to be inserted.
 - The list line is marked black.
2. Click on **[Insert]**.
 - The window *Config Device Dialog* is opened.
3. Set the interface parameters and the interface control of the device (see above: Adding the Device)
4. Click on **[OK]**.
 - The window *Config Device Dialog* is closed. The newly configured device is inserted before the marked line on the device list.

Modifying the Device

1. In the list box *Devices* click on the device to be modified.
 - The list line is marked black.
2. Click on **[Modify]**.
 - The window *Config Device Dialog* is opened.
3. Set the interface parameters and the interface control of the device (see above: Adding the Device)
4. Click on **[OK]**.
 - The window *Config Device Dialog* is closed. The marked device is displayed with the new interface parameters in the device list.

Removing the Device

1. In the list box *Devices* click on the device which is to be removed.
 - The list line is marked black.
2. Click on **[Remove]**.
 - The device is removed from the list box.

2.2.2 Controller Settings

Adding the controller

1. Click in the small box *Controller/(Devices)*, to change to the list box *Controllers*.
2. Click on **[Add]**.
 - The window *Config Controller Dialog* is opened:



- At present only controllers can be added by the type of SC 600.
3. Click on **[▲]** or **[▼]** on the left of the entry field:
 - *Address*, to change the interface address (default **0**).
 - *Baud Rate*, to change the transmission rate (default **9600**).
 - *Protocol*, to change the transmission protocol (default **NONE**).
 4. Click on **[OK]**.
 - The window *Config Controller Dialog* is closed. The newly configured controller is added to the device list.

Inserting the
Controller

1. In the list box *Controllers* click on the controller before which the newly configured controller is to be inserted.
 - The list line is marked black.
2. Click on **[Insert]**.
 - The window *Config Controllers Dialog* is opened.
3. Set the interface parameters of the controller (see above: Adding the Controller)
4. Click on **[OK]**.
 - The window *Config Controller Dialog* is closed. The newly configured controller is inserted before the marked line on the device list.

Modifying the
Device

1. In the list box *Controllers* click on the device to be modified.
 - The list line is marked black.
2. Click on **[Modify]**.
 - The window *Config Controllers Dialog* is opened.
3. Set the interface parameters of the controller (see above: Adding the Controller)
4. Click on **[OK]**.
 - The window *Config Controller Dialog* is closed. The newly configured controller is inserted before the marked line on the device list.

Removing the
Device

1. In the list box *Controller* click on the device which is to be removed.
 - The list line is marked black.
2. Click on **[Remove]**.
 - The controller is removed from the list box.

3 Starting the *digimess*[®] Devices

3.1 Operation of the *digimess*[®] Devices



Attention Calling up the device driver via Soft Workshop, as opposed to individual operation of the Software Sets, results in the following changes for the user:

- Interface parameters **Com-Port**, **Baud Rate** and **Protocol** are not queried. These parameters are set as described in 2.2.
 - Closing the driver via **[ON/OFF]** ends the driver program without further confirmation request.
-

Starting the Device

1. On the task bar in the list box click on the device you wish to start up.
 - The list line is marked black.
2. In the task bar click on **[Start Device]**.
 - The driver is activated and the software and hardware are initialised. The corresponding virtual device interface is then opened, e. g.: UZ 2500:



Depending on your PC configuration, this process can take some time (up to 2 min).

- After successful start, device-specific parameters appear on the display of the virtual device interface and the system is ready for operation.

Initialising Errors

If errors occur during start-up the message **Init Error!** appears on the display of the virtual device interface.

- Check:
 - Switching on state of the device
 - Hardware connections
 - Interface address etc.
- Remove the error and click on the inserted command button **[ERROR]**.
- Click on **[INIT]**, to initialise the virtual instrument once more.
- If the virtual instrument makes nothing, click in the task bar on **[Exit]**, to finish the Soft WS.

Operating the Device

3. To operate the *digimess*[®] device, refer to the accompanying operating instructions of the Software Set.

Extended
Functionality

4. On the virtual device interface click on **[EXT]** to open the *Extended Menu* (see 3.2).

Switching off
the Device

5. On the virtual device interface click on **[ON/OFF]** to switch off the device.
– The virtual instrument is closed and the device drivers are deactivated.

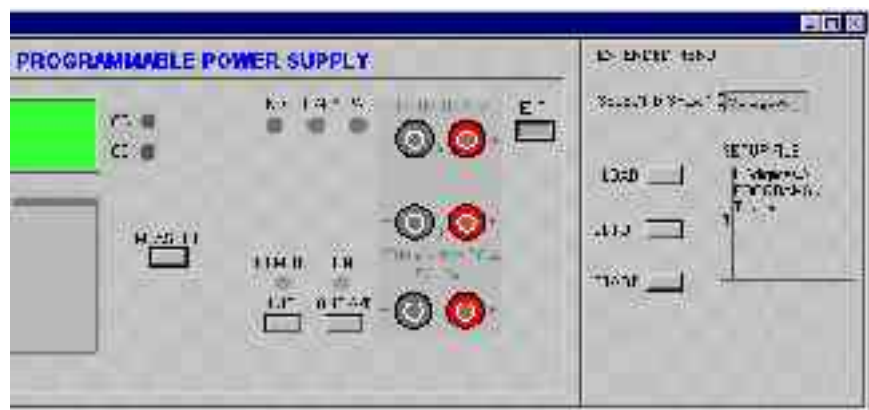
3.2 Extended Menu – Extended Functionality

Introduction

The *Extended Menu* is used to parameterise and to start up the **digimess**[®] display. The measuring data of the installed measuring devices can be visualised and saved with the help of the **digimess**[®] displays.

Opening/Closing the
Extended Menu

1. Start the desired device as virtual instrument (see 3.1).
2. On the virtual device interface click on **[EXT]** to open the *Extended Menu*, e. g. PN 300:



– The virtual device interface is extended to the right with the following elements:

- **[LOAD]** - Loading the **digimess**[®] display settings
- **[SETUP]** - Parameterizing the **digimess**[®] display
- **[START]** - Starting the **digimess**[®] display
- **SETUP FILE** - Path and file name

– Since for every device only a measuring parameter can be represented in the **digimess**[®] display, a further options display appears at device with several measuring parameters, e. g. PN 300:

- **SELECT DISPLAY** - Selecting the parameter to be displayed

3. Parameterizing and starting the **digimess**[®] display (see 4).

4. Click repeated on **[EXT]** to close the *Extended Menu*.

4 digimess® Display – Visualising and Saving the Measuring Data

Introduction

You can view measuring data of the installed devices on the **digimess®** display. When active, these devices supply measuring values which reach the display field via an internal memory. The values can be displayed in graphic or in tabular form.

The **digimess®** display has two modes of operation:

- **SETUP** – Parameterisation of display settings
- **START** – Display of measuring values

For each device, limit values **Limits** can be entered in the SETUP phase. These limits act as measuring value controls in the START phase.

Graphs can be scaled by the functions **Offset** and **Scale**. This allows the user to display different measuring signals in one graph.

The resulting measuring values can be written to a file **Logfile** and further processed.

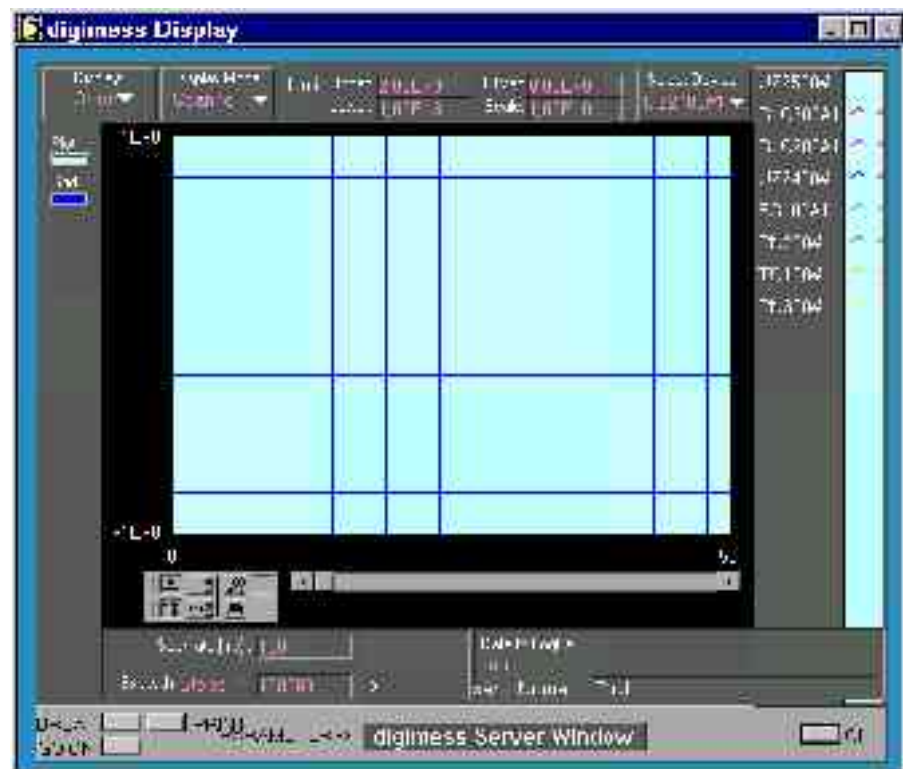
The determined measuring values are not changed by the scaling and can be written and reprocessed in the file **Logfile**. The takeover into spreadsheet programs, e. g. Excel, is straightforwardly possible.

4.1 User Interface of the digimess® Display

Set-up

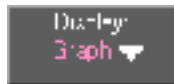
The user interface of the **digimess®** display consists of the following components:

- **Entry Field** for parameterisation
- **Display Window** with legend and palette
- **Status Line** with control functions

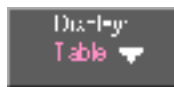


4.1.1 Entry Field

[Display:]

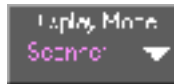


The measuring values are displayed graphically and can be inspected visually.

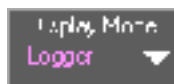


The measuring values are displayed numerically in tabular form. These tables can be stored in the file (Logfile).

[Display Mode]

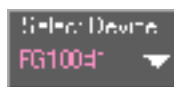


Continual transfer of the measuring values from the measuring value memory. The time interval is defined in the entry window *Scanrate*, e.g.:



Synchronised transfer of current measuring values from the measuring value memory. Only new measuring values are displayed. Inactive devices are not displayed.

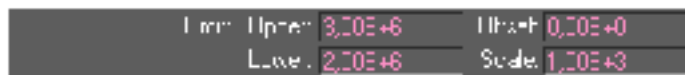
[Select Device:]



Selection of a configured measuring device. Special settings: *Limit*, *Offset* and *Scale* can thus be carried out.

Limit:

- **Upper:** – Entry of an upper limit value
- **Lower:** – Entry of a lower limit value



By comparing the current measuring values with the upper limit **Upper** and the lower limit **Lower**, a good-bad statement is possible. The setting must be device-specific.

Offset:

Entry of an offset value to adjust the measuring values in a graphic display.

Scale:

Entry of a scaling value to adjust the measuring values in a graphic display.

The determined measuring values are not changed by the scaling.

Scanrate:

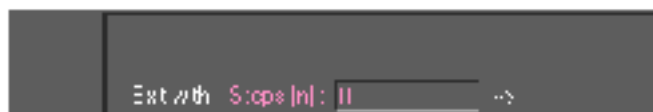
Entry of a time interval during continual transfer of the current measuring values from the measuring value memory (see above: **[Display Mode]**)



Exit with

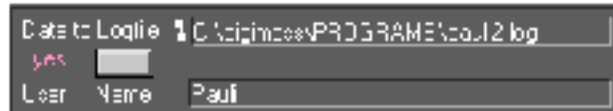
Determining the exit conditions of the **digimes**[®] display by:

- **Steps [n]:** – Maximum number of transferred measuring values
- **Time [s]:** – Time-limited exit condition
- **Ok:** – No automatic exit, exit with **[OK]**



Data to Logfile:

Determining the measuring data save



- **no** – No data save
- **yes:** – The resulting measuring values are written to a file and can be further processed.

4.1.2 Display Window

Introduction

Depending on the type of display (**Graph/Table**) and the data triggering (**Scanner/Logger**) of the measuring values, different display possibilities arise with the corresponding control functions.

4.1.2.1 Scanner (Multiplot Charts)

Setting

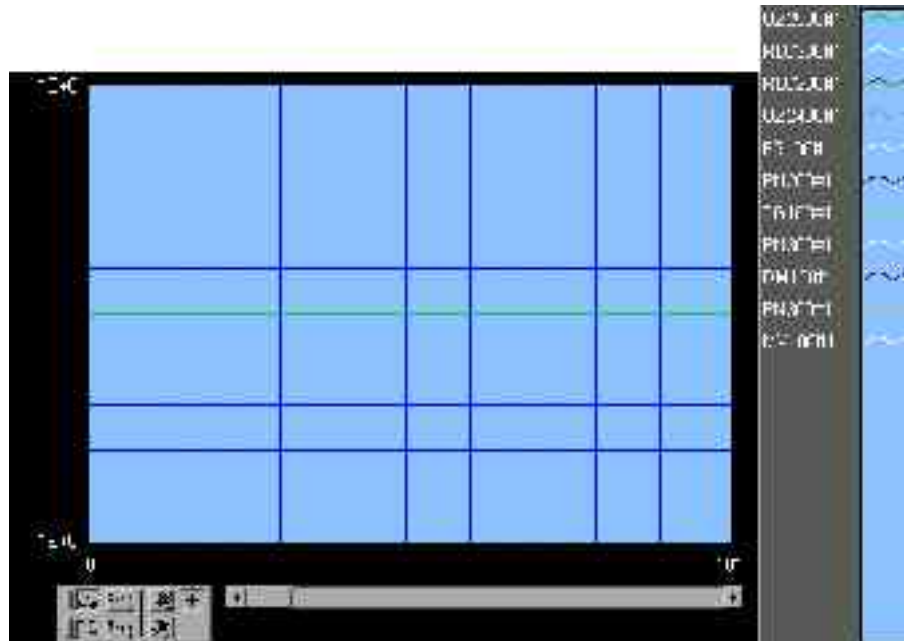
To activate the graphic scanner display the following settings must be selected:

1. Click on **[Display]** and then on **Graph**.
2. Click on **[Display Mode]** and then on **Scanner**.

Window Contents

Charts can hold several curves. The graphic scanner-display consists of:

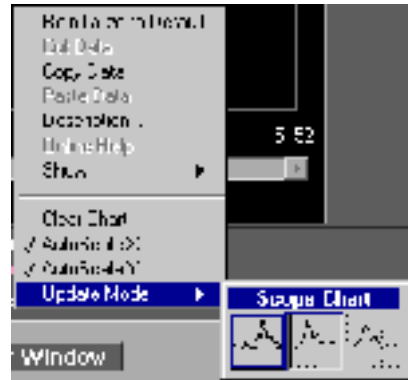
- **Display window** with selected display form of the charts
- **Legend** for formatting measuring value curves of the installed devices
- **Palette** for axis scaling and zoom function
- **Scroll bars** for scrolling along the X axis



Display Forms for Charts

There are three possible display forms for the charts, the forms are available by clicking on the right mouse button on the display and on the option **Update Mode**:

- **Strip Chart** - Strip chart recorder
- **Scope Chart** - Oscilloscope diagram
- **Sweep Chart**

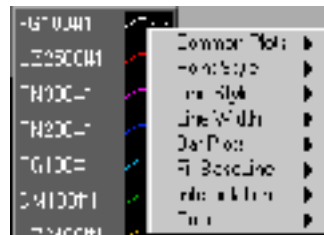


Standard mode is strip chart recorder.

- The scroll bar display of the **Strip Chart Recorder Mode** is similar to a strip chart recorder with ticker tape. If the monitor receives a new value, it displays the value on the right and pushes the old value over to the left.
- The **Oscilloscope Diagram Mode** has a display similar to an oscilloscope. If the device (virtual instrument) receives a new value, it displays the value as a graph on the right beside the last value. If the plot reaches the right of the display range, the virtual instrument clears the plot and begins again with the graphic display on the left-hand side. The oscilloscope diagram is considerably faster than the strip chart recorder since it is not slowed down by the scrolling process.
- The **Sweep Chart Mode** is very similar to the oscilloscope Diagram, but is however, not cleared if the data reach the right-hand side. Instead a moving vertical line marks the beginning of the new data and it moves over the display while the virtual instrument adds new data.

The Legend

The legend is on the right of the graphic window and describes the display form (Point, Line, Plot, Colour, ...) and the terms of the curves.



Clicking the mouse on the corresponding device allows for different settings to be changed manually.

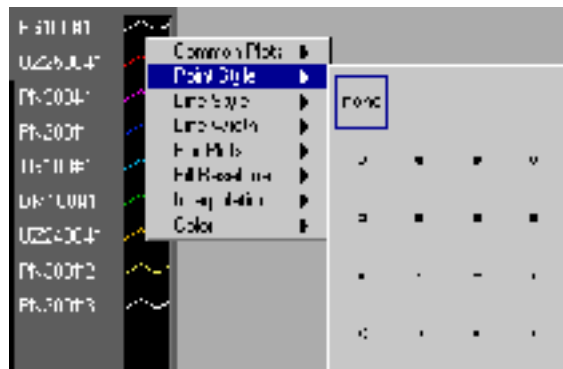
- **Common Plots**

Graphic display of the curve shape



- **Point Style**

Point style of a measuring value for the selected curve



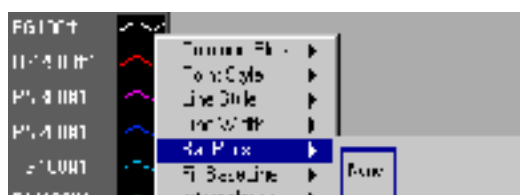
- **Line Style/Line Width**

Line style and width for combined curve features



- **Bar Plots**

Displaying measuring points as bar charts



Axis Scaling and Zoom Function

- Menu points for scaling and formatting the X and Y axes are called up via the command buttons **[x.xx]** and **[y.yy]**:
 - **Format** - Axis Format
 - **Precision** - Display Precision
 - **Mapping Mode** - Display Form



- Autoscaling of the X and Y axes is set with the blocking switches on the left side of the palette and the zoom function is deactivated.
- If autoscaling is switched off, any window section can be viewed using zoom functions.



4.1.2.2 Logger (Multiplot Graph)

Setting

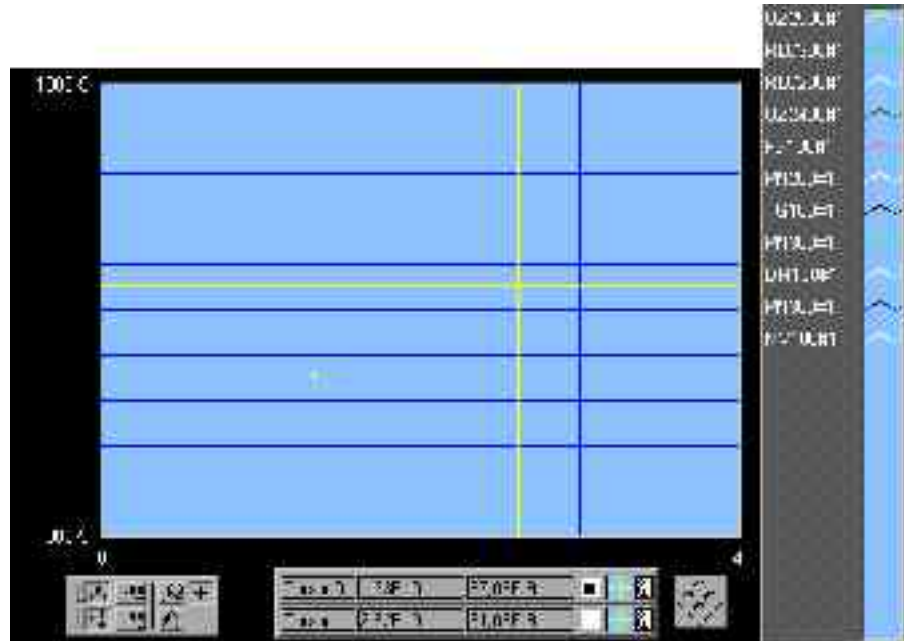
To activate the graphic scanner display, the following settings must be selected:

1. Click on **[Display]** and then on **Graph**.
2. Click on **[Display Mode]** and then on **Logger**.

Window Contents

Graphs can hold several curves with a different number of measuring values. The graphic logger display consists of:

- **Display window**
- **Legend** for formatting measuring value curves of the installed devices, see 4.1.2.1
- **Palette** for axis scaling and zoom function, see 4.1.2.1
- **Cursor Menu** for measuring data evaluation



Cursor Menu

- It is possible to measure the curves or order them in corresponding categories with the cursor menu.
- The cursors are moved within the graphic window with the help of the mouse.
- The cursors can be changed within the graphic window in the form of (**Cursor Style**, **Point Style**), color (**Color**) and function. The function **Bring to Center** places the cursor in the visible window section and the function **Go to Cursor** moves the picture section to the current cursor position.

With a mouse click into the 1. or 2. cursor menu line the corresponding cursor options are called.



4.1.2.3 Scanner & Logger (Table)

Setting To activate the tabular display, click on **[Display]** and then on **Table**.

Window Contents The table displays the measuring values from all the installed devices separated by the **Scanrate**. The table header can be edited and is saved together with the **digimess®** display settings.

Device	No.	Time	Value	Limit
PH0001	0002	5:29:04		Under-Lim:
PH0002	0002	5:29:04		Over-Lim:
PH0002	0002	5:29:04		Over-Lim:
PH0001	0002	5:29:04		Over-Lim:
PH0001	0002	5:29:04		Over-Lim:
PH0001	0002	5:29:04		Over-Lim:
PH0001	0002	5:29:04		Over-Lim:
PH0001	0002	5:29:04		Under-Lim:
PH0001	0002	5:29:04		Under-Lim:
PH0002	0002	5:29:04		Over-Lim:
PH0002	0002	5:29:04		Over-Lim:
PH0001	0002	5:29:04		Over-Lim:
PH0001	0002	5:29:04		Over-Lim:
PH0001	0002	5:29:04		Over-Lim:
PH0001	0002	5:29:04		Under-Lim:
PH0001	0002	5:29:04		Under-Lim:

4.1.3 Status Line

Contents The status line displays the current state of operation:

- **PARAMETER** – Parameterisation of the display settings
- **RUN** – Display of the measuring values

and contains command buttons for operating the **digimess®** display.

[BREAK] Transfer of the current measuring values from the measuring value memory is interrupted.

[GO ON] Transfer of a current measuring value from the measuring value memory if **[BREAK]** is activated.

[PRINT] The current contents of the **digimess®** display are printed. The print settings, landscape or portrait format, should be set in the system printer. The size will be sized to fit.

[OK] The **digimess®** display is switched off. In the operation mode **PARAMETER** a request to save the settings appears.

4.1.4 SETUP – Parameterisation of the *digimess*[®] Display Settings

Introduction	During the SETUP phase, indicated by PARAMETER in the status line, all red coloured text can be changed. The <i>digimess</i> [®] display can thus be formatted and parameterised.
Starting the SETUP Phase	<ol style="list-style-type: none">1. Start a virtual instrument and click on [Measure] in the virtual device interface (see 3.1).2. In <i>Extended Menu</i> click on [SETUP] (see 3.2).<ul style="list-style-type: none">– The <i>digimess</i>[®] display is started up with a default setting.
Selecting the Presentation Form	<ol style="list-style-type: none">3. Click on [Display ▼] and then on:<ul style="list-style-type: none">▪ Graph, to set the graphic display of the measuring values.▪ Table, to set the tabular display of the measuring values.
Selecting the Operation Mode	<ol style="list-style-type: none">4. Click on [Display Mode ▼] and then on:<ul style="list-style-type: none">▪ Scanner, to set the continual transfer of the current measuring values▪ Logger, to set the synchronised transfer of a new measuring value5. When you have selected the operation mode Scanner, enter a time interval into the entry field <i>Scanrate</i>.
Entering the Device-Specific Display Parameters	<ol style="list-style-type: none">6. Click on [Select Device ▼] and then on a device to carry out the following settings:<ul style="list-style-type: none">▪ Limit Upper: - Entry of an upper limit value▪ Limit Lower: - Entry of a lower limit value▪ Offset: - Entry of an offset value▪ Scale: - Entry of a scale value
Determining the Exit Conditions	<ol style="list-style-type: none">7. Click on the red coloured text in front of the entry field <i>Exit with ...;</i>, to determine one of the following exit conditions of the <i>digimess</i>[®] display:<ul style="list-style-type: none">▪ Steps [n]: - maximum number of resulting measuring values▪ Time [s]: - time-limited exit condition▪ Ok: - no automatic exit, exit with [OK]
Determining the Measuring Data Save	<ol style="list-style-type: none">8. Click on the red coloured text below the entry field <i>Data to Log-file;</i>, to set the measuring data save:<ul style="list-style-type: none">▪ no - No data save▪ yes - Data save in a file (see 4.1.4.1)
Formatting and Scaling the Display	<ol style="list-style-type: none">9. In the Legend, click on a device to format the view of the measuring value curve (see 4.1.2.1).10. Click on [Plot], to set the background colour of the display window.11. Click on [Grid], to confirm the grid colour of the display window.12. Click on [x.xx] and [y.yy], to format the axes (see 4.1.2.1).13. Click on the blocking switch on the left of the Palette, to set auto-scaling of the X and Y axes.<ul style="list-style-type: none">– When autoscaling is active the display area on the right beside the blocking switches look engraved.

Setting the Display
Forms for Charts

14. If you have chosen the operation mode **Scanner**, click first on **Update Mode** with the right mouse button on the display window and then on the opened pop-up menu to set one of the following display forms for the charts (see 4.1.2.1):
 - **Strip Chart** - Strip chart recorder
 - **Scope Chart** - Oscilloscope diagram
 - **Sweep Chart**

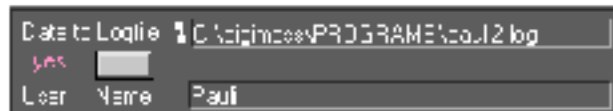
Ending the **digimess**[®]
Display Settings

15. Click on **[OK]**, to end parameterisation.
 - The window *Choose file to write.* is opened and you are requested to enter a name.
16. Click on:
 - **[Save]**, to save the settings
 - **[Cancel]**, to end the SETUP phase without saving the current settings

4.1.4.1 Data to Logfile – Measuring Data Saving

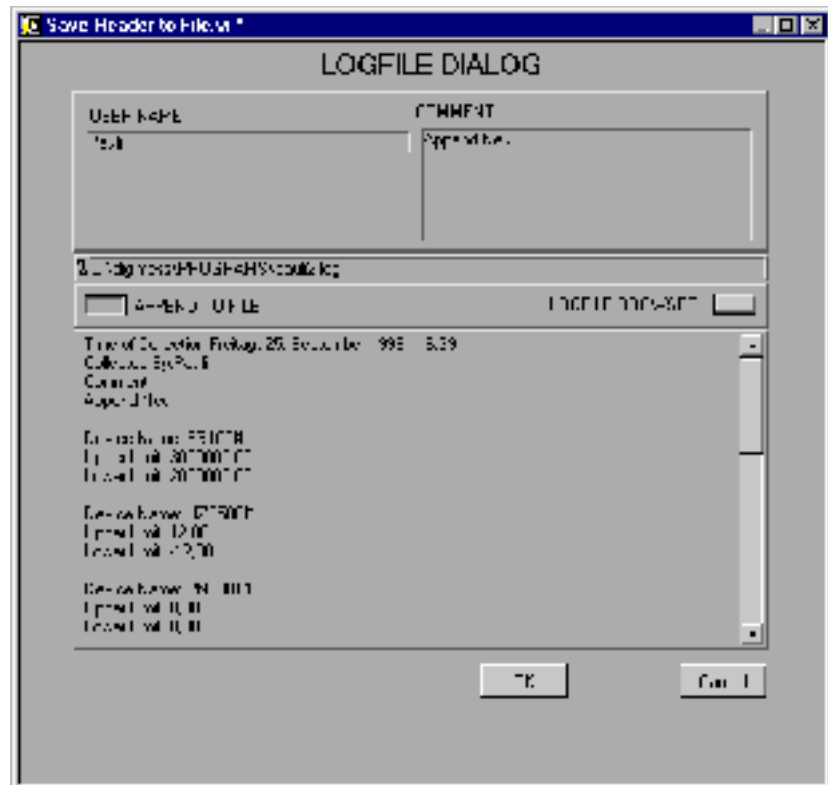
Activating the
Measuring Data Save

1. In the SETUP phase, click on the red coloured text below the entry field *Data to Logfile:* and then on **yes**, to set measuring data save.
 - A command button is inserted on the right of the text **yes**.



Configuring the Logfile

2. Click on the command button on the right of the text **yes**, to configure the Logfile.
 - The dialogue window *Save Header to File* is opened with the following pre-settings:
 - **USER NAME** - User entry
 - **COMMENT** - User Comment
 - **[APPEND TO FILE]** - Append to an existing file



3. Enter the user name (*USER NAME*) and a comment (*COMMENT*).
4. Select the type of data extension:
 - **[APPEND TO FILE]** - Append to an existing file
 - **[NEW FILE]** - Create a new file
5. Click on **[LOGFILE BROWSER]**, select a before created directory and enter a file name.
 - The current path and file name are displayed.
6. Click on:
 - **[OK]**, to save the settings
 - **[Cancel]**, to exit the dialogue window *Save Header to File* without making changes

Ending the Configuration



Attention To be able to save measuring data in the configured Logfile, complete the SETUP phase and call up the START phase (see 4.1.5).

4.1.5 START – Measuring Values Display

Introduction	<p>During the START phase, indicated by RUN in the status line, the saved parameters are loaded in order to format the digimess[®] display. In this mode of operation you can only switch between tabular and graphic displays. By selecting the corresponding device the limit value display is switched over and the limit value analysis can be viewed.</p>
Starting the digimess [®] Display	<ol style="list-style-type: none">1. Start a virtual instrument and click on [Measure] in the virtual device interface (see 3.1)2. Open the <i>Extended Menu</i> and click on [LOAD], to load a saved digimess[®] display setting. (see 3.2).<ul style="list-style-type: none">– The window <i>Select a Parameter File</i> is opened and you are requested to select a *.set-file and to confirm with [Save].– If you have not yet saved a digimess[®] display setting, click on [SETUP] in the <i>Extended Menu</i> to parameterise the digimess[®] display (see 4.1.4).3. Click on [START], to start the digimess[®] display with the selected digimess[®] display settings.<ul style="list-style-type: none">– The measuring values of the installed devices are presented in the display window.
Selecting the Presentation Mode	<ul style="list-style-type: none">• Click on [Display ▼] and then on:<ul style="list-style-type: none">▪ Graph, to present the measuring values in graphic form▪ Table, to present the measuring values in tabular form
Analysing the Limit Values	<ul style="list-style-type: none">• Click on [Select Device ▼] and then on a device to display the device-specific analysis of the limit value.
Scaling and Formatting the Display	<ol style="list-style-type: none">1. In the Legend click on a device to reformat the view of the measuring value curve (see 4.1.2.1).2. Click on [Plot], to reset the background colour of the display window.3. Click on [Grid], to reset the grid colour of the display window.4. Click on [x.xx] and [y.yy], to reformat the axes (see 4.1.2.1).5. Click on the blocking switches on the left of the Palette, to switch autoscaling of the X and Y axes on and off.<ul style="list-style-type: none">– when using autoscaling, all values are automatically adjusted to fit the graph window.– The active autoscaling is the recommended operating mode.
Setting the Display Forms for Charts	<ul style="list-style-type: none">• If the operation mode Scanner is active, with the right mouse button click on the display window and then in the opened pop-up menu click on Update Mode to set the following display modes for the charts (see 4.1.2.1):<ul style="list-style-type: none">▪ Strip Chart - Stripe recorder▪ Scope Chart - Oscilloscope diagram▪ Sweep Chart

Using the
Zoom Functions

- Use the zoom function for a detailed view of the curves (see 4.1.2.1).
 - When autoscaling the X and Y axes, the zoom function is inactive.

Using the
Cursor Functions

1. Click on **[BREAK]**, to break the continual transfer of the current measuring values.
2. Use the cursor functions for a detailed assessment of the curves (see 4.1.2.2).
3. Click on **[BREAK]** once more, to start the continual transfer of the current measuring values.

Executing the
singles measure-
ments manually taken
down

1. Click on **[BREAK]**, to break the continual transfer of the current measuring values.
2. Read the current measuring values from the virtual instrument and click on **[GO ON]**, to update an individual measuring value.
 - The specific transfer of a current measuring value is used to bridge undefined states at the measuring clamps. These can occur during the warming-up period of the measuring objects or at the time of contact.

Printing the Current
digimess[®] Display

- Click on **[PRINT]**, to print the current contents of the **digimess**[®] display.

Ending the
digimess[®] Display

- Click on **[OK]**, to end the **digimess**[®] Display.

5 Smart Sequencer – Creating the Test Program

5.1 General

Application	The Smart Sequencer is an efficient software tool, which simplifies the creation of application programs for PC-controlled measuring and test systems.
Program Philosophy	<p>In contrast to conventional programming, the program setting does not occur line-wise in a high level language or by generating a structogram.</p> <p>Programming the system is reduced to the compilation of pre-set test routines, so-called macros, which can be called up from a library and combined to create a test routine. They are adapted to suit specific measuring tasks by a simple parameterisation as such each macro has its own interface.</p>



Attention The user must not make any alterations to the directory structure otherwise malfunctions will occur within the software, for example by preventing the Smart Sequencer from starting.

5.2 User Interface of the Smart Sequencer

Set-up The user interface set-up is Windows-based and understood by most PC users. The user interface has three components:

- **Menu list**
- **Action window** with list boxes and command buttons
- **Status line**



5.2.1 Menu List

Task All menus and commands can be activated via the menu list.

Menu Reference

File	Edit	Settings	Extras
New	Cut	Show comment	Debugger
Load	Copy	Show library	Continuous run
Delete	Insert		Print test steps
Save	Add		Load protocol
Save as ...	Test file		
Exit			

5.2.2 Action Window

Set-up

The action window contains:

- The list box *Current Test Program*:
- The list box *Macro Library:/Comments*:
- Interfaces for fast and comfortable operation of the Smart Sequencer

Macro Library

The available functionality of the test system is presented as a library, the individual functions are shown as macros (test steps).

Current Test Program

A test procedure (test program) is the combination of required individual functions (test steps), which are then linearly processed.

Comfortable Editing Functions

Macro selection, inclusion in the test program and parameterisation are supported by extensive and comfortable editing functions. Using the clip board you can insert, paste, copy or cut, in lines or in blocks.

Comment

In order to understand the test procedures more thoroughly, comment can be entered into the parameterisation interface of every test step. This is displayed in the user interface of the Smart Sequencer when the comment field is activated. (see 5.6.4).

5.2.3 Status Line

Set-up

The status line is located on the lower edge of the Smart Sequencer user interface. The status line has two fields.

Modification Display

The left field represents the so-called modification display. If you make changes to the loaded procedure or if a new procedure has not yet been saved, this is displayed as *modified*. You are requested to save the changes. After saving, the display is empty.

This field is also activated by calling up the editing functions or by opening the parameterizing interfaces of the macros. This occurs whether or not parameters have been modified.

Path Display

The right field contains the complete path of the current loaded test procedure. This field is empty if a procedure has not yet been loaded or if a new one is being created.

5.3 Macro Library

5.3.1 Macros

Introduction

Programming the test procedures is reduced to the compilation of pre-set test routines, so-called macros, which can be combined in one test program. They are adapted to suit specific measuring tasks by a simple parameterisation of the individual macros.

5.3.1.1 System Reset

Application

The macro **System Reset** is used to reset the **digimess**[®] system and executes the following actions:

- Load configuration file (digimess.ini)
- Reset the data memory of the available commands
- Reset the data memory of the commands to be carried out
- Reset the data memory of the measuring results

Macro Settings

1. In Smart Sequencer double click on the macro **System Reset** (see 5.6.1).
 - The parameterisation interface with the current macro settings is opened.:



- Settings cannot be made in the macro.

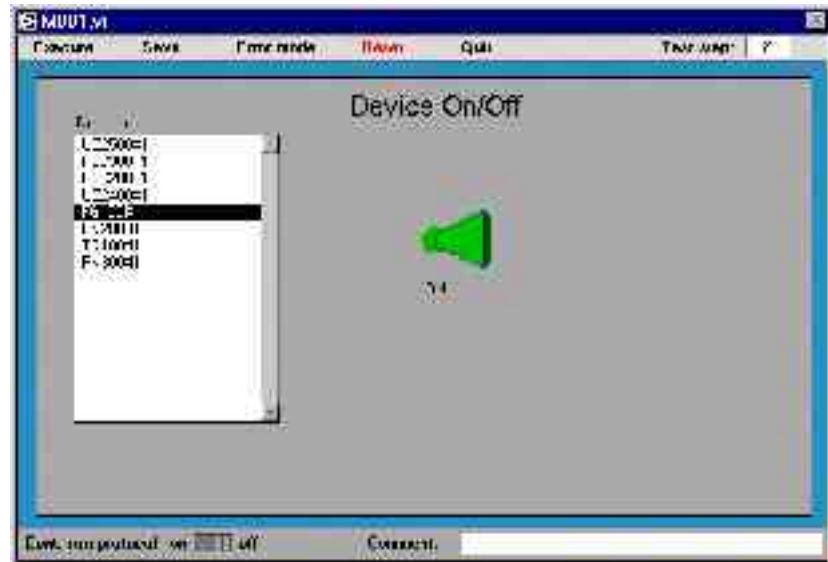
5.3.1.2 Device On/Off

Application

The macro **Device On/Off** is used for starting and ending a **digimess**[®] system device.

Macro Settings

1. In Smart Sequencer double click on the macro **Device On/Off** (see 5.6.1).
 - The parameter interface with the current macro settings is opened:



2. In the list box *Devices* mark the device which is to be switched on or off and click on the switch **OFF/ON**.

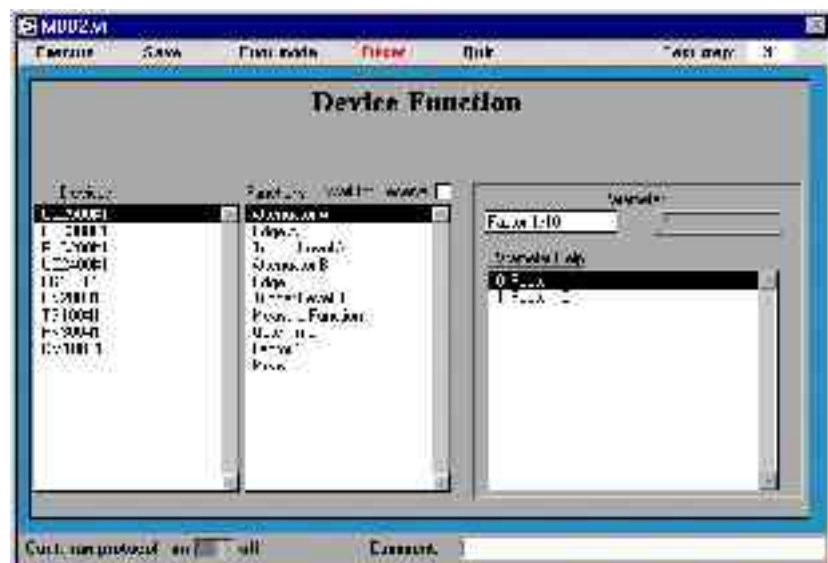
5.3.1.3 Device Function

Application

The macro **Device Function** is used for executing a specific function of a selected device.

Macro Settings

1. In Smart Sequencer double click on the macro **Device Function** (see 5.6.1).
 - The parameter interface with the current macro settings is opened:



2. In the list box *Devices* mark the device which is to be addressed.
 - Depending on the device, the contents of the list box *Functions* change. If the selected device has not been started, the list box remains empty.
3. In the list box *Functions* select the chosen function.
 - If the chosen function requires a parameter, the following appears in the right side of the window:
 - the parameter name, e. g., **Factor 1/10**
 - the parameter entry field, e. g., **1**
 - and the parameter help, e. g., **1: Factor1**.
4. Enter a valid parameter value into the parameter entry field or click on a line in the parameter help.
 - If you select a line in the parameter help (e. g., **1: Factor1**) the characters before the semi-colon (e. g., **1**) of the selected line are entered.

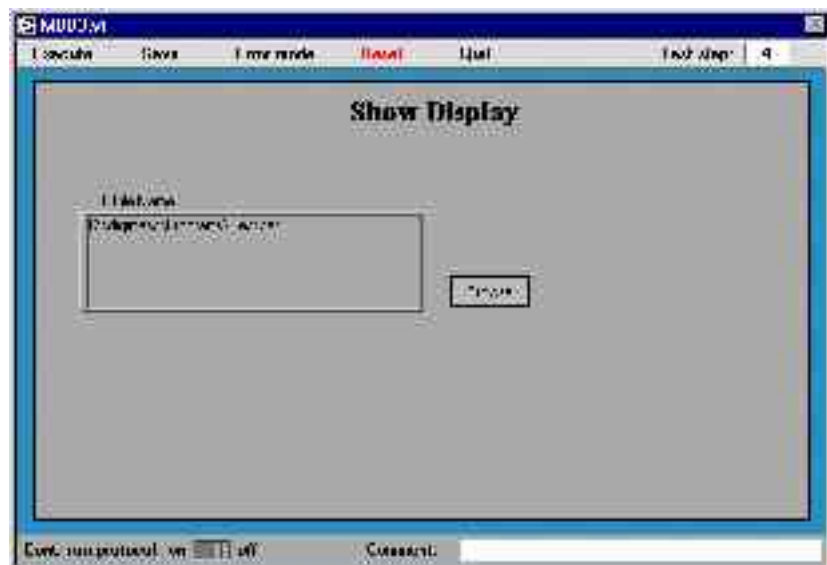
5.3.1.4 Show Display

Application

The macro **Show Display** is used to start the *digimess*[®] display with an pre-specified file which contains the current settings.

Macro Settings

1. In Smart Sequencer double click on the macro **Show Display** (see 5.6.1).
 - The parameterisation interface with the current macro settings is opened:



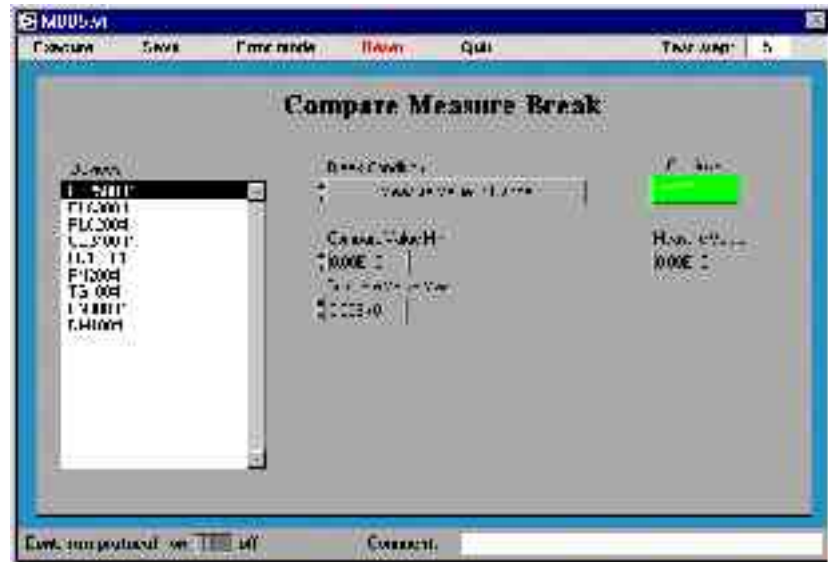
2. In the entry field *SET File Name* enter the file name, with complete path, of your chosen display settings or search for the required file in your directory using **[Browse]**.

5.3.1.5 Compare Measure Break

Application The macro offers the possibility to define different break conditions for the test course.

Macro Settings

1. In Smart Sequencer double click on the macro **Compare Measure Break** (see 5.6.1).
 - The parameterisation interface with the current macro settings is opened:



2. In the list box *Devices* mark the device which you wish to set for break conditions when measuring.
3. Click on [▲] or [▼] on the left of the entry field *Break Condition*, in order to set one of the following break conditions:
 - **Measure Value > Compare Value Min**
 - **Measure Value < Compare Value Min**
 - **Measure Value in Range**
 - **Measure Value out of Range**
4. Click on [▲] or [▼] on the left of the entry field *Compare Value Min*, to set the lower benchmark value or enter a value directly into the entry field.
5. When requested on the display click on [▲] or [▼] on the left of the entry field *Compare Value Max*, to set the upper benchmark value or enter a value directly into the entry field.
6. Click on **[Continue]** to restart the measurement which was stopped due to the break conditions.
 - On the display *Measure Value* the current measuring value appears.



Attention The valid value range for the benchmark values should be taken from the technical parameters of the selected device.

5.3.2 Special Macros

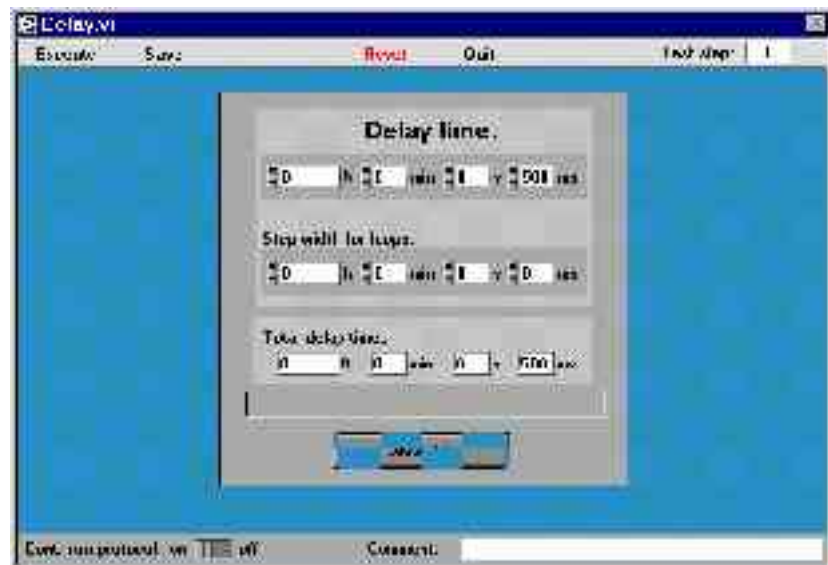
Introduction Special macros contain functions for creating test programs. These macros are necessary for structuring the test procedures.

5.3.2.1 DELAY Function

Application The macro offers the possibility to build delay times into the test procedures, e. g., to bridge the time between signal stimulus and the measuring procedure.

Macro Settings

1. In Smart Sequencer double click on the macro **DELAY** (see 5.6.1).
 - The parameterisation interface with the current macro settings is opened:



2. Click on [▲] or [▼] on the left of the entry field *Delay time*: to enter the base waiting time.
 - Execution of a test step (compared with the previous test step) is delayed for the delay time which has been entered.
3. Click on [▲] or [▼] on the left of the entry field *Step width for loops*: to enter a waiting time.
 - If the macro is part of a loop, the waiting time is increased with each loop run for the time entered.
 - On the display *Total delay time*: the resulting waiting time appears which consists of the base waiting time and the total of the loop runs which have taken place.
4. Click on **Cancel delay**, in order to break the current WAIT cycle while executing the macro.

5.3.2.2 LOOP Function

Application	The macro offers the possibility to generate loops within a test program. The number of program loops within a superior loop, is unlimited.
Inserting the Macros	If a macro LOOP is inserted into a test program, all successive macros in the interface of the smart sequencer are indented to the right. They are thus a component of the program loop. The macro sequence within a loop is broken by the macro ENDLOOP .
Macro Settings	<ol style="list-style-type: none">1. In Smart Sequencer double click on the macro LOOP (see 5.6.1).<ul style="list-style-type: none">– The parameterisation interface with the current macro settings is opened.2. Enter the number of loop runs in the entry field.



Attention The macro **ENDLOOP** has no parameters.

5.3.2.3 CALL Function

Application	The time needed for creating complex macro sequences can be reduced, by using existing test programs as sub-routines. The CALL function supports this. There can be any number of nested sub-routines i. e., within a sub-routine, further sub-routines can be called up.
-------------	-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------



Attention A superior program **cannot** be called from a sub-routine with the **CALL** function.

Macro Settings	<ol style="list-style-type: none">1. In Smart Sequencer double click on the macro CALL (see 5.6.1).<ul style="list-style-type: none">– The parameterisation interface with the current macro settings is opened:2. Enter the file name, with complete path, of the required sub-routine in the entry field or search for the required file in your directory using [Browse].
----------------	---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------

5.4 Beginning Work

- File New
- Select the command **File/New**, to open a new (empty) test program.
 - If a test program has already been loaded, changed and not saved, a request to save appears before opening the new (empty) program.
- File Load
- Select the command **File/Load**, to load a saved test program.
 - The desired test program can be searched for and opened via the window *Open*.
- File Delete
- Select the command **File/Delete**, to clear a saved test program.
 - The desired test program can be searched for and cleared via the window *Open*. For safety reasons you are requested if you wish to clear this test program.
- File Save
- Select the command **File/Save**, to save an edited test program under the old program name.
 - If the test program is new, you will be requested to enter a name via the window *Test Program Name*.
- File Save as
- Select the command **File/Save as ...**, to save a loaded or a newly edited test program under a new program name.
 - The window *Save as...* is opened and you are requested to enter a new name.
- File Exit
- Select the command **File/Exit**, to end the Smart Sequencer.
 - If a loaded test program is newly edited a prompt to save follows.
 - If the test program is new, you are prompted to enter a name via the window *Test Program Name*.

5.5 Creation of the Test Procedure

5.5.1 Edit Functions

- Cut
- Select the command **Edit/Cut**.
 - The marked test steps are cut and copied to the clip board. These test steps are now available for appending and pasting functions.
 - The function is only available in the list box *Current Test Program*:
- Copy
- Select the command **Edit/Copy**.
 - The marked test steps are copied to the clip board. These test steps are now available for appending and pasting functions.
 - The function is available in both list boxes.
- Insert
- Select the command **Edit/Insert**.
 - This function requires that the test steps are present in the clip board.
 - The contents of the clip board are inserted before the position which has been marked. When more than one object is selected, paste always takes place before the first marked position.
 - The function is only available in the list box *Current Test Program*:
- Add
- Select the command **Edit/Add**.
 - This function requires that at least one test step is marked in the list box or that test steps are present in the clip board.
 - The marked test steps are appended to the end of the current test program. The marked test steps are simultaneously copied to the clip board and are available to the **Insert** function.
 - If the contents of the clip board are to be appended to the current test program, you should no longer mark test steps. If this happens, the marked steps are appended and they overwrite the contents of the clip board.
- Calling up the Edit Functions
- Each edit function can be called up in three different ways:
- Entering the command **Edit/...**
 - Activating the command buttons [...]
 - Pushing the function keys [F...]

5.5.2 Marking the Test Steps

Introduction Test steps (macros) must be marked in order to insert them into a test procedure or to remove from a test procedure. Individual selection as well as multiple selection of test steps is possible.

Individual Selection

- Click on the macro you want to select.
 - The selected line is marked in black.

Multiple Selection

- Click on the macros you wish to select, while holding down the [Shift] button.
 - The selected lines are marked in black:



Clear Selection If a selected line is clicked on again with the mouse, while holding down the [Shift] key, it is no longer marked and thus no longer selected.



Attention Selecting steps in both list boxes simultaneously is not possible.

5.5.3 Edit Test Procedure

Program Creation

1. Open or load a test program (see 5.4)
2. Mark one or more test steps in the list box *Macro Library*: (see 5.5.2).
3. Insert test step(s) into the list box *Current Test Program*: (see 5.5.1).
4. Enter parameters for the test program (see 5.6).
5. Edit the test file (see 5.5.4).

5.5.4 Test File

Contents	A comment text, in which information can be stored about changes to the parameters and experience with the test object etc., accompanies each test procedure. Since this text is tied to the test program and contains information about it, it has been given the name <i>Test file</i> .
Edit Test File	<ol style="list-style-type: none">1. Select the command Edit/Test file.<ul style="list-style-type: none">– The window <i>Test File</i> with the comment text is opened:2. Edit the comment text using the keyboard.3. Conclude the entry with [OK] or click on [Cancel], to close the window without making any changes.

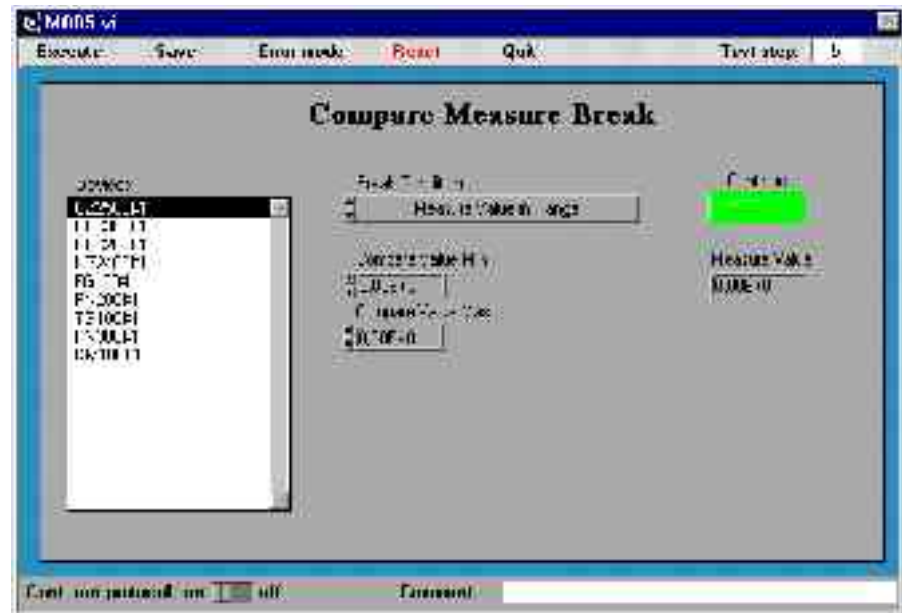
5.6 Parameterisation of the Test Steps (Macros)

5.6.1 General

Introduction	The test steps (macros) from the macro library are general templates. For correct test program run, the individual test steps must be parameterised according to the test objective.
Parameterisation of the Macro Library	To minimise the parameterisation when creating the test procedure, frequently selected device settings can be pre-set in the macro library. These then become the standard parameters for the test steps.
Parameterisation of the Test Procedure	Parameterisation of the test steps in the test program can be carried out at any time and has no influence on the test steps in the macro library.
Starting the Parameterisation	<ul style="list-style-type: none">• Double click on the test step you want to parameterise.<ul style="list-style-type: none">– The parameterisation interface with the current macro settings is opened.

5.6.2 Parameterisation Interface

- General Set-up The parameterisation interface consists of the following components:
- **Menu Line** with command buttons and test step display
 - **Macro Settings**, see 5.3
 - **Status Line** with continuous protocol switch and comment box



5.6.2.1 Menu Line

- [Execute]** Execute the current parameterisation.
- [Save]** Saves the current parameterisation as default.
- [Error Mode]** Opens the dialogue box *Error Mode*.
The response of the test program, in the case of errors, can be set via the switch *Error Exit*.
- [Reset]** Resets the test program
- [Quit]** Ends the parameterisation of the test steps.
- Test Step Indicator The current number indicates the position in the test program at which the test step is executed.

2.1.1.1 Macro Settings

- Contents This field contains the setting possibilities for the current macro (see 5.3.1)

2.1.1.2 Status Line

Continuous Protocol Switch	The continuous protocol, which records all the activities of the current test step can be switched on or off via this switch.
Comment Box	The test step can be described in more detail using this entry field. The comment of the respective test steps can be displayed in the action window. (see 5.6.4).

5.6.3 Parameterisation of the Test Steps

Executing the Parameterisation	<ol style="list-style-type: none">1. Double click on the test step you wish to parameterise.<ul style="list-style-type: none">– The parameterisation interface and the corresponding macro is opened.2. Set the parameters for the corresponding device (see 5.3).3. Switch the continuous protocol switch on or off with the mouse.4. Enter your text in the comment box via the keyboard.5. Click on [Error mode], to set program cancel for when errors occur.<ul style="list-style-type: none">– The dialogue box <i>Error mode</i> is opened and the response of the test program, in the case of errors, can be set via the switch <i>Error Exit</i>.– Clicking with the mouse on [Quit] closes the dialogue box.6. Click on [Execute], to set the current parameterisation.7. Click on [Save], to save the current parameterisation.8. Click on [Reset], to reset the test program.9. Click on [Quit], to end the parameterisation.
--------------------------------	-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------



Attention To be able to parameterise the test step *Device Function* the corresponding device driver must have been previously opened with the function *Device On/Off*.

5.6.4 Comment Display

Introduction	For better understanding of the test procedures a comment may be entered in the parameterisation interface of each test step. This may be presented in the user interface of the Smart Sequencer in the list box <i>Comments</i> :
Display Comment	<ul style="list-style-type: none">• Execute one of the following to switch over from the list box <i>Macro Library</i>: to the list box <i>Comments</i>:<ul style="list-style-type: none">▪ Input command Settings/Show comments▪ Activate the command button [Macros / Comments F10]▪ Push the function key [F10]



Attention You can switch between the display of the list boxes *Macro Library*: and *Comment* by pushing the function key [F10] or by activating the command button **[Macros / Comments F10]** repeatedly.

5.7 Debugger – Implementing the Test Program

5.7.1 General

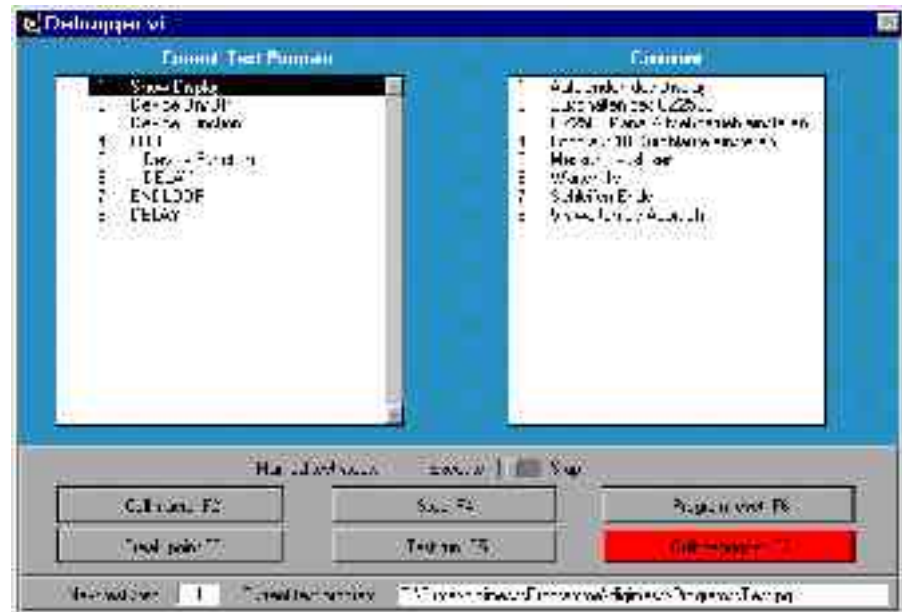
Introduction	The debugger is used for implementing the test runs which have previously been set in the Smart Sequencer.
Starting the Debugger	<ul style="list-style-type: none">• Execute one of the following to start the debugger from the Smart Sequencer:<ul style="list-style-type: none">▪ Enter the command Extras/Debugger▪ Activate the control button [Debugger F11]▪ Push the function key [F11]– The debugger interface for the test of the test procedure is opened.

5.7.2 Debugger Interface

Set-up

The debugger interface consists of the following components:

- **Display Range** with list boxes *Current Test Program:* and *Comment:*
- **Control Panel** with command buttons and switches
- **Status Line** with test step display and path display



5.7.2.1 Display Range

Set-up

The display range contains:

- The marking box for deactivating the test step
- The list box *Current Test Program:* with marking box
- The list box *Comment:*

Marking Box

Individual test steps can be marked using the marking box to switch off the steps in test run and in continuous run.

Current Test Program

The box contains the test steps of the test program which is to be implemented.

Comment

The box contains the comment for each test step which leads to a better understanding of the test procedure.

2.1.1.3 Control Panel

Switch

Determines if the test steps, indicated in the marking box, are executed or skipped.

[Call up Macro]

Opens the parameterisation interface and enables parameterisation of the test step (see 5.3.1).

[Break Point]

Defines selected test steps as a break point for the test run of the test program.

[Step]

Executes the selected test step.

Repeated activation of this command button executes step by step processing of the test procedure.

[Test Run]

Executes the test run of the entire test procedure.

[Reset Program]

Sets the current test step to the beginning of the program and initialises the hardware.

This function should always be called up if a defined initial state of the test object is necessary or if a dangerous situation occurs.

[Exit Debugger]

Ends the implementation of the test procedure.

2.1.1.4 Status Line**Test Step Display**

The test program loaded in the debugger can be executed by the user in step mode. In the box *Next test step*, the next step to be executed is displayed.

Path Display

The field contains the complete path of the current test program.

5.7.3 Marking the Test Steps**Introduction**

Individual test steps can be marked for skipping either in the test run or in the continuous run of the test program. It is thus possible to blank out parts from the created test procedure, thus enabling universal test program.

Mark Test Steps

- Click on the marking box to the left of the list box *Current Test Program*: to mark the test step in the line.
 - The marking is displayed in red.





Attention Test steps can also be marked in the Smart Sequencer interface.

5.7.4 Calling up the Test Steps

Introduction Parameter settings and comment of a test step can be set or changed according to the specific test object (see 5.6).

Starting the Parameterisation

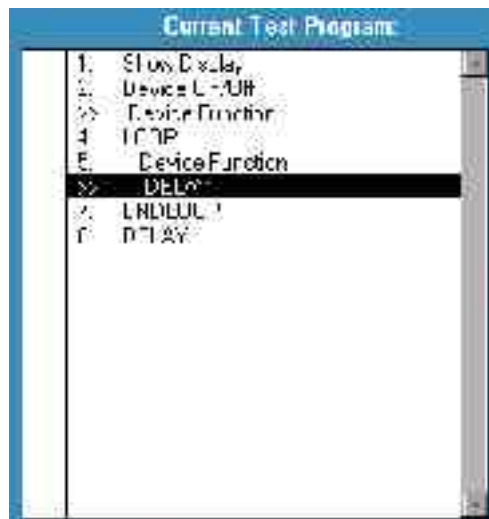
- Click on **[Call macro F2]**, to set the parameters of a selected test step.
 - The window for entering the parameter is opened (see 5.6.3).

5.7.5 Inserting the Break Points

Introduction Setting the break points within a test procedure simplifies its implementation. For example, the parameter settings of test steps can be changed (see 5.6).

Setting/Deleting the Break Points

1. Click on **[Break point F3]**, to define a selected test step as the break point.
 - The test step numbering sequence is replaced by >>.



2. Click on **[Break point F3]** again, to delete the test step defined as Break Point.
 - The number of the test step is displayed again.



Attention The break points are only important for the test procedure, they are ineffective in step mode.

5.7.6 Step Mode of the Test Program

Introduction The test program loaded in the debugger can be executed in step mode.

The test program is always processed sequentially. Jumping to a specific step is not possible.

Deactivating the Test Steps

1. Mark the test steps which are not to be executed (see 5.7.3).
2. Set the switch *Marked test steps*: to the position **skip**.
 - The marked test steps are not executed.

Starting the Step Mode

3. Click on **[Step F4]**, to execute the first test step.
 - The next step to be executed is displayed in the status line in the box *Current Test Step*.
4. Click repeatedly on **[Step F4]**, to execute the other test steps.



Attention Break points are not considered in step mode because the current test step can be called up at any time (see 5.7.4). Sub-routines are not executed in step mode. These must be loaded separately into the Smart Sequencer.

5.7.7 Test Run of the Test Program

Starting/Canceling the Test Run

1. Click on **[Test run F5]**, to open the *Testlauf* window.
 - The current (first) test step with numbering, name and comment is displayed in the window.



2. Click on **[Start test run (F1)]**, to start the test run.
 - The tests are executed sequentially and at maximum processing speed. At the end of the test run, the test reverts to the debugger.
3. Click on **[Abort test run (ESC)]**, to abort the test run.
 - The test reverts to the debugger.

Marked Test Steps

The test run considers marked test steps. If the switch is activated in the debugger (see 5.7.2.2), the marked test steps are skipped.

Cancelling the Test Procedure

If the test run reaches a test step defined as a break point or if an error occurs during execution of a test step, the macro interface appears. The test step can thus be individually tested. If an error occurs, the cause is easy to detect.

Test Step Error

In case of error during a test step the error window appears before the macro interface.



- The error source is described in the display *Ursache:* and error mode can be determined with the switch.
- The error mode decides if the current test step continues with the macro which follows the test step where the error occurred, or if it is set to the beginning of the program.

Sub-Routine

Sub-routines which are part of the test procedure, are executed in the test run. If a sub-routine is entered its name appears on the interface. Setting break points within sub-routines is not possible. Marks for skipping test steps are however considered.

5.7.8 Resetting the Test Program

Introduction

If a defined initial state of the test object is required or if a dangerous situation occurs, the program can be reset.

Resetting the Test Program

- Click on **[Program reset F6]**.
 - The current test step is set at the beginning of the program and the hardware is initialised.

5.8 Testing the Test Procedure

5.8.1 Continuous Run of the Test Program



Attention The continuous run can only be executed if all changes to the loaded program have been saved. This is strictly a safety measurement since the test run can last hours or days.

Preparing the Test Run

1. In Smart Sequencer select the command **Extras/Continuous run** to open the window *Continuous test program run!*.
 - The following parameters are displayed:
 - Current cycle and set cycle number of test runs
 - Current (first) test step with numbering, name and comment
 - Set save cycle for the protocol
 - Set path for saving the protocol file



2. Enter the number of test runs (default 1) in the entry field **Cycles**.
 - Cycle number 0 means that the continuous run can only be ended with the help of the command button **[Abort continuous run (ESC)]**.
3. Set the switch to **Yes** to skip the marked test steps.
4. Enter the complete path, including protocol file name, in the entry field *Protocol file*.
 - Or click on **[Browse]**, to enter the complete path, including file name.
 - The continuous run generates a protocol which holds the test results of the test step, as well as statistical information.
5. Determine in the entry field **Memory cycle** which test runs have their results written into the protocol.

Starting the Test Run

6. Click on **[Start continuous run (F1)]**, to start the continuous run.
 - The test steps are executed sequentially and at maximum processing speed.
 - During continuous run the current active test step with macro name as well as accompanying comment are displayed in the field *Current cycle* on the interface.
 - The continuous run reverts to the debugger when it has ended.
7. Click on **[Abort continuous run (ESC)]**, to abort the continuous run.
 - This causes a return to the debugger.

Sub-Routine

If a sub-routine is entered an additional display, which contains the name of the sub-routine, appears on the interface. According to the switch setting on the interface, marked test steps are skipped or executed also in the sub-routine.

5.8.2 Loading the Protocol

Introduction

Protocols which are generated as a result of test procedures during continuous run can be loaded and displayed. The protocol has Excel format to allow for more user friendly processing of results.

It is not necessary to load MS Excel if the user does not require analysing functions. A text window displays the contents of the protocol. Changes are not possible.

Protocol Contents

The continuous run generates a protocol which holds the test results of the test step as well as statistical information e. g., number of executed runs, total running time, etc.

Displaying the Protocol

- In Smart Sequencer select the command **Extras/Load protocol**, to load a saved protocol.
 - The protocol can be searched for and opened via the window *Please select protocol file:*.

5.8.3 Printing the Test Steps

Introduction

This function is intended exclusively for documentation purposes. The print out begins with a small header, which contains the name of the test program, the current date and time. There follows a list of available test steps in the loaded test program with accompanying comment.

Printing the Test Steps

- In Smart Sequencer select the command **Extras/Print test steps**, to print the test steps of the loaded test program.



Attention

If no test program has been loaded, the message: **There are no test Steps to be printed.** appears and you should confirm this with **[OK]**.
