



OmniScan MXU Software

User's Manual

Software Version 4.4

DMTA-20072-01EN [Q1000010] — Rev. C
January 2017

This instruction manual contains essential information on how to use this Olympus product safely and effectively. Before using this product, thoroughly review this instruction manual. Use the product as instructed. Keep this instruction manual in a safe, accessible location.

Olympus Scientific Solutions Americas, 48 Woerd Avenue, Waltham, MA 02453, USA

Copyright © 2015, 2016, 2017 by Olympus. All rights reserved. No part of this publication may be reproduced, translated, or distributed without the express written permission of Olympus.

This document was prepared with particular attention to usage to ensure the accuracy of the information contained therein, and corresponds to the version of the product manufactured prior to the date appearing on the title page. There could, however, be some differences between the manual and the product if the product was modified thereafter.

The information contained in this document is subject to change without notice.

Software version 4.4
Part number: DMTA-20072-01EN [Q1000010]
Rev. C
January 2017

Printed in Canada

All brands are trademarks or registered trademarks of their respective owners and third party entities.

Table of Contents

List of Abbreviations	xi
Important Information – Please Read Before Use	1
Intended Use	1
Instruction Manual	1
Safety Symbols	1
Safety Signal Words	2
Note Signal Words	3
Warnings	3
Warranty Information	4
Technical Support	4
Introduction	5
Other Relevant Olympus Documents	5
1. Instrument Overview	7
1.1 Starting Up the OmniScan Instrument	9
1.2 Shutting Down the OmniScan MXU Software	10
1.3 Connecting UT and PA Probes on the OmniScan MX2	11
1.4 Connecting UT and PA Probes on the OmniScan SX	12
1.5 Main Controls	14
1.6 OmniScan MX2 Increment Steps	16
1.7 OmniScan SX Increment Steps	16
1.8 OmniScan MX2 Function Keys	16
1.9 OmniScan SX Function Keys	19
1.10 Using OmniScan Function Keys as Software Function Shortcuts	20
1.11 Editing Parameters Using OmniScan Function Keys	21
1.12 Touch Screen	22
1.12.1 Entering Values on the Software Keyboard	22

1.12.2	Using the Touch Screen	24
1.13	Indicators	26
1.13.1	Power Indicator Light	26
1.13.2	Acquisition Indicator Light	27
1.13.3	Alarm Indicators	29
1.14	Navigating in the Menu Using Hardware Controls	29
2.	Software User Interface	33
2.1	Shortcut Menus	34
2.1.1	Title Bar Shortcut Menus	34
2.1.1.1	Menu-Screen and Full-Screen Modes	35
2.1.1.2	Single or Multiple Group Display (OmniScan MX2 Only)	35
2.1.1.3	Layout Selection	36
2.1.1.4	Group Selection Section (OmniScan MX2 Only)	37
2.1.1.5	Scan Section	37
2.1.1.6	Index Section	38
2.1.1.7	Angle/VPA Section	38
2.1.1.8	Display/Hide Options	39
2.1.2	Display Area Shortcut Menu	39
2.1.3	Shortcut Menus in the Reading-Field Area	40
2.1.4	Adjusting the Acquisition Parameters	43
2.2	Main Features of the Interface	44
2.3	OmniScan MXU Software Navigation	45
2.3.1	Navigating in the OmniScan MXU Software	46
2.3.2	Menu Organization	46
2.3.2.1	Menus Used to Define the Setup	47
2.3.2.2	Menus Used for Inspection	48
2.3.2.3	Menus Used for Configuration	48
2.4	Reading-Field Area	49
2.5	Gain	50
2.6	Status Indicators	51
2.7	Battery Charge Indicator(s)	54
2.8	Acquisition Modes	55
2.8.1	Inspection Mode	56
2.8.2	Analysis Mode	56
2.9	Data Screen	57
2.9.1	Scans, Views, and Layouts	57
2.9.2	RayTracing	60
2.9.2.1	Using RayTracing to Define the Setup	61
2.9.2.2	Using RayTracing to Inspect a Part	62
2.9.2.3	Using RayTracing to Analyze the Indications	62
2.9.2.4	Multiple Group Layouts in OmniPC	64

2.9.3	Rulers/Scales	65
2.10	Outline Colors	68
2.10.1	Reading-Field Colors	68
2.10.2	Parameter Button Colors	70
2.11	Parameter Buttons	71
2.12	Compression Feature	75
2.13	Online Help	75
2.13.1	Displaying Contextual Help	76
2.13.2	Displaying Online Help for Wizard Steps	76
3.	General Procedures	79
3.1	Setting Preferences	79
3.1.1	Setting the Date and Time	79
3.1.2	Selecting the Length Measurement Units	80
3.1.3	Configuring the Digital Inputs	80
3.1.4	Changing the Screen Color Scheme	81
3.1.5	Toggling the Wizard Help Screen	81
3.2	Managing Files	81
3.2.1	Saving Files	82
3.2.1.1	Saving a Setup	82
3.2.1.2	Saving a Data File	82
3.2.2	Opening Files	83
3.2.2.1	Opening Any Type of File	83
3.2.2.2	Opening Setup Files	83
3.2.2.3	Opening Connectivity Files	84
3.2.2.4	Opening Data Files	84
3.2.2.5	Opening Image Files	84
3.2.2.6	Opening Report Files	85
3.2.3	Setting the Save/Print Key	85
3.3	Producing Reports	86
3.3.1	Configuring a Report	86
3.3.2	Printing a Report from a Computer	88
4.	Setup Procedures	91
4.1	Creating the Application Setup with the Wizards	91
4.1.1	Configuring the Part and Weld	92
4.1.2	Modifying the Setup	93
4.2	Selecting the Probe and the Wedge	93
4.3	Defining a Probe	96
4.4	Defining a Wedge	98
4.5	Defining the Part to Be Inspected	101

4.6	Characterizing a Probe Using the FFT	102
4.7	Working with Multiple Groups (OmniScan MX2 Only)	105
4.8	Selecting the Group Mode on the OmniScan MX2	106
4.9	Selecting the Group Mode on the OmniScan SX	106
5.	Calibration Procedures	109
5.1	Selecting the Element to Calibrate	109
5.2	Reflector Types	111
5.3	Scan Types	113
5.4	Ultrasonic Calibration	113
5.4.1	Ultrasonic Velocity Calibration	114
5.4.2	Wedge Delay Calibration for a UT Group	117
5.4.3	Wedge Delay and Probe Center Separation Calibration (TOFD Only)	119
5.4.4	Quick TOFD Calibration	120
5.4.5	Velocity and Wedge Delay Calibration for a UT Group	121
5.4.6	Wedge Delay Calibration for a PA Group	123
5.4.6.1	Calibrating the Wedge Delay for All Focal Laws	124
5.4.6.2	Calibrating the Wedge Delay for Two or Three Focal Laws in a PA Group	126
5.4.7	Sensitivity Calibration for a UT Group	128
5.4.8	Sensitivity Calibration for a PA Group	129
5.4.8.1	Calibrating the Sensitivity for All Focal Laws	129
5.4.8.2	Calibrating the Sensitivity for Two or Three Focal Laws	132
5.5	Sizing Calibration	135
5.5.1	DAC Calibration	135
5.5.2	TCG Calibration	138
5.5.3	DGS Calibration	141
5.5.4	AWS Calibration	146
5.6	Encoder Calibration	148
6.	Inspection Procedures	151
6.1	Setting the General Inspection Parameters	151
6.2	Setting the Tx/Rx Mode (UT Group Only)	151
6.3	Configuring the Reference Gain	152
6.4	Setting Up a PA-TOFD Inspection (OmniScan MX2 Only)	153
6.5	Configuring a Thickness C-Scan	155
6.6	Using the Export C-Scan Function	157
6.7	Using the Profile B-Scan View	158
6.8	Using the Export B-Scan Function (OmniPC Only)	159
6.9	Setting Alarms and Outputs	160
6.9.1	Setting an Alarm	161

6.9.2	Setting an Alarm Output	161
6.9.3	Setting an Analog Output (OmniScan MX2 Only)	162
6.10	Setting Sizing Functions	162
6.10.1	Manually Configuring the DAC Curve	162
6.10.2	Automatically Configuring the Linear DAC Curve	165
6.10.3	Manually Configuring the TCG Curve	166
6.11	Setting Measurements	167
6.11.1	Configuring Reading Fields	168
6.11.2	Configuring and Building an Indication Table	169
6.12	Configuring the Screen	171
6.12.1	Changing the Current Layout and the Current View	171
6.12.2	Displaying the Cursors	172
6.12.3	Displaying the Gates	173
6.12.4	Displaying Various A-Scan Related Overlays	174
6.12.5	Displaying the Weld Geometry Overlay	174
6.12.6	Displaying the Leg Overlay	175
6.12.7	Configuring the Grid	175
6.12.8	Configuring the View Parameters	176
6.12.9	Changing the Color Palettes	177
6.13	Performing an Inspection with an Encoder	179
6.14	Indexing a Scan	181
7.	Complementary Procedures	183
7.1	Using the File Manager	183
7.1.1	File Manager Interface	183
7.1.2	File Manager Navigation	185
7.2	Setting the Administrator Password	186
7.3	Creating a Custom Report Template	187
7.4	Changing the Company Logo	189
7.5	Creating a Color Palette	190
7.5.1	Color Palette File Format	191
7.5.2	Color Palette Rules	193
7.6	Characterizing a Probe	194
7.7	Connecting the OmniScan MX2 Directly to a Computer	196
7.7.1	Connecting the Hardware	196
7.7.2	Configuring the Computer Network Connection — Windows XP	197
7.7.3	Creating a User Account on a Computer — Windows XP	200
7.7.4	Creating a Shared Folder on a Computer — Windows XP	203
7.7.5	Configuring OmniScan MX2 Network Connection — Windows XP	209
7.7.6	Configuring the Computer Network Connection — Windows 7	211
7.7.7	Creating a User Account on a Computer — Windows 7	214
7.7.8	Creating a Shared Folder on a Computer — Windows 7	217

7.7.9	Configuring the OmniScan MX2 – Windows 7	222
7.7.10	Saving OmniScan Data on a Computer	223
7.8	Connecting the OmniScan MX2 to a Network	224
7.9	Importing OmniScan Data into TomoView	225
7.10	Transferring Data – OmniScan MX2 with MCDU-02 and TomoView	226
7.11	Importing a .law File into the OmniScan Instrument (PA Group Only)	226
7.12	Loading a Custom Color Palette	228
8.	Menu Description	229
8.1	File Menu	229
8.1.1	Setup Submenu	229
8.1.2	Report Submenu	231
8.1.2.1	Open/Save Category	231
8.1.2.2	Format Category	232
8.1.2.3	User Fields Category	234
8.1.3	Data Submenu	235
8.1.4	Image Submenu	235
8.1.5	Data Settings Submenu	236
8.2	Wizard Menu	238
8.2.1	Part & Weld Submenu	240
8.2.2	Setup Submenu	240
8.2.3	Calibration Submenu	242
8.3	UT Settings Menu	244
8.3.1	General Submenu	244
8.3.2	Pulser Submenu	246
8.3.3	Receiver Submenu	251
8.3.4	Beam Submenu	253
8.3.5	Advanced Submenu	255
8.4	Measurements Menu	257
8.4.1	Cursors Submenu	257
8.4.2	Reading Submenu	259
8.4.2.1	Generic Reading Codes	262
8.4.2.2	Gate Category Reading	262
8.4.2.3	Positioning Category Reading	265
8.4.2.4	Cursor Category Readings	268
8.4.2.5	Sizing Category Readings	273
8.4.2.6	Code Category Reading	274
8.4.2.7	Envelope Category Readings	275
8.4.2.8	Corrosion and Composite Category Readings	276
8.4.2.9	Immersion Category Readings	278
8.4.2.10	UT Settings Category Readings	279
8.4.3	Indication Table Submenu	279

8.5	Display Menu	280
8.5.1	A-Scan Settings Submenu	280
8.5.2	Overlay Submenu	283
8.5.3	RayTracing Submenu	286
8.6	Sizing Menu	286
8.6.1	Type Submenu	287
8.6.2	Operator Mode Submenu	288
8.6.3	Type Setup Submenu	290
8.6.3.1	Parameters for DAC, Linear DAC, and TCG Curves	290
8.6.3.2	Parameters for DGS	292
8.6.3.3	Parameters for AWS Curves	293
8.6.4	Curves Setup Submenu	294
8.6.4.1	Parameters for DAC and TCG Curves	294
8.6.4.2	Parameters for Linear DAC Curve	294
8.7	Gates/Alarms Menu	295
8.7.1	Gates Submenu	295
8.7.1.1	Gate Position	300
8.7.1.2	Saturated Gate	300
8.7.2	Alarm Submenu	301
8.7.3	Output Submenu	302
8.7.4	Analog Submenu (OmniScan MX2 Only)	303
8.7.5	Thickness Submenu	304
8.8	Group/Probe & Part Menu	306
8.8.1	Probe & Wedge Submenu	306
8.8.1.1	Probe Definition Manager	307
8.8.1.2	Wedge Definition Manager	310
8.8.2	Position Submenu	312
8.8.3	Parts Submenu	314
8.9	Focal Law Menu (PA Group Only)	315
8.9.1	Configuration Submenu	316
8.9.2	Aperture Submenu	318
8.9.3	Beam Submenu	318
8.10	Scan Menu	319
8.10.1	Inspection Submenu	319
8.10.2	Encoder Submenu	320
8.10.3	Area Submenu	323
8.10.4	Data Submenu	324
8.10.5	Start Submenu	324
8.11	Preferences Menu	326
8.11.1	Setup Submenu	326
8.11.1.1	DIN Category	326
8.11.1.2	Gates Category	328

8.11.1.3	250 % Mode Category	328
8.11.1.4	Interpolation Category	328
8.11.1.5	Interleave Category	328
8.11.2	Instrument Submenu	328
8.11.2.1	Units Category	329
8.11.2.2	Screen Category	329
8.11.2.3	System Category	330
8.11.2.4	Network Settings Category (OmniScan MX2 Only)	330
8.11.2.5	External Storage Category (OmniScan MX2 Only)	331
8.11.2.6	Help Category	332
8.11.3	Tools Submenu	332
8.11.4	FFT Submenu	334
8.12	Setup Configuration Menu	336
8.13	Scan Configuration Menu	339
List of Figures		341
List of Tables		347
Index		349

List of Abbreviations

Acq	acquisition
AM	ante meridiem
ASME	American Society of Mechanical Engineers
AVG	Abstand-Verstärkung-Größe
AWS	American Welding Society
CSC	curved-surface correction
DAC	distance-amplitude correction
DC	direct current
DGS	distance gain size
DHCP	Dynamic Host Configuration Protocol
DIN	digital inputs
ERS	equivalent reflector size
FFT	fast Fourier transform
FSH	full-screen height
FW	full wave
HTML	hypertext markup language
HW	half wave
ID	inside diameter
IP	Internet Protocol
JIS	Japan Industrial Standards
MCDU	motor controller drive unit
ML	material loss
N/A	not applicable
ND	no detection (of signal)

NS	no synchronization
OD	outside diameter
P/C	pitch-catch
P/E	pulse-echo
PA	phased array
PCS	probe center separation
PM	post meridiem
PRF	pulse repetition frequency
PW	pulse width
RF	radio frequency
RGB	red green blue
SDH	side-drilled hole
TCG	time-corrected gain
TCP/IP	Transmission Control Protocol/Internet Protocol
TT	through-transmission
TTL	transistor-transistor logic
US	United States
USB	universal serial bus
UT	ultrasonic technology
VPA	virtual probe aperture
WD	wedge delay
XML	extensible markup language

Important Information — Please Read Before Use

Intended Use

OmniScan MXU is a software program for OmniScan products used in nondestructive inspections on industrial and commercial materials.

Instruction Manual

This instruction manual contains essential information on how to use this Olympus product safely and effectively. Before using this product, thoroughly review this instruction manual. Use the product as instructed.

Keep this instruction manual in a safe, accessible location.

IMPORTANT

Some of the details of components and software images shown in this manual may differ from your instrument's components or software display. However, the principles remain the same.

Safety Symbols

The following safety symbols might appear on the instrument and in the instruction manual:



General warning symbol

This symbol is used to alert the user to potential hazards. All safety messages that follow this symbol shall be obeyed to avoid possible harm or material damage.



Shock hazard caution symbol

This symbol is used to alert the user to potential electric shock hazards. All safety messages that follow this symbol shall be obeyed to avoid possible harm.

Safety Signal Words

The following safety symbols might appear in the documentation of the instrument:



DANGER

The DANGER signal word indicates an imminently hazardous situation. It calls attention to a procedure, practice, or the like that if not correctly performed or adhered to will result in death or serious personal injury. Do not proceed beyond a DANGER signal word until the indicated conditions are fully understood and met.



WARNING

The WARNING signal word indicates a potentially hazardous situation. It calls attention to a procedure, practice, or the like that if not correctly performed or adhered to could result in death or serious personal injury. Do not proceed beyond a WARNING signal word until the indicated conditions are fully understood and met.



CAUTION

The CAUTION signal word indicates a potentially hazardous situation. It calls attention to a procedure, practice, or the like that if not correctly performed or adhered to may result in minor or moderate personal injury, material damage, particularly to the product, destruction of part or all of the product, or loss of data. Do not proceed beyond a CAUTION signal word until the indicated conditions are fully understood and met.

Note Signal Words

The following symbols could appear in the documentation of the instrument:

IMPORTANT

The IMPORTANT signal word calls attention to a note that provides important information, or information essential to the completion of a task.

NOTE

The NOTE signal word calls attention to an operating procedure, practice, or the like, which requires special attention. A note also denotes related parenthetical information that is useful, but not imperative.

TIP

The TIP signal word calls attention to a type of note that helps you apply the techniques and procedures described in the manual to your specific needs, or provides hints on how to effectively use the capabilities of the product.

Warnings



WARNING

General Warnings

- Carefully read the instructions contained in this instruction manual and in the *OmniScan MX and MX2 User's Manual* or *OmniScan SX User's Manual* prior to turning on the instrument.
- Keep the instruction manuals in a safe place for further reference.
- Follow the installation and operation procedures.
- It is imperative to respect the safety warnings on the instrument and in the instruction manuals.
- If the equipment is used in a manner not specified by the manufacturer, the protection provided by the equipment could be impaired.

Warranty Information

Olympus guarantees your Olympus product to be free from defects in materials and workmanship for a specific period, and in accordance with conditions specified in the *Olympus Scientific Solutions Americas Inc. Terms and Conditions* available at <http://www.olympus-ims.com/en/terms/>.

The Olympus warranty only covers equipment that has been used in a proper manner, as described in this instruction manual, and that has not been subjected to excessive abuse, attempted unauthorized repair, or modification.

Inspect materials thoroughly on receipt for evidence of external or internal damage that might have occurred during shipment. Immediately notify the carrier making the delivery of any damage, because the carrier is normally liable for damage during shipment. Retain packing materials, waybills, and other shipping documentation needed in order to file a damage claim. After notifying the carrier, contact Olympus for assistance with the damage claim and equipment replacement, if necessary.

This instruction manual explains the proper operation of your Olympus product. The information contained herein is intended solely as a teaching aid, and shall not be used in any particular application without independent testing and/or verification by the operator or the supervisor. Such independent verification of procedures becomes increasingly important as the criticality of the application increases. For this reason, Olympus makes no warranty, expressed or implied, that the techniques, examples, or procedures described herein are consistent with industry standards, nor that they meet the requirements of any particular application.

Olympus reserves the right to modify any product without incurring the responsibility for modifying previously manufactured products.

Technical Support

Olympus is firmly committed to providing the highest level of customer service and product support. If you experience any difficulties when using our product, or if it fails to operate as described in the documentation, first consult the user's manual, and then, if you are still in need of assistance, contact our After-Sales Service. To locate the nearest service center, visit the Service Centers page at: <http://www.olympus-ims.com>.

Introduction

The OmniScan MXU software features ultrasonic inspection functions suitable for numerous nondestructive testing applications, and it combines both conventional ultrasound (UT) and phased array (PA) ultrasonic operation modes. The OmniScan MXU software runs on the innovative and portable OmniScan MX2 and OmniScan SX instruments.

NOTE

This manual covers the MXU software for both OmniScan MX2 and OmniScan SX instruments. However, certain sections, procedures, steps, or descriptions are specific to only one instrument:

- When a particular section, procedure, step, or description applies to only one instrument, this section, procedure, step, or description is clearly identified with the instrument model it applies to.
 - When there is no mention of the instrument model in a section, procedure, step, or description, this means that this section, procedure, step, or description applies to both the OmniScan MX2 and the OmniScan SX instruments.
 - The term “OmniScan instrument” refers to both the OmniScan MX2 and the OmniScan SX instruments.
-

Other Relevant Olympus Documents

Several other Olympus documents are relevant to the OmniScan instrument:

OmniScan MX and MX2 – User’s Manual

Provides a description and instructions on how to use the OmniScan MX and the OmniScan MX2. Refer to this document to understand, in particular, how to navigate in the software user interface using the hardware keys on the instrument.

OmniScan MX2 – Getting Started

A short leaflet containing essential information on how to quickly start operating the OmniScan MX2.

OmniScan SX – User’s Manual

Provides operation information to help you learn how to use the OmniScan SX instrument.

OmniScan SX – Getting Started Guide

A short leaflet containing essential information on how to quickly start operating the OmniScan SX.

1. Instrument Overview

The OmniScan instrument features front panel controls for easy and efficient operation of the OmniScan MXU software. Figure 1-1 on page 8 and Figure 1-2 on page 9 show the OmniScan MX2 and OmniScan SX front panels and the available controls and indicators. The OmniScan instrument allows you to connect a USB keyboard and USB mouse to extend its user interface.

NOTE

In this document, hardware controls that you press to activate are referred to as *keys*. The term *button* is reserved for software controls.

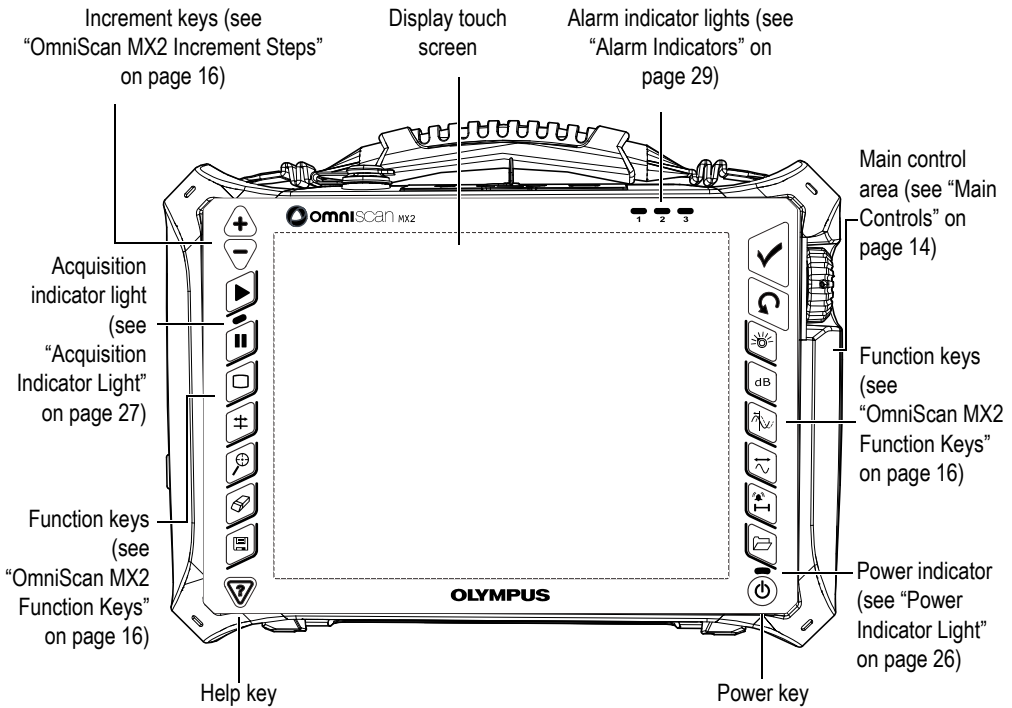


Figure 1-1 Front panel controls of the OmniScan MX2

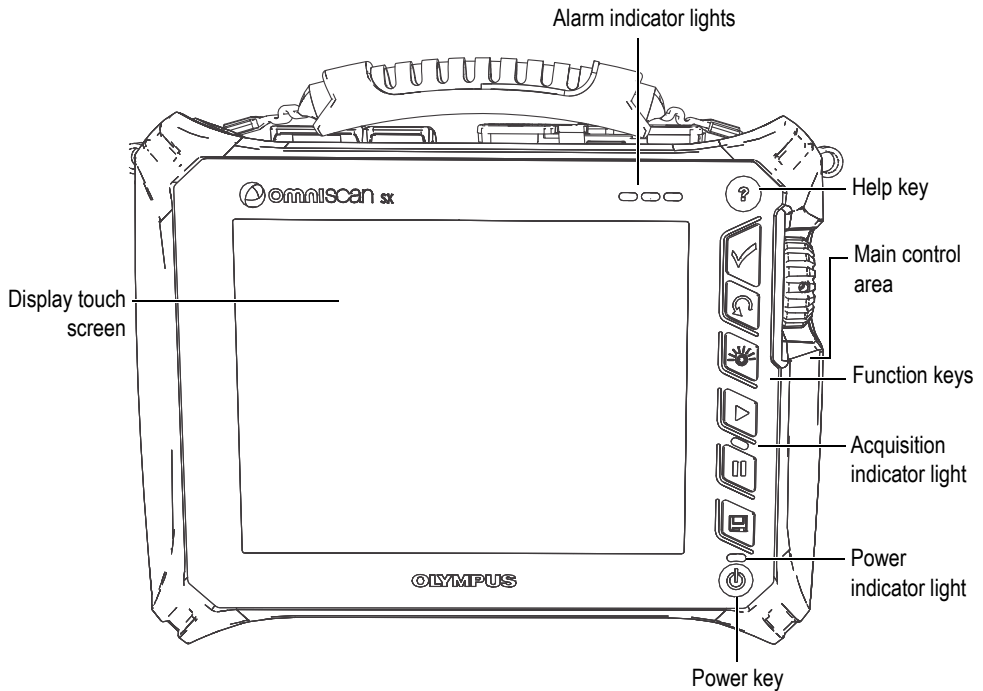


Figure 1-2 Front panel controls of the OmniScan SX

1.1 Starting Up the OmniScan Instrument

To start up the OmniScan instrument

1. Press and hold the Power key (⏻) for one second; the key is located at the lower-right corner of the instrument front panel.
The instrument's power indicator illuminates, followed by the touch screen, which displays the Olympus wallpaper, and then the OmniScan logo. Next, the OmniScan MXU software loads automatically.
2. If more than one program is available on the instrument storage card, a series of vertical buttons appears on the screen, allowing you to select which program you want to start:
 - a) Choose the desired program.

- b) If you always choose the same program, you can skip the program selection step for future restarts by selecting **Always boot the selected application** below the software buttons.
- c) To regain the ability to choose the program at startup, select **Preferences > Instrument > Category = System**, and then **Manual boot**.

NOTE

At start-up, the OmniScan instrument loads the OmniScan MXU software from the storage memory card inserted in the built-in card reader on the right side of the instrument.

1.2 Shutting Down the OmniScan MXU Software

The OmniScan MXU software shuts down automatically when you turn off the OmniScan instrument.

To turn off the OmniScan instrument

1. Quickly press the Power key.
The “Select a command” message appears (see Figure 1-3 on page 10).

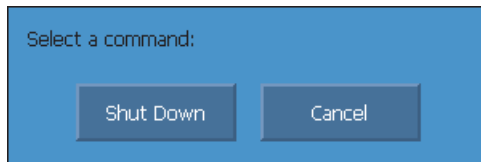


Figure 1-3 The Shut Down button

2. Select **Shut Down**.
A message asking you if you want to save your setup appears (see Figure 1-4 on page 11).

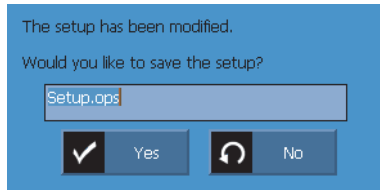


Figure 1-4 Saving the setup

3. To save your setup, select **Yes**.

NOTE

You can also turn off the OmniScan by pressing and holding the Power key for ten seconds. However, your setup will **NOT** be saved.

1.3 Connecting UT and PA Probes on the OmniScan MX2

The OmniScan MXU software combines both the phased array (PA) and conventional ultrasound (UT) operation modes. Depending on the module installed at the back of the OmniScan MX2, you can use UT or PA probes. Connect UT probes to the UT connectors, as shown in Figure 1-5 on page 12. Use a dual element probe for the pitch-catch (P/C) mode. You can also plug a UT probe into the PA connector using an adaptor.

Connect a phased array probe to the OmniScan MX2 PA connector, as shown in Figure 1-6 on page 12.

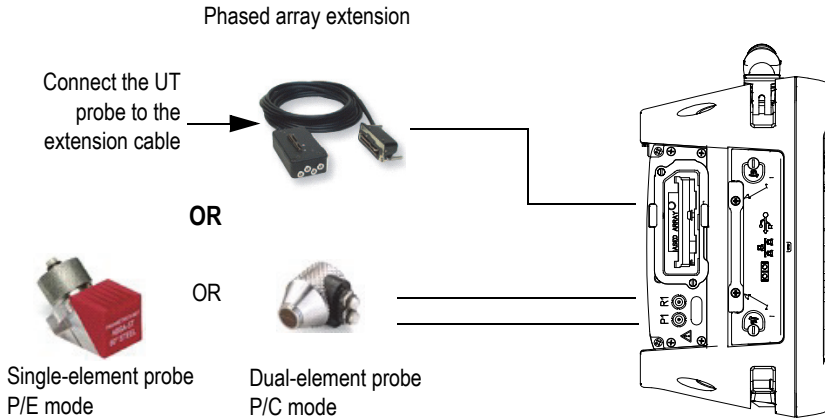


Figure 1-5 Requirements for UT probe connection

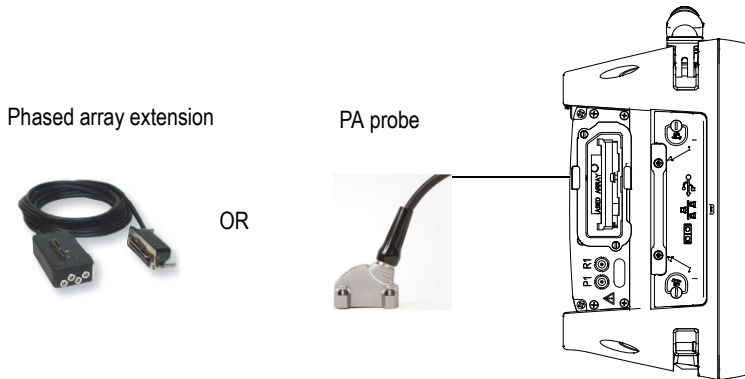


Figure 1-6 Requirements for PA probe connection

1.4 Connecting UT and PA Probes on the OmniScan SX

The OmniScan MXU software can combine both the phased array (PA) and conventional ultrasound (UT) operation modes. Depending on the instrument configuration, you can use UT and/or PA probes. Connect UT probes to the UT connectors, as shown in Figure 1-7 on page 13. Use a dual element probe for the pitch-catch (P/C) mode.

Connect a phased array probe to the OmniScan SX PA connector, as shown in Figure 1-8 on page 13.

NOTE

The OmniScan SX UT model is not equipped with a PA connector.

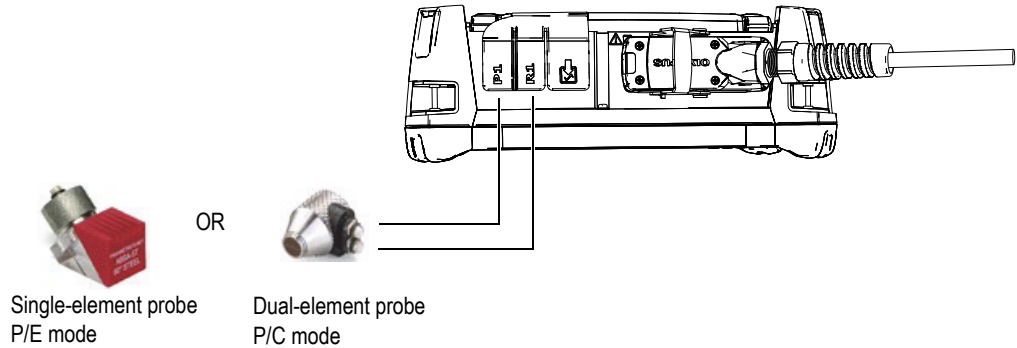


Figure 1-7 UT probe connection

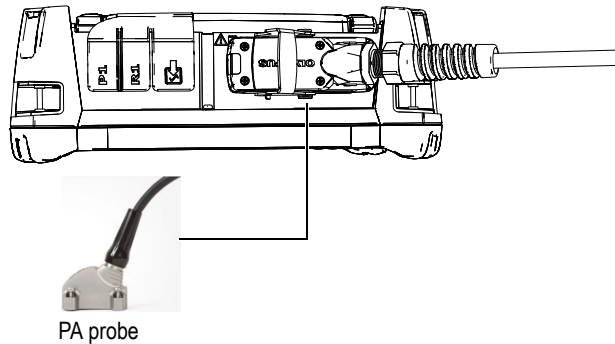


Figure 1-8 PA probe connection

1.5 Main Controls

The three main controls shown in Figure 1-9 on page 14 allow for full operation of the OmniScan MXU software.

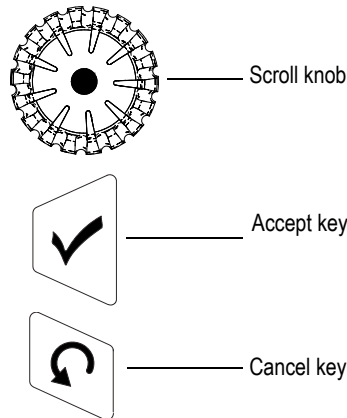





Figure 1-9 The main controls

- Rotating the Scroll knob clockwise or counterclockwise allows you to select a desired software button, or change a parameter value.
- Pressing the Accept key activates the current selection, and lets you move on to the next level in the menu hierarchy.
- Pressing the Cancel key cancels the current selection, and lets you move back to the previous level in the menu hierarchy.

The Scroll knob, Accept key, and Cancel key are versatile; their functions adapt to the context of the operation, as shown in Table 1 on page 15.

Table 1 Main control functions for various contexts

Context	 Rotating the Scroll knob	 Pressing the Accept key	 Pressing the Cancel key
Menu button selected	Scrolls through menu list	Goes to the first submenu of the selected menu button	Returns to the previously selected menu button
Submenu button selected	Scrolls through submenu list	Goes to the first parameter of the selected submenu button	Returns to the previously selected submenu button
Parameter button selected	Scrolls through parameter options	Selects or edits a parameter button value	Returns to the previously selected submenu button
Parameter value selected	Scrolls through predefined parameter values, or increases/decreases the parameter value	Accepts the selected or edited parameter value	Cancels the selection or the edit made to the parameter value, and returns to the parameter level

Editing Values with the Main Controls

Turning the Scroll knob in a vertical list shifts the selection upward/downward. Similarly, turning the Scroll knob in a horizontal list shifts the selection to the right/left.

In a numeric parameter value field, turning the Scroll knob increases/decreases the value.

In an alphanumeric parameter value field, tapping the parameter twice or pressing the Accept key twice opens the software keyboard.

1.6 OmniScan MX2 Increment Steps

The OmniScan MX2 increment keys can be used to modify the increment steps in edit mode.

Raise increment key ()

Used to turn on the full-screen mode or to increase the increment step.

Lower increment key ()

Used to turn off the full-screen mode or to lower the increment step.

1.7 OmniScan SX Increment Steps

When using the OmniScan SX, increment steps can only be changed when the selected numeric parameter is in edit mode.

To modify the increment steps of a numeric parameter

1. Tap the parameter twice or press the Accept key twice to open the software keyboard.
2. Use the + Δ or - Δ button to increase or decrease the increment (select from the predefined list of increments).

1.8 OmniScan MX2 Function Keys

There are 14 multipurpose function keys on the keypad on the left and right sides of the OmniScan MX2 front panel (see Figure 1-10 on page 17).

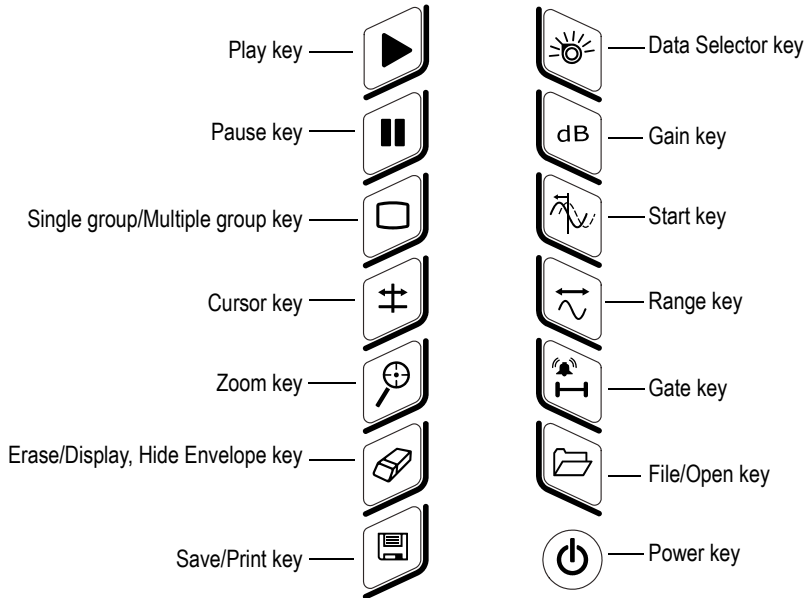


Figure 1-10 Functions on MX2 keypad

Table 2 on page 17 presents the function of each key. Depending on how long you press a key (short or long keystroke), you can activate different software functions.

Table 2 Key shortcuts for OmniScan MX2




Key	Short keystroke	Long keystroke
	Used to restart the inspection data acquisition, and/or the encoders, depending on the configuration in the Scan > Start menu.	N/A
	Used to toggle between the inspection and analysis modes.	N/A
	Used to turn the multiple group display on and off.	N/A

Table 2 Key shortcuts for OmniScan MX2 (continued)











Key	Short keystroke	Long keystroke
	Used to set the parameters for the displayed cursors.	Brings up Measurements > Cursors > Category .
	Used to zoom in/out in the current view according to the zoom settings (between the cursors). Press twice to reset the zoom.	N/A
	Used to reset the signal envelope and the alarms (when turned on).	Used to activate or deactivate the signal envelope.
	Used to save the report, data, or image, depending on the configuration in the File > Data Settings menu.	N/A
	In inspection mode: <ul style="list-style-type: none"> • When a PA group is selected, used to select the current focal law. • N/A when a UT group is selected. In analysis mode: <ul style="list-style-type: none"> • When a PA group is selected, used to select the angle, scan, and index positions (if applicable). When a UT group is selected, used to select the scan and index positions (if applicable).	Used to change the current group.
	Used to set the signal gain.	Brings up the UT Settings menu.
	Used to set the A-scan screen starting position.	Brings up the UT Settings menu.
	Used to set the A-scan screen range.	Brings up the UT Settings menu.
	Used to set the parameters for the displayed gates.	Brings up the Gates/Alarm menu.

Table 2 Key shortcuts for OmniScan MX2 (continued)

Key	Short keystroke	Long keystroke
	Used to start the file browser.	N/A

1.9 OmniScan SX Function Keys

There are 5 multipurpose function keys on the keypad on the right side of the OmniScan SX front panel (see Figure 1-11 on page 19).

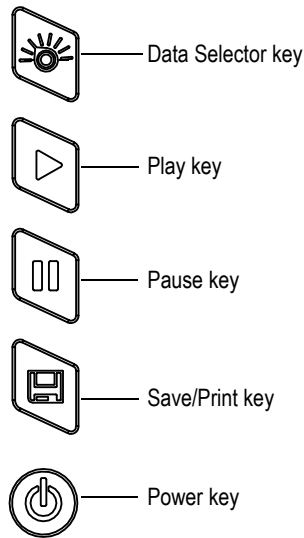






Figure 1-11 Functions on SX keypad

Table 3 on page 20 presents the function of each key. Depending on how long you press a key (short or long keystroke), you can activate different software functions.

Table 3 Key shortcuts for OmniScan SX

Key	Short keystroke	Long keystroke
	Used to restart the inspection data acquisition, and/or the encoders, depending on the configuration in the Scan > Start menu.	N/A
	Used to toggle between the inspection and analysis modes.	N/A
	Used to save the report, data, or image, depending on the configuration in the File > Data Settings menu.	N/A
	<p>In inspection mode:</p> <ul style="list-style-type: none"> When a PA group is selected, used to select the current focal law. N/A when a UT group is selected. <p>In analysis mode:</p> <ul style="list-style-type: none"> When a PA group is selected, used to select the angle, scan, and index positions (if applicable). When a UT group is selected, used to select the scan and index positions (if applicable). 	In inspection mode, used to select UT settings (gain, start, and range).

1.10 Using OmniScan Function Keys as Software Function Shortcuts

The primary purpose of the keypad is to provide shortcuts to frequently used software functions.

When pressed, many of the function keys bring up a pop-up button on the upper-left corner of the screen (see example shown in Figure 1-12 on page 21). The pop-up buttons allow you to quickly edit parameter values without changing the current menu and submenu selection, which lets you keep the current button selection. Use the Scroll knob to modify the value of the parameter. Use the Cancel key to close the pop-up button.

Pop-up buttons allow you to modify one or multiple parameters. For example, pressing the Gain key only allows you to modify the gain value in the pop-up button. However, repeatedly pressing the Gate key (when the pop-up is not in edit mode) allows you to access and modify the values of the **Start**, **Width**, and **Threshold** for all displayed gates.

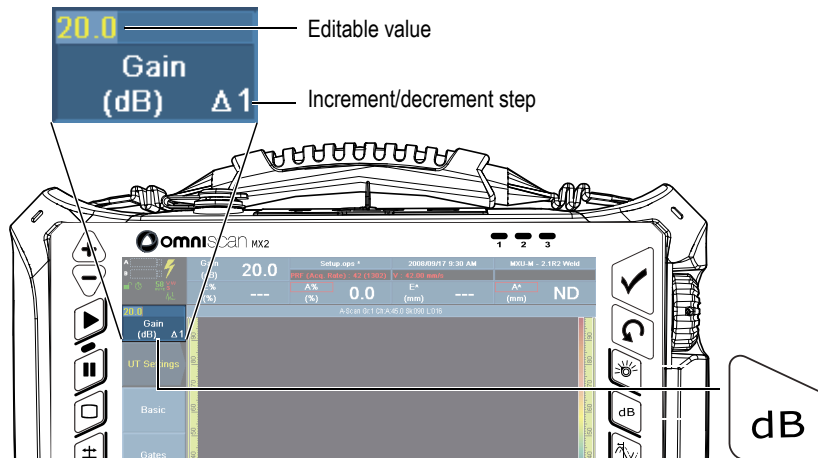


Figure 1-12 OmniScan MX2: Example pop-up button for the Gain function key

1.11 Editing Parameters Using OmniScan Function Keys

The following procedure explains how to edit the value of a parameter using a function key.

To edit the value of a parameter using the function key

1. Press the function key corresponding to the parameter you want to edit (see Table 2 on page 17 and Table 3 on page 20 for available functions).
The associated pop-up button appears in the upper left corner, or the associated menu, submenu, and parameter is chosen.
2. For a pop-up button with multiple parameters, rotate the Scroll knob to select the desired parameter, and then press the Accept key.
3. To change the value in a pop-up button, proceed as follows:
 - ◆ Tap the pop-up button twice, and enter the value using the virtual keyboard. You can also press the Accept key twice to open the virtual keyboard.

OR

Tap the pop-up button once, and then rotate the Scroll knob clockwise to increment, or counterclockwise to decrement the parameter value by the displayed increment/decrement step. If needed, change the displayed increment/decrement step using the Raise increment or Lower increment keys. After the knob has been used, you cannot display the virtual keyboard.

4. To abort editing of the value, press the Cancel key.
5. To accept the edited value, proceed as follows:
 - ◆ Press the Accept key.

OR

If you used the Scroll knob to edit the value, press another function key.

This approves the edited value and activates the associated software function.

OR

Press anywhere on the layout to close the virtual keyboard/pop-up button and accept the value.

1.12 Touch Screen

This section explains how to use the touch screen functions.

NOTE

You can enable/disable the touch screen by selecting **Preferences > Instrument > Category = Screen and Touch Screen = On/Off**.

1.12.1 Entering Values on the Software Keyboard

You can use the virtual keyboard to enter numerical parameter values.

To enter numerical parameter values using a virtual keyboard

1. Tap the parameter to be modified twice in order to use the virtual keyboard.
2. Tap the # button to display the numerical keypad (see Figure 1-13 on page 23).
3. Enter the value, and then tap the Accept button on the virtual keyboard.

OR

Rotate the Scroll knob clockwise or counterclockwise to increase or decrease the value.

OR

Use the Up and Down arrows to increase or decrease the value in the predefined increments.

4. To erase the character to the left of the cursor, tap the Delete button on the virtual keyboard.
5. Depending on your OmniScan instrument model, proceed as follows:

MX2	SX
<ul style="list-style-type: none"> ◆ To change the incrementation, press the Raise increment or Lower increment keys. 	<ul style="list-style-type: none"> ◆ To change the incrementation: <ol style="list-style-type: none"> a) Tap the parameter twice, or press the Accept key twice to open the software keyboard. b) Use the $+\Delta$ or $-\Delta$ buttons to increase or decrease the increment (select from the predefined list of increments).

6. To accept the edited value, press the Accept key on the OmniScan instrument or the Accept button on the virtual keyboard.
7. To revert to the previous value, press the Cancel key on the OmniScan instrument or the Cancel button on the virtual keyboard.



Figure 1-13 Virtual keyboard with numeric keypad (*left*) or without (*right*)

TIP

You can move the virtual keyboard to any location on the screen. To do so, tap and hold the virtual keyboard's interface, and drag it to the desired location.

1.12.2 Using the Touch Screen

This section describes how to use the touch screen.

IMPORTANT

In some cases, gate or cursor selection zones overlap (see Figure 1-14 on page 24). If you attempt to select a cursor or gate at the point where they overlap, they will be selected in this order of priority: Reference cursor, Measurement cursor, Data cursor, gate A, gate B, and gate I.

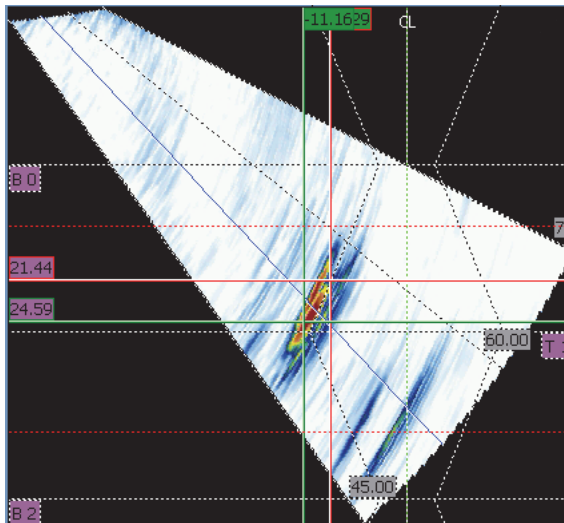


Figure 1-14 Example of selection zone overlaps

To use the zoom

The zoom starting point must be in a zone that does not contain any elements (gates or cursors).

1. To zoom in a zone, drag diagonally.
2. To zoom in the vertical axis, move your finger vertically.
3. To zoom in the horizontal axis, move your finger horizontally.

To pan within a zoomed view

- ◆ Tap the scroll box for the axis you want to pan, and then use the knob to move the zoomed area.

To use the cursors

1. To display the cursor pop-up button, tap any cursor.
2. To change the active cursor, tap the applicable cursor.
3. To move the last cursor used, double-tap any view.

To use the gates

1. To open the gate **Start** parameter pop-up button in edit mode, tap the left end of the gate.
2. To open the gate **Threshold** parameter pop-up button in edit mode, tap the middle of the gate.
3. To open the gate **Width** parameter pop-up button in edit mode, tap the right end of the gate.

NOTE

When a gate is short, it might be impossible to tap a specific zone. A visual reference indicates which sections of the gate are available (see Figure 1-15 on page 26).

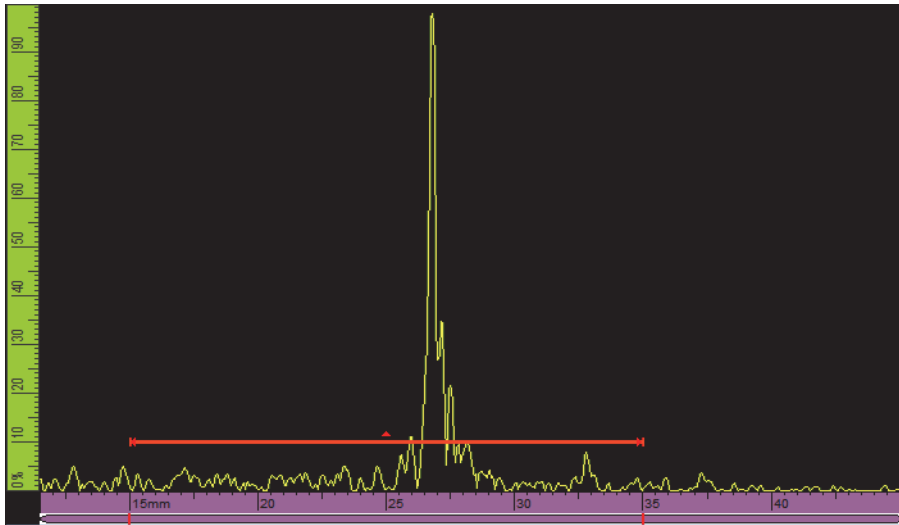


Figure 1-15 The visual reference on the gate

1.13 Indicators

There are three types of indicators on the OmniScan instrument front panel (see Figure 1-1 on page 8 for their location):

- Power indicator
- Acquisition indicator
- Alarm indicator

1.13.1 Power Indicator Light

The power indicator light is located above the Power key (see Figure 1-16 on page 27). The power indicator color identifies the power status of the OmniScan instrument (see Table 4 on page 27).

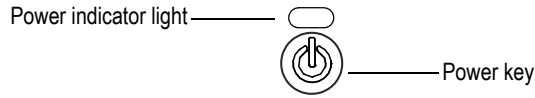


Figure 1-16 The power indicator

Table 4 OmniScan power indicator meaning

Power indicator color	Power status	Description
Off	Off	Normal
Green	On	<ul style="list-style-type: none"> • Normal • Battery is charging.
Flashing orange	Off	Battery is charging.
Orange	Off	Battery charge complete
Flashing red	On/Off	Critical factor (excessive temperature, very weak battery, etc.)

1.13.2 Acquisition Indicator Light

The acquisition indicator light is located below the Play key (see Figure 1-17 on page 28 or Figure 1-18 on page 28). The acquisition indicator color identifies the acquisition mode of the OmniScan instrument (see Table 5 on page 29).

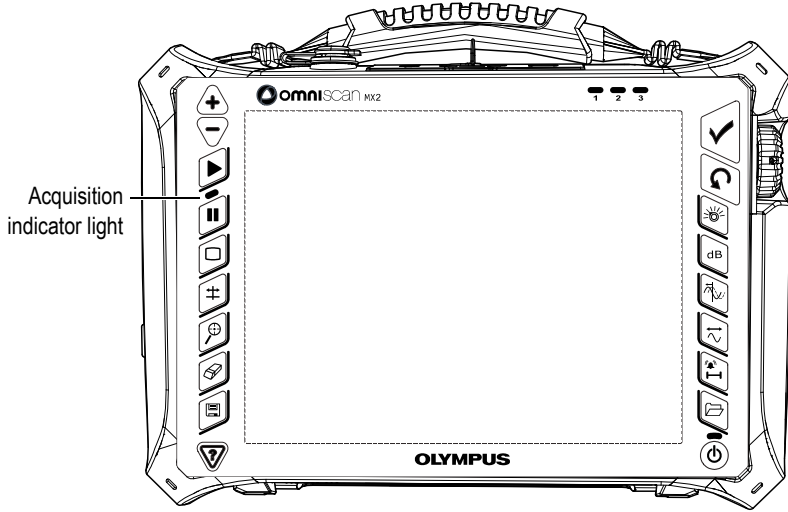


Figure 1-17 OmniScan MX2 acquisition indicator light

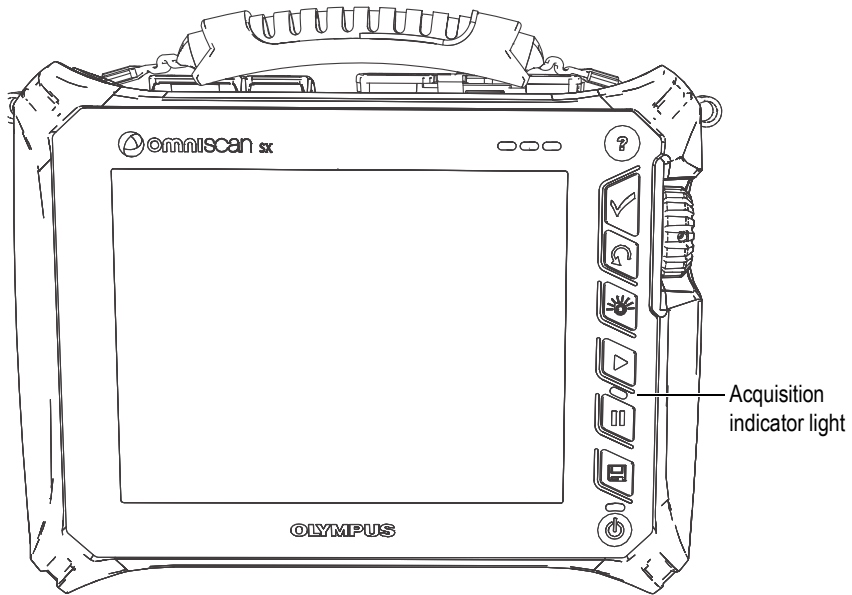


Figure 1-18 OmniScan SX acquisition indicator light

Table 5 Acquisition indicator meaning

Acquisition indicator color	Acquisition status	Acquisition mode
Off	Continuous acquisition (similar to an oscilloscope display)	Inspection mode
Orange	<ul style="list-style-type: none"> Paused acquisition Previously acquired data is available for analysis. 	Analysis mode

1.13.3 Alarm Indicators

The three alarm indicators numbered 1, 2, and 3 are located in the top right corner of the front panel (see Figure 1-1 on page 8 or Figure 1-2 on page 9). An alarm indicator turns red when the associated alarm condition is encountered. See “Gates/Alarms Menu” on page 295 for more information on alarms.

1.14 Navigating in the Menu Using Hardware Controls

The OmniScan instrument offers several different methods for navigating through the software menu, submenu, and parameter button hierarchy. There are several ways of entering or editing parameter values, and opening shortcut menus. The OmniScan instrument features a touch screen, from which you can select a button by simply pressing it directly on the screen. You can also use the front panel controls, the external USB keyboard, and the mouse (see Table 6 on page 29 to Table 12 on page 32).

Table 6 Choosing a menu from the menu list

Method	Action
Touch screen	Tap the Menu button, and then select the desired menu in the list.

Table 6 Choosing a menu from the menu list (continued)

Method	Action
Main controls	If you are not at the menu level, press the Cancel key repeatedly until the menu list is displayed. Use the Scroll knob to choose the desired menu, and then press the Accept key.
USB keyboard	Press ESC repeatedly until the menu list is displayed. Use the arrow keys to choose the desired menu, and then press ENTER.
Mouse	Click the Menu button to display the menu list. Click the desired menu in the list.

Table 7 Choosing a submenu from a menu

Method	Action
Touch screen	Tap the desired submenu button.
Main controls	Use the Scroll knob to choose the desired submenu, and then press the Accept key.
USB keyboard	Use the arrow keys to choose the desired submenu, and then press ENTER.
Mouse	Click the desired submenu button.

Table 8 Choosing a parameter from a submenu

Method	Action
Touch screen	Tap the desired parameter button.
Main controls	Use the Scroll knob to choose the desired parameter, and then press the Accept key.
USB keyboard	Use the arrow keys to choose a parameter, and then press ENTER.
Mouse	Click the desired parameter button.

Table 9 Choosing a value from a parameter list

Method	Action
Touch screen	Tap the desired value.
Main controls	Use the Scroll knob to choose the desired value, and then press the Accept key.
Increment keys (OmniScan MX2 only)	Use either the Raise increment or Lower increment key to select the desired value. Press the Accept key to accept the selection.
USB keyboard	Use the arrow keys to choose the desired value, and then press ENTER.
Mouse	Click the desired value.

Table 10 Going back one level or canceling a selection

Method	Action
Main controls	Press the Cancel key.
USB keyboard	Press ESC.

Table 11 Entering a value in an edit field

Method	Action
Main controls	See “Editing Values with the Main Controls” on page 15.
Increment keys (OmniScan MX2 only)	Use the Raise increment and Lower increment keys to change the increment.
USB keyboard	Use the alphanumeric keys to enter the desired information, and then press ENTER to leave the edit field. To leave the edit field without making any changes, press ESC.

Table 11 Entering a value in an edit field (continued)

Method	Action
Mouse	The mouse can only be used to select the edit field. To enter data, you must use another method.

Table 12 Opening a shortcut menu

Method	Action
Touch screen	Tap and hold the appropriate area (see “Gain” on page 50)
Mouse	Right-click the appropriate area.

IMPORTANT

The primary input method documented in this manual is the touch-screen method.

2. Software User Interface

This chapter presents the general features of the OmniScan MXU software user interface. A general overview is provided in “Main Features of the Interface” on page 44. The shortcut menus that are available in full-screen mode are detailed in “Shortcut Menus” on page 34.

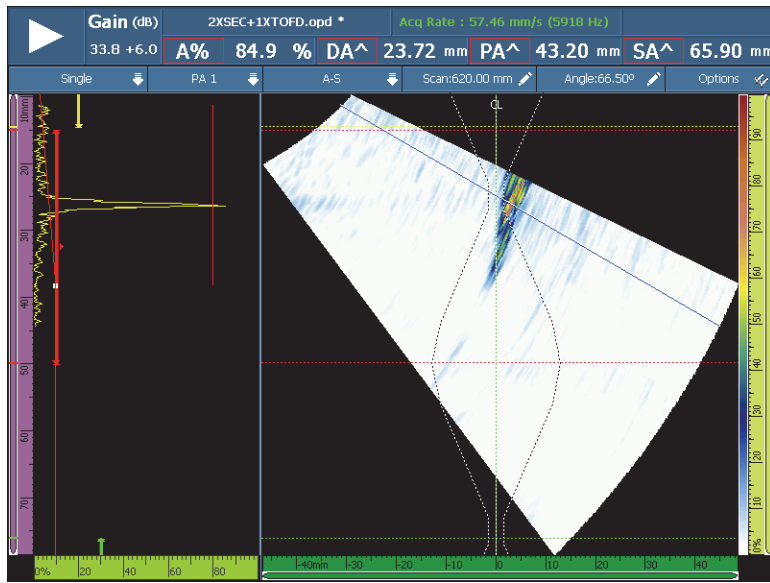


Figure 2-1 Example screen image of the user interface

2.1 Shortcut Menus

You can use the shortcut menus to quickly access the software's main functions, even when the full-screen mode is enabled.

2.1.1 Title Bar Shortcut Menus

You can use the title bar shortcut menus (see Figure 2-2 on page 34) as follows:

- Switch from menu-screen mode to full-screen mode, and vice versa
- Display the current group (**Single**) or all the groups (**Multiple**) simultaneously (for OmniScan MX2 only)
- Display one of the available layouts, which vary according to the selected technology
- Select the Scan, Index, and Angle data cursors
- Display or hide different options
- Manage defect indications
- Modify encoders or data source options
- Modify the scan and setup configuration

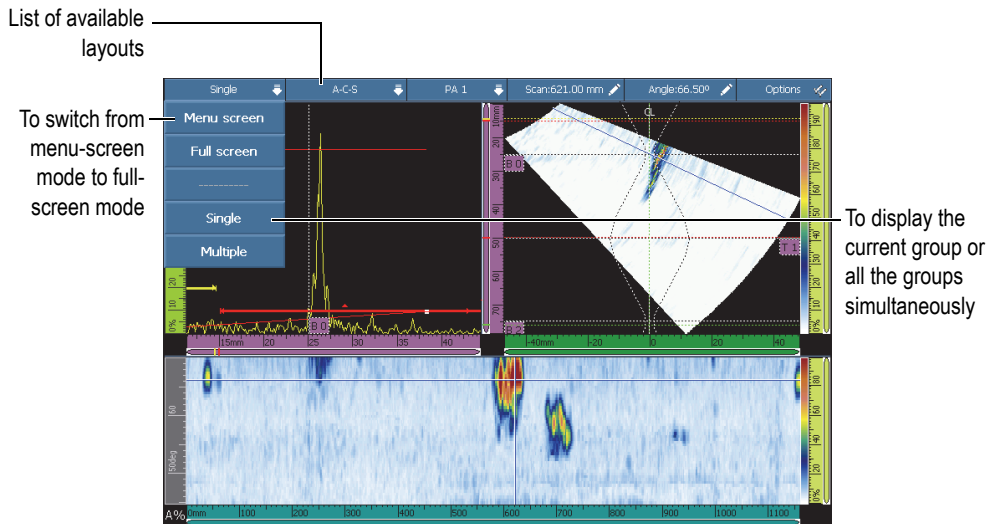


Figure 2-2 The title bar shortcut menus

2.1.1.1 Menu-Screen and Full-Screen Modes

To toggle between menu-screen and full-screen modes

1. Tap the Mode section of the title bar (see Figure 2-2 on page 34, near top left).
The title bar shortcut menu appears.
2. To enable the full-screen mode, select **Full Screen**.
3. To go back to the menu-screen mode, tap the title bar, and then select **Menu Screen**.

NOTE

You can also use the increment keys to turn on or off the full-screen mode when not in edit mode (OmniScan MX2 only).

2.1.1.2 Single or Multiple Group Display (OmniScan MX2 Only)

The OmniScan MXU software allows you to display the current group (**Single**), or multiple groups (**Multiple**) simultaneously.

The multiple group display function supports combinations of up to 8 groups. However, if you are using phased array, a maximum of 6 groups of either the linear or sector scan are supported. If you are using TOFD and conventional UT, a maximum of 4 groups are supported, either from the UT or PA connector.

To toggle between the Single and Multiple group displays

1. Tap the Mode section of the title bar.
The title bar shortcut menu appears.
2. To display the current group, select **Single**.
3. To display multiple groups, select **Multiple**.

NOTE

- When the current group is displayed (**Single**), both menu-screen and full-screen modes are available.

- When multiple groups are displayed (**Multiple**), the full-screen mode is automatically enabled.
-

2.1.1.3 Layout Selection

Your layout selection combines the most useful views.

NOTE

For more details about layouts and views, see “Scans, Views, and Layouts” on page 57.

To select a layout using the title bar shortcut menu

1. Tap the Layouts section of the title bar.
2. On the title bar shortcut menu, select the layout(s) to be displayed.

When one group is displayed, the layout combinations can include the following views:

- A (A-scan)
- B (B-scan)
- C (C-scan)
- D (D-scan)
- S (S-scan)
- E (End view)
- R (RayTracing)

When multiple groups are displayed (OmniScan MX2 only), the following layouts are possible:

- **A-C-End (PA Merged)**: Displays the C-scans of PA groups, End view of all groups merged together, and A-scan of the currently selected group.
 - **A-C-S (PA)**: Displays the S-scans and C-scans of PA groups and the A-Scan of the currently selected group.
 - **A-B (UT)** Displays the A-scans and B-scans of UT/TOFD groups.
-

- **A-B-S (PA-UT):** Displays the S-scans of PA groups, B-scans of UT/TOFD groups, and A-Scan of the currently selected group.
 - **A-B-C (PA-UT Scrolling):** Displays the C-scans of PA groups, B-scans of UT/TOFD groups, and A-scan of the currently selected group.
 - **A-S (PA):** Displays the S-scans and the A-scan of the currently selected group.
-

NOTE

Groups are displayed in the layout according to the actual scanner configuration and the offsets.

2.1.1.4 Group Selection Section (OmniScan MX2 Only)

To select a group

1. Tap the Group section of the title bar (see Figure 2-2 on page 34).
A list of available groups appears.
 2. Select the desired group from the list.
-

TIP

If you have a multiple group setup, you may want to rename the groups as follows:

1. Tap and hold the Group section of the title bar.
 2. Enter the desired name with the virtual keyboard.
-

2.1.1.5 Scan Section

To adjust the Scan data cursor position

1. Tap the **Scan** section of the title bar (see Figure 2-2 on page 34).
2. Use the Scroll knob to adjust the **Scan** data cursor position.

To quickly access the scan encoder options

- ◆ Tap and hold the **Scan** section of the title bar.
A list of options is displayed (see “Encoder Submenu” on page 320 for more details).

To quickly access the Scan Configuration menu

1. Tap and hold the **Scan** section of the title bar.
2. Select **Scan Configuration**.
For more details, see “Scan Configuration Menu” on page 339.

2.1.1.6 Index Section

To adjust the Index data cursor position

1. Tap the **Index** section of the title bar (see Figure 2-2 on page 34).
2. Use the Scroll knob to adjust the **Index** data cursor position.

To quickly access the index encoder options

- ◆ Tap and hold the **Index** section of the title bar.
A list of options is displayed (see “Encoder Submenu” on page 320 for more details).

To quickly access the Scan Configuration menu

1. Tap and hold the **Index** section of the title bar.
2. Select **Scan Configuration**.
For more details, see “Scan Configuration Menu” on page 339.

2.1.1.7 Angle/VPA Section

To adjust the Angle/VPA data cursor

1. Tap the Angle/VPA section of the title bar (see Figure 2-2 on page 34).
2. Use the Scroll knob to adjust the Angle/VPA cursor position.

To quickly change the A-scan data source

1. Tap and hold the Angle/VPA section of the title bar.
2. Select either the Normal or Highest (%) data source (see “A-Scan Settings Submenu” on page 280 for more information).

To quickly access the Setup Configuration menu

1. Tap and hold the Angle/VPA section of the title bar.
2. Select **Setup Configuration**.
For more details, see “Setup Configuration Menu” on page 336.

2.1.1.8 Display/Hide Options

To display or hide options

1. Tap the **Options** section of the title bar (see Figure 2-2 on page 34).
A list of available options appears.
2. Select the options to be displayed.

2.1.2 Display Area Shortcut Menu

The display area shortcut menu is used to quickly access the parameters linked to the current view.

To access the display area shortcut menu

- ◆ Tap and hold the display area.

The following parameters are available in all display area shortcut menus:

- **Set Data Cursor**
- **Set Reference Cursor**
- **Set Measure Cursor**

The following parameters are available for the A-scan:

- **Enable/Disable Envelope**
- **Clear Envelope**
- **Wedge Delay & PCS (TOFD only)**

- **Lateral Synchro** (TOFD only)
- **Reset Lateral Synchro** (TOFD only)

The following parameters are available for the B-scan:

- **Scan Offset**
- **Wedge Delay & PCS** (TOFD only)
- **Export** (OmniPC only)
- **Export All** (OmniPC only)
- **Lateral Synchro** (analysis mode, TOFD only)
- **Reset Lateral Synchro** (analysis mode, TOFD only)

NOTE

After a calibration has been performed, the **Wedge Delay & PCS** parameter (which appears in the A-scan, B-scan, and ultrasound axis shortcut menus) takes the same name as the last (previous) calibration that was performed (**Wedge Delay, WD & PCS**, or **Velocity & WD**).

The following parameters are available for the C-scan:

- **Scan Offset** parameter
- C-scan source selection
- Thickness source selection
- **Export** (analysis mode)

The following parameters are available for the S-scan:

- **Index Offset**
- **Skew Left (90°)**
- **Skew Right (270°)**

2.1.3 Shortcut Menus in the Reading-Field Area

You can use the readings shortcut menu for the following actions:

- Select a reading list
- Modify a particular reading
- Modify gain parameters

- Manage indicators
- Manage files

To use the readings shortcut menu

1. Tap and hold the reading-field area.
The readings shortcut menu appears (see Figure 2-3 on page 41).



Figure 2-3 The readings shortcut menu

2. To select a reading list, proceed as follows:
 - a) On the readings shortcut menu, choose **Select Reading List**.
 - b) From the list on the left-hand side of the mode selection dialog box, select the required measurement mode, and then choose **Select**.
3. To select and modify one reading field, proceed as follows:
 - a) In the reading-field area, tap and hold the reading to be modified.
 - b) On the readings shortcut menu, choose **Select Reading**.
 - c) From the list on the left-hand side of the dialog box, select the desired reading category, and then select the desired reading from the list on the right-hand side of the dialog box.
 - d) Choose **Select**.
4. To add an indication, proceed as follows:
 - a) In the reading-field area, tap and hold any reading.
 - b) Select **Add Indication**.
5. To delete an indication, proceed as follows:
 - a) In the reading-field area, tap and hold any reading.
 - b) Select **Delete Last indication**.

To use the File Manager shortcut menu

1. Tap and hold the **File Name** area.

The File Manager shortcut menu appears (see Figure 2-4 on page 42).

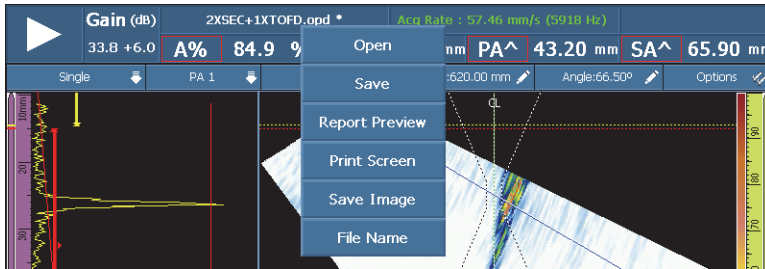


Figure 2-4 The File Manager shortcut menu

2. To load a setup (inspection mode), to load a data file (analysis mode), to preview the report, or for any other File Manager option, tap and hold the file name, and then make your selection using the File Manager.

In acquisition mode, the available choices are: **Open**, **Save As** (save your setup), **Report Preview**, **Print Screen**, **Save Image**, and **File Name**.

In Analysis mode, the available choices are: **Open**, **Save** (save your data), **Report Preview**, **Print Screen**, **Save Image**, and **File Name**.

NOTE

Print Screen is used to capture the entire screen. **Save Image** only saves the data screen (display) area, and is equivalent to **File > Image > Save**.

To use the System shortcut menu

1. Tap and hold the MXU version area of the display.

The System shortcut menu appears (see Figure 2-5 on page 43).

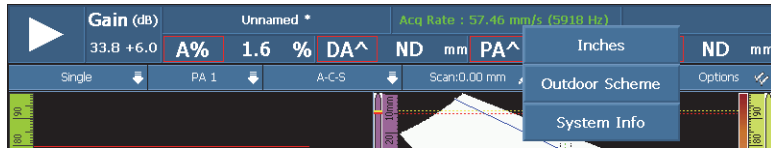


Figure 2-5 The System shortcut menu

- To modify system parameters, select an item in the System shortcut menu list. The available choices are **Inches/Millimeters**, **Indoor Scheme/Outdoor Scheme**, and **System Info**.

2.1.4 Adjusting the Acquisition Parameters

The acquisition parameters shortcut menu is used to pause the acquisition, reset the scan, adjust the gain, or adjust the acquisition rate.

To adjust the acquisition parameters

- To start or pause the acquisition, tap the Play/Pause button (see Figure 2-6 on page 43, on the left).

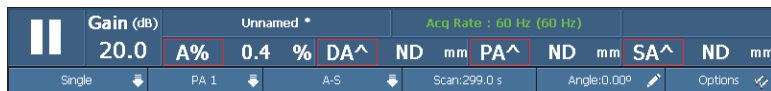


Figure 2-6 The acquisition parameters

- To reset the scan, tap and hold the Play/Pause button. The available options are **Reset All**, **Reset Data**, and **Reset Encoder**.
- To adjust the **Gain** value, tap the **Gain** button during acquisition.
- To select from the available options **Enable/Disable Reference** and **Set to Reference**, tap and hold the **Gain** button during acquisition.
- To adjust the attenuation gain, tap the **Gain** button during analysis.
- To adjust the acquisition rate value, tap the **Acq Rate** button.

2.2 Main Features of the Interface

Figure 2-7 on page 44 shows the OmniScan MXU software user interface, and identifies its main components.

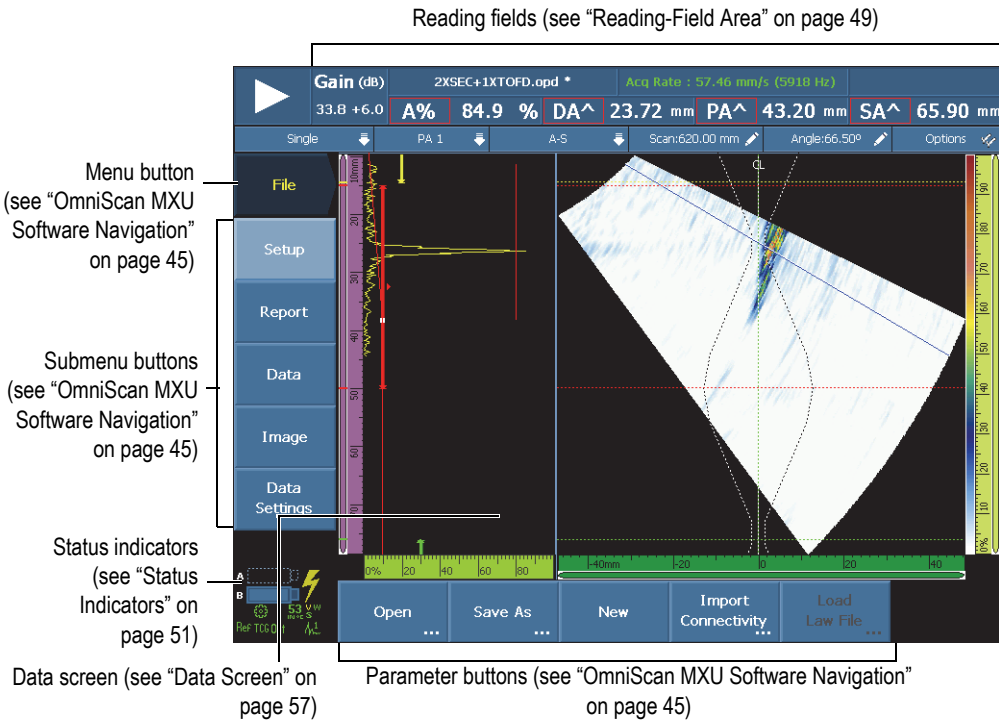


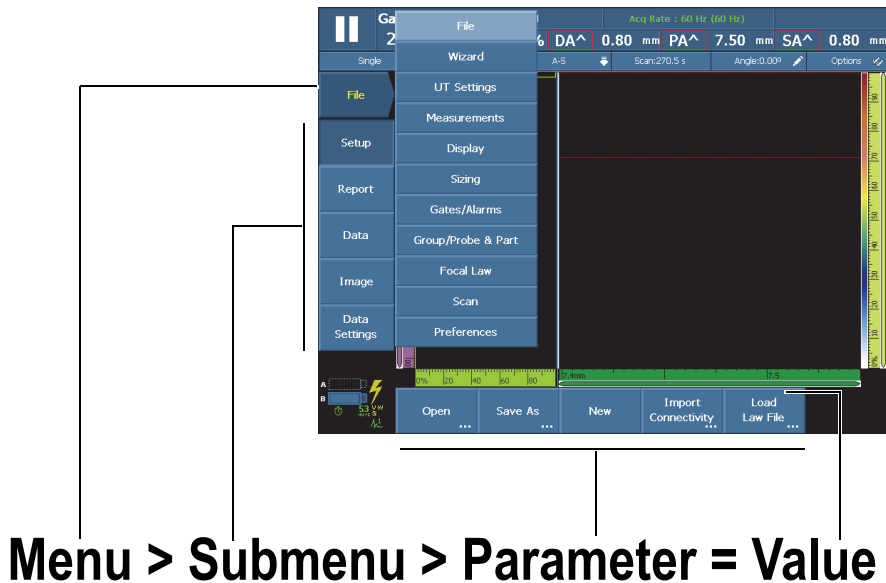
Figure 2-7 The OmniScan MXU software window components

NOTE

In this manual, the OmniScan MXU software screen images are displayed using the default color scheme, which is designed for indoor use. However, an alternative color scheme is available for outdoor operation. It consists of a white or pale-color background with black or dark-colored text. Select **Preferences > Instrument > Category = Screen**, and then **Scheme** to select the desired color scheme.

2.3 OmniScan MXU Software Navigation

Figure 2-8 on page 45 shows the three menu levels of the OmniScan MXU software, and describes the syntax used throughout this manual to systematically select the menu, submenu, and parameter buttons and to optionally enter or select a parameter value. For example, **Gates/Alarms > Gates > Start** signifies that you first select the **Gates/Alarms** menu, then select the **Gates** submenu, and finally select the **Start** parameter.



Note: see “Navigating in the OmniScan MXU Software” on page 46 for the complete procedure.

Figure 2-8 The menu hierarchy and identification syntax

In the menu hierarchy, if a button has a pale background, it is selected, and the focus is on that particular menu. The menu list temporarily appears vertically over the data screen area while the focus is on the menu level. The submenu buttons appear vertically on the left side of the screen, below the menu button. The parameter buttons appear at the bottom of the screen.

2.3.1 Navigating in the OmniScan MXU Software

The following procedure provides a detailed description on how to select menu, submenu, and parameter buttons in the user interface of the OmniScan MXU software.

To navigate in the OmniScan MXU software

1. Choose the desired Menu button.

The menu list appears to the right of the button you have chosen (see Figure 2-8 on page 45).

2. Proceed as follows to choose the desired item from the menu list:

◆ Tap the desired item on the touch screen.

OR

Rotate the Scroll knob, and then press the Accept key.

OR

Use the Raise increment or Lower increment key to select the desired item, and then press the Accept key to accept the selection (OmniScan MX2 only).

OR

With a USB mouse, click the desired menu item.

The software immediately reassigns the functions of the submenu buttons to those corresponding to the chosen menu item.

3. Choose the desired submenu button using the method in step 2.

The software immediately reassigns the functions of the parameter buttons to those corresponding to the chosen submenu button.

4. Choose the desired parameter button using the method in step 2.

5. To go back one level in the menu, submenu, parameter, and value hierarchy, press the Cancel key.

2.3.2 Menu Organization

The order in which the menu items, submenus, and parameter buttons appear corresponds with typical sequences during use. The menus can be organized into three groups: setup, configuration, and inspection (see Figure 2-9 on page 47).

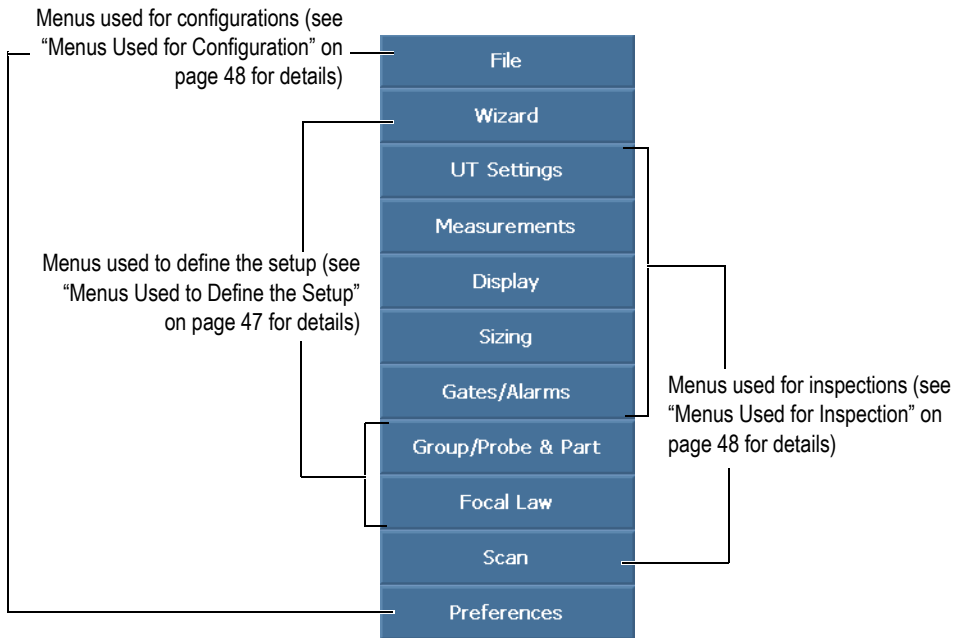


Figure 2-9 Menu item groups

2.3.2.1 Menus Used to Define the Setup

The following menus are used to define the setup (see "Setup Procedures" on page 91 for details):

Wizard

Use this menu to create a complete application setup. The step-by-step approach ensures that you enter or change all relevant parameter values. The help screens provide specific information on how to adjust the parameters. The menu includes a complete step-by-step calibration wizard, which is an essential part of creating a setup.

Group/Probe & Part

Use this menu to define one group or several groups (OmniScan MX2 only), and to fine-tune the parameters related to the probe or the part previously defined with the **Setup** wizard. See "Group/Probe & Part Menu" on page 306 for a complete description.

Focal Law

Use this menu to fine-tune the parameters related to the focal laws originally defined with the **Focal Law** wizard. See “Focal Law Menu (PA Group Only)” on page 315 for a complete description.

2.3.2.2 Menus Used for Inspection

The following menus are used during inspections (see “Inspection Procedures” on page 151 for details on use):

UT Settings

This menu contains the parameters regularly modified during inspection, such as the gain and pulser/receiver parameters (see “UT Settings Menu” on page 244 for a complete description).

Measurements

This menu contains the parameters related to various measurement tools (see “Measurements Menu” on page 257 for a complete description).

Display

This menu contains the parameters related to the data views and the information visible on screen.

Sizing

This menu contains parameters for selecting and configuring a sizing function (DAC, Linear DAC, TCG, AWS, and DGS). See “Sizing Menu” on page 286 for a complete description.

Gates/Alarms

This menu contains parameters for configuring gates, alarms, and output signals (see “Gates/Alarms Menu” on page 295 for a complete description).

Scan

Use this menu to adjust the parameters related to the scan initially defined with the wizards. See “Scan Menu” on page 319 for a complete description.

2.3.2.3 Menus Used for Configuration

The following menus are used to configure the OmniScan MXU software and hardware (see “General Procedures” on page 79 for details):

File

This menu is used to open and save files and to format and build an inspection report (see “File Menu” on page 229 for a complete description).

Preferences

This menu contains various parameters related to instrument configuration that are typically set when you start using the instrument. Examples include the measurement unit (millimeters or inches) and the date and time (see “Preferences Menu” on page 326 for a complete description). It also includes an FFT utility to characterize your probe.

2.4 Reading-Field Area

The reading-field area is located at the top of the OmniScan MXU software user interface (see Figure 2-10 on page 49).

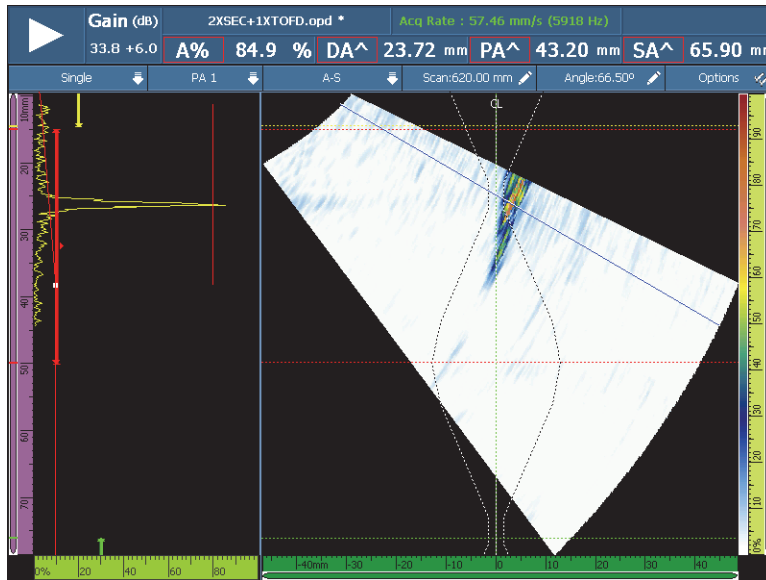


Figure 2-10 The reading fields and data screen

Figure 2-11 on page 50 identifies the various parameters and reading fields available, and lists sections in this manual that provide more information. Figure 2-12 on page 50 identifies the information available in a reading field.

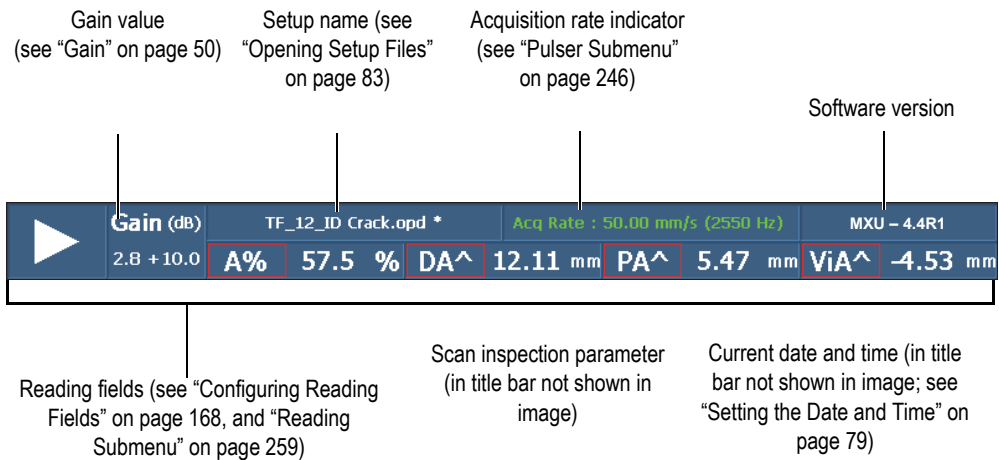


Figure 2-11 The parameters in the reading-field area

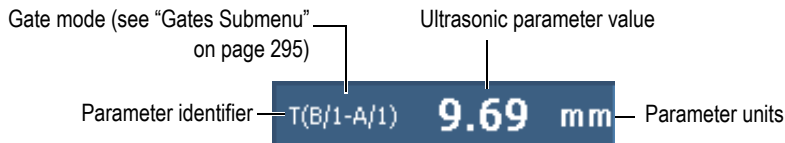


Figure 2-12 The information available in a reading field

2.5 Gain

The gain value that is applied to the signal is an important parameter. The gain value applied to all focal laws appears in the upper-left corner of the reading-field area (see Figure 2-11 on page 50). Figure 2-13 on page 51 shows the information displayed in the **Gain** value field.

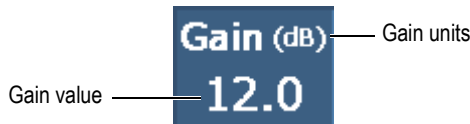


Figure 2-13 The Gain value field

The **Gain** value field presents two values after the **UT Settings > Advanced > Reference dB** parameter is turned **On** (see Figure 2-14 on page 51). Setting **Reference dB** to **On** freezes the current gain value as the reference gain. An adjustment gain value appears in order to show gain value changes. With an active reference, the gain applied to all focal laws is the total of the reference gain plus the adjustment gain. See “Advanced Submenu” on page 255 for more information on the **Reference dB** function.

In analysis mode, an attenuation gain parameter is available for adjustment. This parameter allows you to adjust the signal amplitude after the acquisition has been performed. The adjustment range is 0 dB to -10 dB.

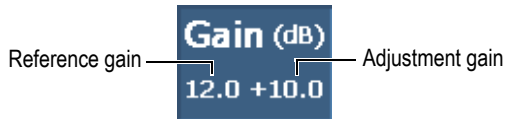


Figure 2-14 The Gain value field with an active reference

2.6 Status Indicators

The status indicators show the current status of the OmniScan instrument. They are located in the bottom-left corner of the screen in menu-screen mode.

Figure 2-15 on page 52 shows an example of the status indicators.

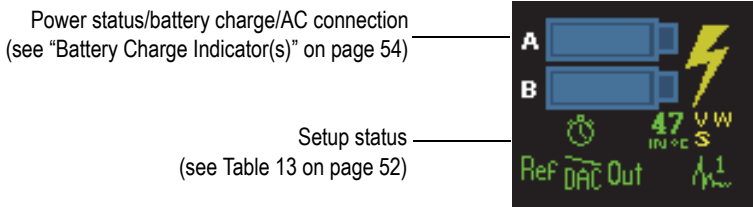


Figure 2-15 An example of the status indicators

Table 13 on page 52 provides a list of the status indicators and their meanings.

Table 13 The status indicators and their meanings


























Indicator	Meaning
	Normal A-scan mode is enabled.
	All focal laws mode is enabled as follows: Display > A-Scan Settings > Data Source = All Laws (see "Data Source" on page 281).
	Tracking A-scan mode is enabled as follows: Display > A-Scan Settings > Data Source = Highest or Thinnest (see "Data Source" on page 281).
	The data in gate A is the skip number found in the symbol.
	The C-scan data is located farther than the fourth skip.
	The distance-amplitude correction (DAC) curve is enabled (see "Type Submenu" on page 287).
	The linear distance-amplitude correction (DAC) curve is enabled (see "Type Submenu" on page 287).
	The number of data points on the inspected area exceeds the number of available pixels (see "Compression Feature" on page 75).
	The time-corrected gain (TCG) is enabled (see "Type Submenu" on page 287).

Table 13 The status indicators and their meanings (continued)

Indicator	Meaning
	The time-corrected gain (TCG) is enabled, but it exceeds the dynamic gain range (see “Parameters for DAC and TCG Curves” on page 294).
	The distance-gain sizing (DGS) is enabled (see “Type Submenu” on page 287).
	Reference mode is enabled.
	The acquisition synchronization is set to clock mode.
	The acquisition synchronization is set to encoder mode.
	This indicator displays the OmniScan instrument’s internal temperature in degrees Celsius.
	The sensitivity is not calibrated.
	The sensitivity is calibrated.
	The sensitivity is calibrated for two or three selected angles/VPA, and interpolated in between the calibrated angles/VPA.
	The TOFD is not calibrated.
	The TOFD is calibrated.
	The ultrasonic velocity is not calibrated.
	The ultrasonic velocity is calibrated.
	The wedge delay is not calibrated.
	The wedge delay is calibrated.
	The wedge delay is calibrated for two or three selected angles/VPA, and interpolated in between the calibrated angles/VPA.

2.7 Battery Charge Indicator(s)

There are two battery charge indicators for the OmniScan MX2. The OmniScan SX has a single indicator. The indicators show the amount of power remaining in the battery or batteries (see Figure 2-16 on page 54 or Figure 2-17 on page 54).



Figure 2-16 OmniScan MX2 battery charge status example



Figure 2-17 OmniScan SX battery charge status example








The remaining estimated operation time is displayed inside the battery charge indicator in the hh:mm format (hour:minute).

The charge indicator (bar) inside the battery charge indicator displays the approximate amount of power remaining in that battery. Table 14 on page 55 describes the various states of the battery charge indicator.

NOTE

- If you attempt to turn the OmniScan instrument on with one or two batteries that are too low for operation, the power indicator light blinks red rapidly for about three seconds. Replace the battery or batteries, or plug in the DC power adaptor to operate the OmniScan instrument.
 - The OmniScan MX2 uses one battery at a time; the battery currently in use is highlighted.
-

Table 14 Battery charge indicator variations

Indicator	Outline	Fill	Meaning
	Dotted	N/A	There is no battery in that compartment.
	Blue	Blue	The battery is functioning properly.
	Blue	Orange	The battery is too hot for operation.
	Yellow (blinking)	Blue	The battery is charging.
	Orange	Blue	The battery is too hot to be charged or the internal temperature of the instrument is too high to enable the charge (over 45 °C).
	Red (blinking)	Pale-blue	The battery charge is critically low (less than 10 %). A beeping sound is heard if no other battery is available.
	N/A	Yellow	The OmniScan instrument is being powered externally through the DC power adaptor.

2.8 Acquisition Modes

The OmniScan instrument features two acquisition modes: inspection mode and analysis mode. Figure 2-18 on page 56 illustrates the basic operations of each mode and how to switch between modes.

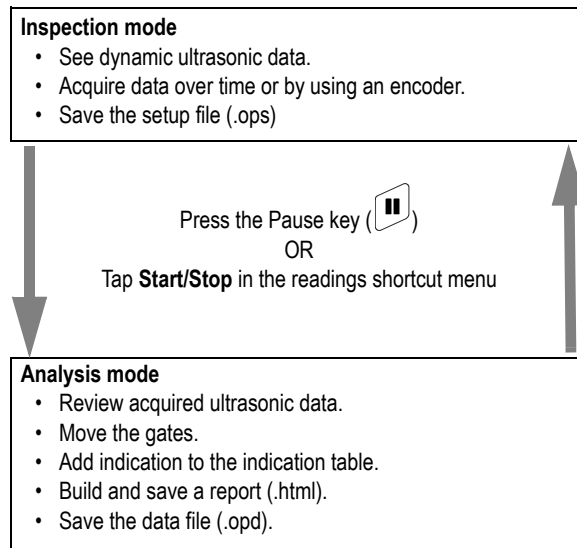


Figure 2-18 The acquisition modes

2.8.1 Inspection Mode

Inspection mode is the default acquisition mode when the OmniScan instrument is turned on. Inspection mode has the following characteristics:

- The instrument continuously produces ultrasonic beams, and dynamically displays ultrasonic data.
- Pressing the Play key (or tapping and holding **Start/Stop** in the readings shortcut menu and then selecting **Reset All**) starts data recording for a scanned area (using an encoder) or for a predetermined period of time.
- Pressing the Pause key (or tapping **Start/Stop** in the readings shortcut menu) pauses the data acquisition and activates analysis mode.

2.8.2 Analysis Mode

Analysis mode is used to analyze recorded data after an inspection. Analysis mode has the following characteristics:

- In this mode, the instrument stops acquiring data, and the recorded data is available for analysis.

- The acquisition indicator illuminates solid orange (see “Acquisition Indicator Light” on page 27 for details).

The typical sequence of operations to be performed in Analysis mode is as follows:

1. Place the gate over the desired indications.
2. Use the cursors and reading fields to measure the size and the position of the indications.
3. Add the indications to the indication table.
4. Produce and save the reports (.html).
5. Save the inspection data (.opd).
6. Return to inspection mode.

2.9 Data Screen

The data screen area displays the various ultrasonic data views and layouts.

2.9.1 Scans, Views, and Layouts

A scan is a 2-D graphical representation of ultrasonic data with a ruler or scale corresponding to the horizontal and vertical axis (see “Rulers/Scales” on page 65). For example, an A-scan and a C-scan are two different types of scans.

A view is a volumetric representation of a part, which includes signal overlays. Like a scan, a view has two axes. However, instead of being related to a specific group, a view is linked to the part. A signal that originates from a single group or from multiple groups can be displayed without affecting the view dimensions.

A layout is an organization of one or more scans and views. For example, the A-B-C layout is a graphical arrangement of an A-scan, a B-scan, and a C-scan.

TIP

Tap the Layouts section of the title bar to select the layout.

Table 15 on page 58 lists the basic ultrasonic views (scans), which are illustrated in Figure 2-19 on page 59.

Table 15 Basic data views

View	Point of view	Axis content
A-scan	Looking down on the material	Amplitude versus Ultrasound
B-scan	Side	Ultrasound versus Scan
C-scan	Top	Scan versus Index
D-scan, S-scan, End view, Ray Tracing	End	Ultrasound versus Index

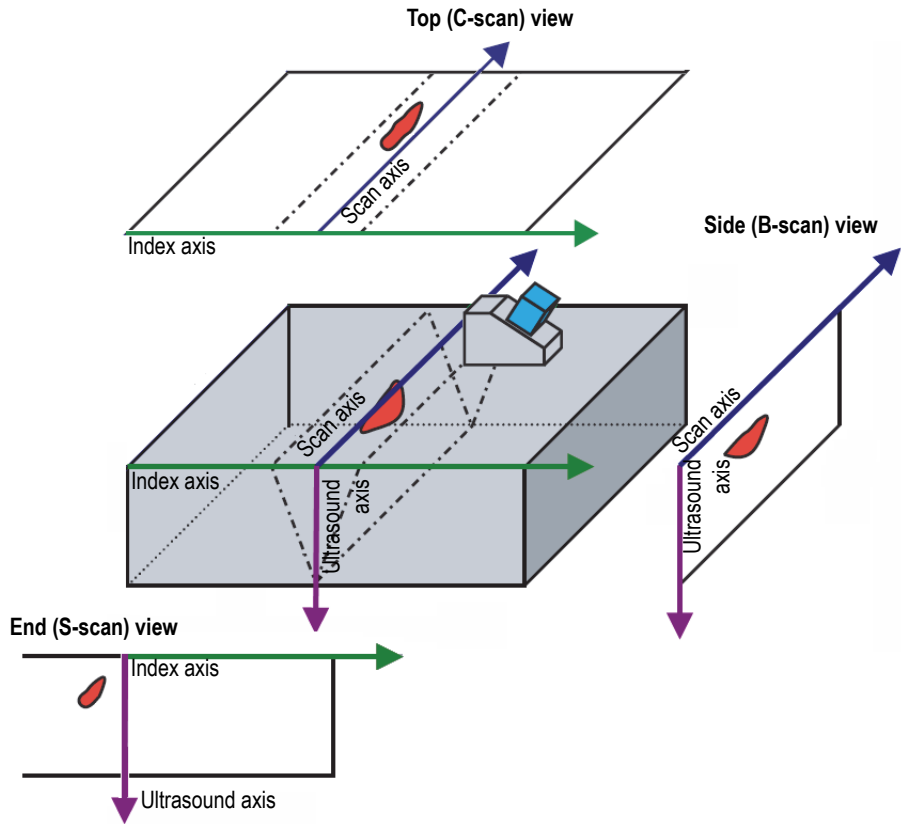


Figure 2-19 Example of ultrasonic scan views

The following scans and views are available in the layouts:

A-scan

The scan on which all other scans are based. An A-scan is a representation of the received ultrasonic pulse amplitude versus time-of-flight (ultrasonic path), or a waveform.

A peak in the signal corresponds to the echo of a reflector or a discontinuity in the part. Peaks at the start and at the end of the ultrasound axis generally correspond to the echoes of the part entry surface and the back wall.

B-scan

2-D side representation of the part showing ultrasonic data with the scan length on one axis, and the ultrasound path on the other axis.

C-scan

2-D top representation of the part showing gated ultrasonic data with the scan length on one axis, and the index length on the other axis. One of the available parameters (for example, the maximum amplitude) is projected on the index-scan plan for each point (pixel).

D-scan

2-D end representation of the part showing ultrasonic data with the index length on one axis, and the ultrasound path on the other axis.

S-scan (PA group only)

2-D representation of ultrasound data presenting all the A-scans generated by the focal laws in an angular sector or sweep range in order to create a cross section of the part. The A-scans are represented by lines on which the amplitude is color-coded, and they are corrected for delay and true depth so that their positions are accurate relative to the ultrasound axis.

End view

S-scan data view displayed with the ultrasound beam skips in the part. A single group or multiple groups can be represented simultaneously and will be merged. Improves the defect positioning in a weld inspection. The End view is intended only for flat parts, and therefore curved parts will be displayed as if they were flat parts.

RayTracing

Dynamic cross-section illustration of the propagation of the ultrasonic beams in the part.

2.9.2 RayTracing

The RayTracing view is a dynamic cross-section illustration of the part, the propagation of the ultrasound beams in the part, the limits of gate A, the location of the indications in the part, and the weld drawing. Figure 2-20 on page 61 presents the elements of a RayTracing illustration.

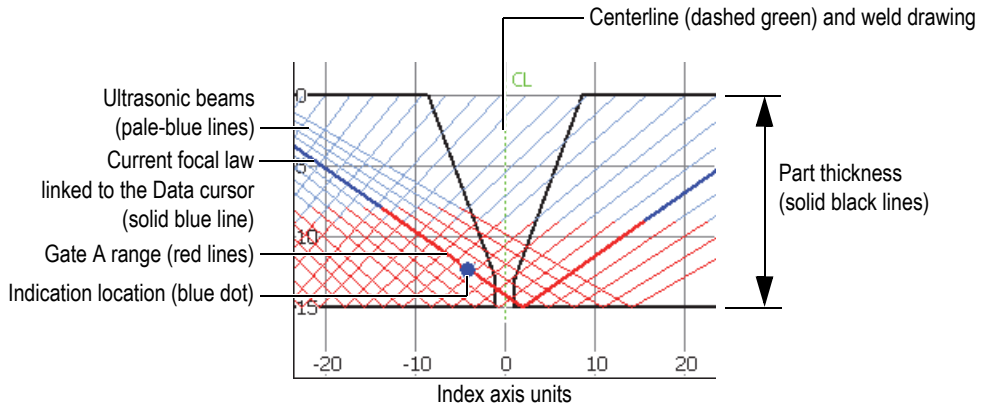


Figure 2-20 The RayTracing graphical elements

The thickness of the part is shown on the vertical axis, and the index axis is shown on the horizontal axis. Pale-blue lines represent the ultrasonic beams. A dark-blue line represents the path for the current focal law.

2.9.2.1 Using RayTracing to Define the Setup

Using the wizards allows you to quickly and easily define the setup. The RayTracing dynamic cross-section illustration is integrated into the **Setup** wizard in order to help you to visualize the changes in each configuration step of the setup.

In the wizards, the RayTracing illustration appears at the bottom of the data screen area (see Figure 2-21 on page 62).

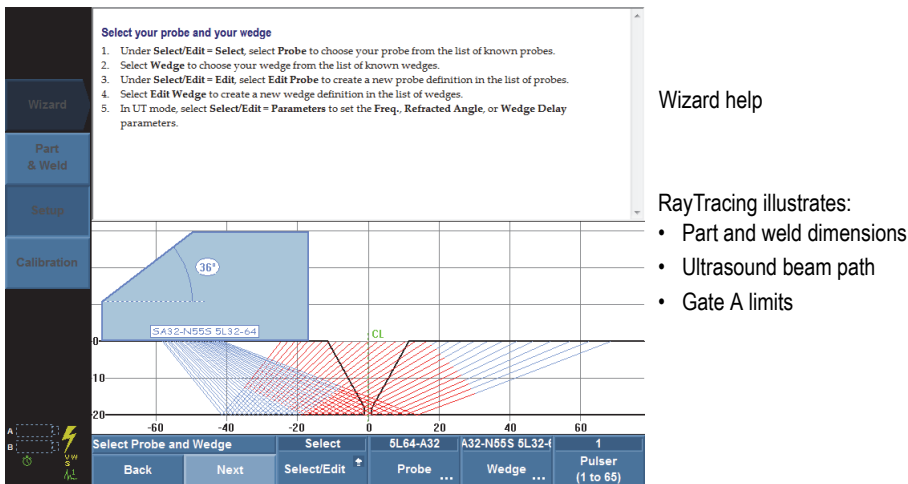


Figure 2-21 Example of RayTracing in the Setup wizard

2.9.2.2 Using RayTracing to Inspect a Part

The RayTracing view allows you to visualize a part, and dynamically highlight the strongest reflector as you perform an inspection. The A-C-R-S layout combines the A-scan, S-scan, RayTracing, and C-scan views. As you move the probe over the part, the RayTracing dynamically identifies the location in the part with the strongest signal crossing gate A. The RayTracing view displays a blue dot at the location of the strongest signal, and a solid-blue line for the associated law.

To perform an inspection using the RayTracing view

1. Use the **Part & Weld** and **Setup** wizards to define the setup (see “Creating the Application Setup with the Wizards” on page 91).
2. Select the **A-C-R-S** layout from the title bar.
3. Move the probe over your part until you find an indication.

2.9.2.3 Using RayTracing to Analyze the Indications

In analysis mode, the RayTracing view allows you to visualize a part and the indications as you analyze the inspection results. You can use the A-C-R-S layout and the indication table to review the indications entered in the indication table, and visualize their location in the inspected part on the RayTracing view.

To analyze the inspection using the RayTracing view and the indication table

1. Select the **A-C-R-S** layout from the title bar.
2. Activate the indication table from the **Options** menu in the title bar.
3. Perform the inspection, leaving the probe over a reflector of interest.
4. Enter the analysis mode by pressing the Pause key, or by selecting **Scan > Start > Pause = On**.

The data screen freezes.

5. Press the Data Selector key, and then select the **Index** and the **Scan** pop-up buttons. Position the Data cursors appropriately in the C-scan view using the Scroll knob (blue line on the C-scan) over an indication.
6. Select the **Angle** pop-up button, and then position the Data cursor on the most significant law of the S-scan view. When the signal of the current focal law crosses gate A, a blue dot linked to the Data cursor appears on the RayTracing view to show the location of the indication in the part.
7. Tap and hold the reading area and then select **Add Indication**.
The blue dot turns red in the RayTracing view.
8. If more than one indication appears in the C-scan view, repeat steps 5 to 7.

In the RayTracing view (see Figure 2-22 on page 64):

- A blue dot represents the indication corresponding to the current Data cursor position.
- A red dot represents the currently selected indication table entry.
- An X accompanied by a number in superscript represents an indication table entry. This number corresponds to the indication number in the table.

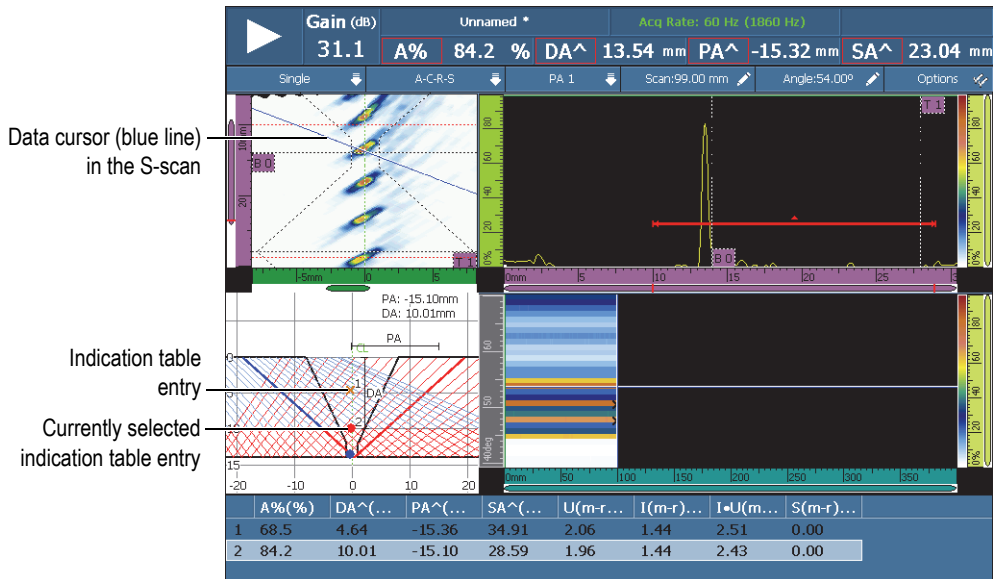


Figure 2-22 Example of indications in the RayTracing view

- To save the data, choose **File > Data > Save**.

TIP

Select **File > Data Settings > Save key = Data** to configure the Save/Print key to save inspection data. Pressing the Save/Print key is thus equivalent to selecting **File > Data > Save**.

- Press the Pause key to return to the inspection mode, and to continue the inspection.

2.9.2.4 Multiple Group Layouts in OmniPC

In the OmniPC software, Multiple Group layouts are available in the menu screen mode (see Figure 2-23 on page 65). This feature is not available on the OmniScan MX2 instrument.

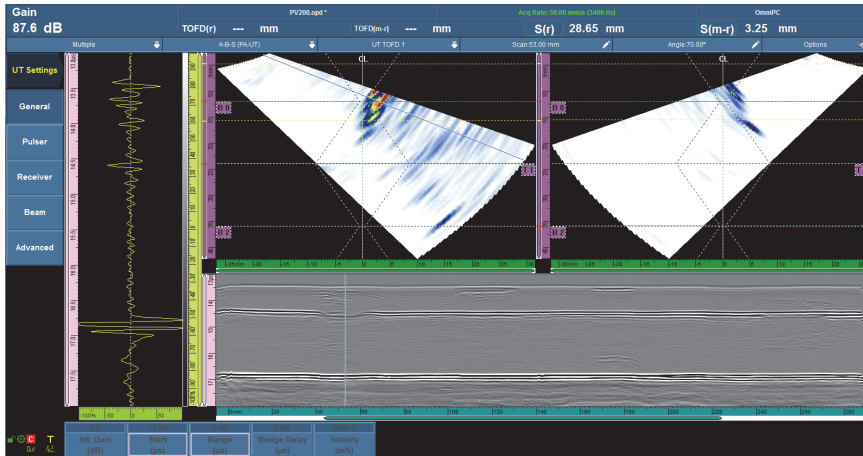


Figure 2-23 The Multiple Group layouts

2.9.3 Rulers/Scales

Rulers/scales appear on the vertical or horizontal sides of the view. These rulers/scales are associated with the axis they represent. Figure 2-24 on page 66 provides an example of multiple views featuring various rulers/scales.

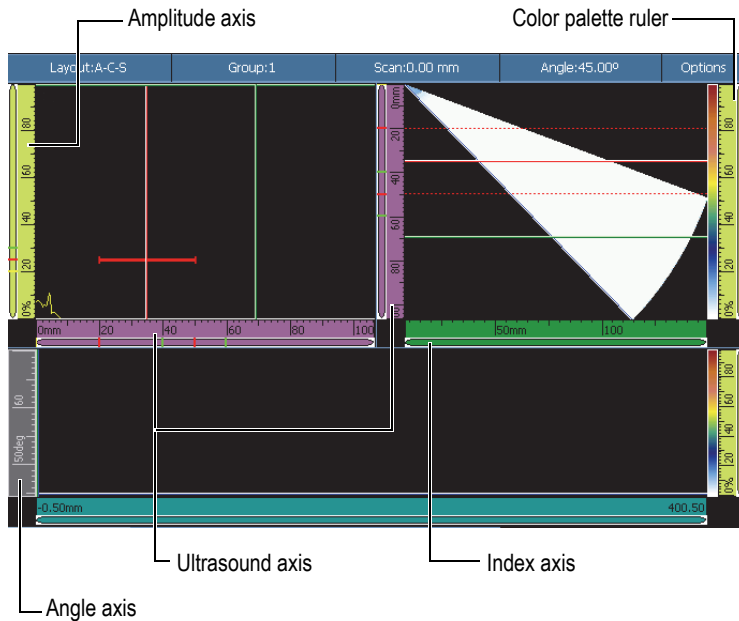


Figure 2-24 Example of multiple views with various rulers/scales

TIP

- To zoom in on a section of the data screen, tap and drag to draw a rectangle around the section to be enlarged, and then release the screen. To zoom in the vertical axis, move your finger vertically. To zoom in the horizontal axis, move your finger horizontally. Zoom can only be used along axis rulers/scales that contain a scroll bar.
- To return to the unzoomed view, double-tap the x-axis and y-axis ruler/scales or press the Zoom key (OmniScan MX2 only). To unzoom all rulers to their original scale, tap and hold a ruler and select **Zoom Out**.

Each ruler/scale is filled with a dedicated color to help you identify the axis on the various views. Table 16 on page 67 provides a list of the available rulers/scales, and their associated colors and functions.

A basic color is attributed to each axis. The axis appears in various tints of this basic color. The lightest tint corresponds to the raw data representation. Progressively darker tints correspond to increasing data correction complexity relative to the axis. A darker tint is also used for an axis that appears as a reference, in which case the zoom bar is not available.

Table 16 Ruler/scale colors^a















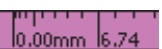

Axis	Basic color	Appearance	Specific color	Note
Scan	Blue		Cyan	Uncorrected (length or degree units depending on scan-axis unit)
			Dark blue	No zoom
			Blue	Volumetric correction (strip chart only)
Index	Green		Pale-green	N/A
			Green	Volumetric correction
Amplitude	Yellow		Yellow	N/A
Ultrasound	Pink		Pale pink	Time units
			Pink	Sound path units
			Purple	True-depth units
Color palette ruler	Yellow		Yellow	Displayed next to the color palette

Table 16 Ruler/scale colors^a (continued)

Axis	Basic color	Appearance	Specific color	Note
Angle	Grey		Grey	Sectorial inspection only Units in degrees
VPA			Grey	Linear inspection only Units in ultrasonic beam number
Time	Grey		Pale grey	N/A
Group			Dark grey	N/A
TOFD	Pink		Pale pink	Displayed when a TOFD group is selected
			Dark pink	Displayed after the lateral wave synchronization has been performed.

a. The colors in this document differ slightly from those on the OmniScan instrument screen.

2.10 Outline Colors

Colors are used to outline functions and reading fields in order to facilitate tracing of the elements with applicable parameters or to indicate the origin of the information.

2.10.1 Reading-Field Colors

The reading fields can be outlined in one of seven colors. The color indicates the interface element corresponding to the value appearing in the field (see Figure 2-25 on page 69).

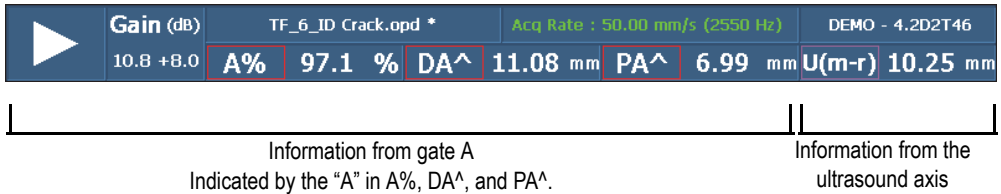


Figure 2-25 Three readings outlined in red, and one in purple

Gate colors

- Red: The information is from gate A.
- Green: The information is from gate B.
- Yellow: The information is from gate I.

Cursor colors

- Red: The information is from the Reference cursor.
- Green: The information is from the Measurement cursor.
- Blue: The information is from the Data cursor.

NOTE

Red and green refer to either the gate information or the cursor information. When the information is from gate A or gate B, the reading-field code includes the gate letter. In Figure 2-25 on page 69 the first three reading fields present information from gate A. For more details on reading-field codes, see "Reading Submenu" on page 259.

UT unit colors

The following information applies to the ultrasound axis:

- Pale-pink: **Time** (μ s)
- Pink: **Sound Path** (mm) [also called *half path*]
- Purple: **True Depth** (mm)

2.10.2 Parameter Button Colors

In certain submenus, some or all of the parameter buttons are outlined in one of six colors. The color indicates the interface element to which a parameter applies.

Colors referring to gates

Three colors refer to the gates (see Figure 2-25 on page 69):

- Red: The parameter applies to gate A.
- Green: The parameter applies to gate B.
- Yellow: The parameter applies to gate I.

Colors referring to ultrasound units

Three colors refer to the ultrasound units used on the ultrasound axis (see example in Figure 2-26 on page 71). The ultrasound units are selected using **UT Settings > General > UT Mode**. Note that the **UT Mode** settings can only be changed using this menu or, if in PA mode, using the **Stacked A-Scan** application in the **Setup** wizard.

- Pale pink: **Time** (μs)
- Pink: **Sound Path** (mm) [also called *half path*]
- Purple: **True Depth** (mm)

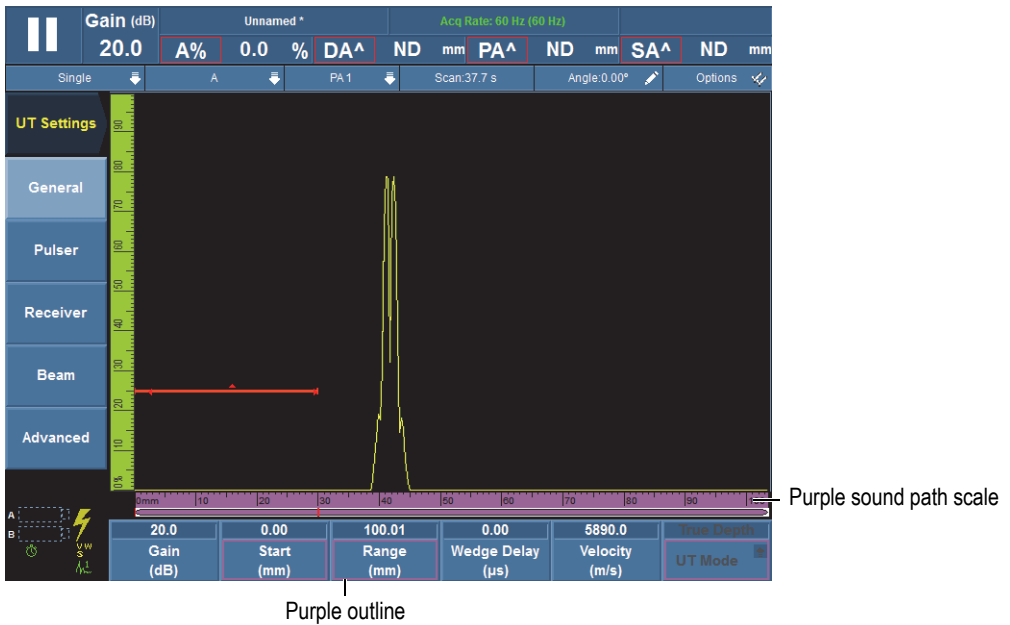


Figure 2-26 The purple color for ultrasound axis and mode button

2.11 Parameter Buttons

The parameter buttons are located at the bottom of the interface display. There can be up to six parameter buttons per submenu. Table 17 on page 72 lists the types of parameter buttons. See Table 11 on page 31 for the procedure on editing parameter values.

NOTE

For some submenus, selecting the left-most parameter button in the list changes the other available parameter buttons.

Table 17 Parameter button types


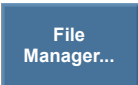
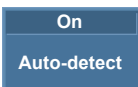
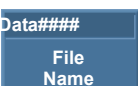
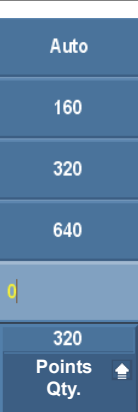
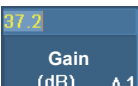
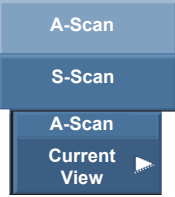
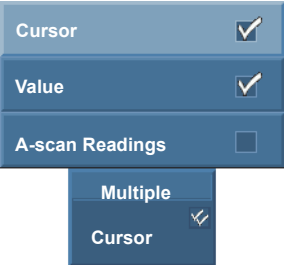
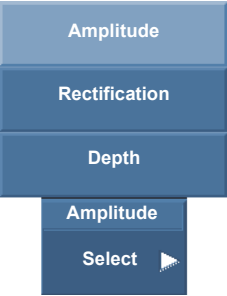
Type	Example	Description
Command		Causes a specific action to be carried out.
Command launch		Opens a dialog box or another screen providing more controls related to the command.
Toggle		Alternates between two settings.
Edit		Used to enter a numeric or alphanumeric value or to modify an existing value.
Edit list		Displays a list with an editable field as the last item, enabling you to select a predefined item or to create a new item.
Edit/cyclic		Contains two different values that can be edited separately: <ul style="list-style-type: none"> a) At the top, a numeric value similar to an edit button value b) At the bottom, a numeric increment/decrement step (Δ) value

Table 17 Parameter button types (*continued*)

Type	Example	Description
List		Displays a list of predefined items for selection.
Check list		<p>Displays a list of one or more check box items that can be selected individually.</p> <p>Select an item in the list, and then press the Accept key to toggle the state of the check box. The effect is immediate.</p> <p>Press the Cancel key or select another button to close the parameter list.</p>
Parameter category		Displays a list of parameter categories. The selection changes the parameters displayed on the right side of this button.

Many parameter buttons contain a numeric value. There are several ways to edit the value of the parameter button.

When you select a parameter with a numeric value, the value appears in yellow, and a flashing cursor appears to the right of the value to show that it can be edited. An increment/decrement step value also appears in the lower right corner of the button (see Figure 2-27 on page 74). Several increment/decrement step values are available to enable quick editing of parameter button values.

NOTE

For more details about increment/decrement steps, see “OmniScan MX2 Increment Steps” on page 16 or “OmniScan SX Increment Steps” on page 16.

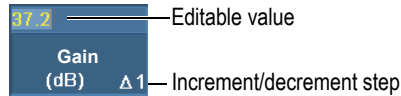


Figure 2-27 Editing a parameter value with the increment/decrement step

To edit the numeric value of a parameter

1. Choose the parameter to enable the edit mode.

MX2	SX
<ol style="list-style-type: none"> 2. On the instrument, press the Raise increment key to increase the value of the step, or press the Lower increment key to decrease the value of the step. 	<ol style="list-style-type: none"> 2. Tap the parameter twice or press the Accept key twice to open the software keyboard. Use the +Δ or -Δ button to increase or decrease the increment (select from the predefined list of increments).

3. On the OmniScan instrument, rotate the Scroll knob clockwise to increase the value of the parameter, or counterclockwise to decrease the value of the parameter.
OR
On the OmniScan instrument, type the value using the virtual keyboard.
4. Accept the edited value as follows:
 - ◆ On the OmniScan instrument, press the Accept key.
OR
Press another key or button.
OR
Tap any layout view.

2.12 Compression Feature

To support the corrosion mapping and composites applications, the Compression feature is available. Compression is included on the B-scan and C-scan to ensure that the most relevant information in a pixel is displayed at all times. For an amplitude C-scan or B-scan, the pixel color is determined by the data point of highest amplitude. For a “Time-of-flight” or “Position” C-scan, the pixel color is determined by the data point of shortest time-of-flight (thinnest). If the inspected area has more data points than pixels, the Compression feature will be turned on automatically in order to select which data will be displayed for each pixel, and the “C” symbol will be displayed on the status indicator, as shown below (see Figure 2-28 on page 75):

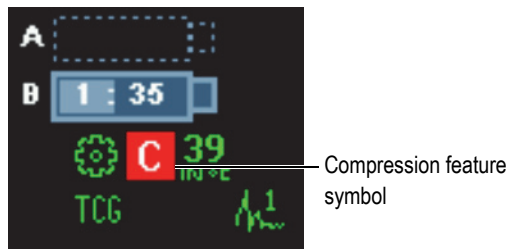


Figure 2-28 The Compression feature symbol

Also, if a zoom is performed on the C-scan, and all the data points are displayed, the Compression symbol and the Compression indicator will no longer be displayed. This function is always active, and does not require configuration.

Compression Parameter

Used in analysis mode, the **Compression** parameter is available when a TOFD group is active and when **Display > View Settings > Category = B-Scan** is selected. When set to **Off**, this parameter ensures that no compression is active by performing an automatic zoom on the current view. Set the **Compression** parameter to **On** to avoid hiding a data value; however, this will change the zoom.

2.13 Online Help

The OmniScan MXU software offers various types of help information:

- Help information related to wizard steps.
- Contextual help regarding the functions corresponding to each menu, submenu, or parameter button. The content of this help system is the same as that in “Menu Description” on page 229.

2.13.1 Displaying Contextual Help

The OmniScan MXU software features contextual help related to menu, submenu, and parameter buttons.

To display contextual help

1. Choose the menu, submenu, and parameter button for which you want to see contextual help information.
2. Press the Help key on the front panel.
The associated help information appears in the data screen area.
3. To display help information on a different function, simply choose the button corresponding to the function.
The contextual help information displayed is updated.
The help information disappears when you press the Help key a second time.

NOTE

When pressing the Help key, no help information appears unless a button is selected. Make sure that a menu, submenu, or parameter button is selected.

2.13.2 Displaying Online Help for Wizard Steps

The step-by-step procedures of the wizards provide specific help information for each step. The information that appears in the data screen area of the OmniScan MXU software window helps you select appropriate values for the parameters available for the current step.

NOTE

The **Calibration** wizard does not provide help information. The data screen is needed to show real-time graphical data as you move the probe over the calibration part.

3. General Procedures

This chapter contains general procedures that use functions from the **File** and **Preferences** menus, and it describes commonly performed tasks that have an overall impact on the OmniScan MXU software interface.

3.1 Setting Preferences

The OmniScan MXU software provides functions to set a number of system preferences through the **Preferences** menu.

3.1.1 Setting the Date and Time

This section describes how to set the date and time on the OmniScan instrument. The date and time appear in the reading-field area of the OmniScan instrument screen, and they are recorded along with acquired data.

To set the time

1. Select **Preferences > Instrument > Category = System**.
2. Select **Clock Set** once, and then use the Scroll knob to set the hour.
3. Tap the numeric parameter to move to the value for minutes.
4. Enter the minutes using the Scroll knob.
5. Tap the numeric parameter to move to the value for seconds.
6. Enter the seconds using the Scroll knob.
7. Tap the numeric parameter to move to the value for half-day period.
8. Use the Scroll knob to toggle between **AM** (for morning) and **PM** (for the afternoon).
9. Press the Accept key to complete the task.

To set the date

1. Select **Preferences > Instrument > Category = System**.
2. Select **Date Set** once, and then use the Scroll knob to set the year.
3. Tap the numeric parameter to move to the value for months.
4. Enter the month using the Scroll knob.
5. Tap the numeric parameter to move to the value for days.
6. Enter the day using the Scroll knob.
7. Press the Accept key to complete the task.

3.1.2 Selecting the Length Measurement Units

This section describes how to select the length measurement units.

To select the length measurement units

- ◆ Tap and hold the software version field of the reading-fields area, and then select the new measurement units (**Millimeters** or **Inches**).
- OR
- Select **Preferences > Instrument > Category = Units**, and then, in the **Length Units** list, select the preferred measurement unit (**Millimeters** or **Inches**).

3.1.3 Configuring the Digital Inputs

This section explains how to configure the digital inputs (DIN).

To configure the digital inputs

1. Select **Preferences > Setup > Category = DIN**.
2. In the **Select DIN** list, select the key (**DIN1** through **DIN4**) to which a digital input is to be assigned.
3. In the **Assign DIN** list, select the parameter that is to be displayed next to the key number selected in step 2.
4. Repeat steps 2 and 3 to configure the other digital inputs.

3.1.4 Changing the Screen Color Scheme

The screen color scheme can be changed in order to optimize it for either indoor or outdoor operation.

To change the screen color scheme

- ◆ Tap and hold the software version field of the reading-fields area, and then select the new color scheme (**Indoor** or **Outdoor**).

OR

Select **Preferences > Instrument > Category = Screen**, and then, in the **Scheme** parameter list, select the appropriate color scheme (**Indoor** or **Outdoor**).

3.1.5 Toggling the Wizard Help Screen

This section explains how to toggle the wizard help screen. The wizard help is displayed by default. Once you are familiar with the wizards, you can choose to hide the wizard help so that the full graphical area is available for RayTracing.

To toggle the wizard help screen

1. Select **Preferences > Instrument > Category = Help**.
2. Toggle the value of the **Wizard Help** parameter (**On** or **Off**).

NOTE

When using the wizard, deactivating the wizard help function will speed up the refresh time for each step, for quicker operation.

3.2 Managing Files

The location where files are saved or retrieved can be selected by choosing **File > Data Settings > Storage**.

You can save or load any file type to the storage card, to a network (OmniScan MX2 only), or to an external hard drive (USB storage) at any time.

3.2.1 Saving Files

This section outlines the procedure for saving files on the OmniScan instrument.

3.2.1.1 Saving a Setup

Save the setup data regularly to prevent accidental data loss.

To save a setup

1. In the **File > Data Settings > Storage** list, select the location where you want to save the setup.
2. Tap and hold the file name field in the reading-fields area, and then select **Save As**.
OR
Select **File > Setup > Save As**.
3. In the window that opens, select **File Name**, and then enter the desired name for the setup file.
4. Select **Save** to save the file and close the window.

3.2.1.2 Saving a Data File

It is recommended that you save data regularly to prevent accidental data loss.

To save an inspection data file

1. Tap and hold the file name field in the reading-fields area, select **File Name**, and then enter a file name root before the series of ####.
OR
Select **File > Data Settings > File Name**, and then enter a file name root before the series of #### (see Figure 3-1 on page 82).
For more information on file naming codes, see “Data Settings Submenu” on page 236.

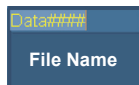


Figure 3-1 File Name parameter button

2. Select **File > Data > Save**.

OR

If you are in Analysis mode, tap and hold the file name field in the reading-fields area, and then select **Save**.

TIP

Select **File > Data Settings > Save key = Data** to configure the Save/Print key to save inspection data. Pressing the Save/Print key is then equivalent to choosing **File > Data > Save**.

3.2.2 Opening Files

This section describes how to open different files in the OmniScan instrument.

3.2.2.1 Opening Any Type of File

To open any type of file

1. Tap and hold the file name field in the reading-fields area, and then select **Open**.
2. In the File Manager, select **File Type**.
3. Use the Scroll knob to select the desired file, and then select **Open**.

3.2.2.2 Opening Setup Files

To open a setup file

1. In the **File > Data Settings > Storage** list, select the location where the desired setup file is saved.
2. Select **File > Setup > Open**.
3. In the File Manager, select **File Type = Setup**.
4. Use the Scroll knob to select the desired setup file, and then select **Open**.

3.2.2.3 Opening Connectivity Files

To open a connectivity file

1. In the **File > Data Settings > Storage** list, select the location where the desired connectivity file is saved.
2. Select **File > Setup > Import Connectivity**.
3. Use the Scroll knob to select the desired connectivity file, and then select **Open**.

3.2.2.4 Opening Data Files

To open a data file

1. In the **File > Data Settings > Storage** list, select the location where the desired data file is saved.
2. Select **File > Setup > Open**.
3. In the File Manager, select **File Type = Data**.
4. Use the Scroll knob to select the desired data file, and then select **Open**.

3.2.2.5 Opening Image Files

To open an image file

1. In the **File > Data Settings > Storage** list, select the location where the image is saved.
2. Select **File > Image > Open**.
3. In the File Manager, select **File Type = Image**.
4. Use the Scroll knob to select the desired image file.
A preview of the image contained in the file appears in the destination pane.
5. Select **Open**.
The image fills the entire screen.
6. Select **Close** to return to the File Manager.
7. In the File Manager, select **Close** to return to the data screen.

3.2.2.6 Opening Report Files

To open a report file

1. In the **File > Data Settings > Storage** list, select the location where the report is saved.
2. Select **File > Report > Open**.
3. In the File Manager, select **File Type = Report**.
4. Use the Scroll knob to select the desired report file.
A preview of the report contained in the file appears in the destination pane.
5. Select **Open**.
The report appears on the entire screen, providing you with two options: printing the report or closing it.
6. Select **Close** to go back to the File Manager.
7. Select **Close** once again to return to the data screen.

3.2.3 Setting the Save/Print Key

The Save/Print key on the front panel of the OmniScan instrument allows you to instantly create a file containing a specific type of data. You need to set the Save/Print key for the type of data to be saved, and to set the format for the name of the file and the destination where the file is to be saved.

To set the Save/Print Key

1. In the **File > Data Settings > Save Key** list, select the item you want from the list (**Report, Data, Image, or Data&Image**).
2. Select **File Name**, and then enter an automatic file naming format using “#” characters in order to specify automatically increasing numbers.
For example, entering DATA#### tells the system to create files with automatic file names starting with DATA0001, DATA0002, and so on.
3. In the **Storage** list, select the storage destination.
The OmniScan instrument saves the data type any time you press the Save/Print key.

For more information regarding the parameters listed in the above procedure, see “Data Settings Submenu” on page 236.

3.3 Producing Reports

This section outlines the procedures for configuring and printing reports.

NOTE

If you want to customize the reports with your company logo, see “Changing the Company Logo” on page 189 for the procedure on creating a custom report template.

3.3.1 Configuring a Report

This section explains how to configure a report.

To configure a report

1. Select **File > Report > Category = Format**.
2. Set the parameter buttons (see “Format Category” on page 232) at the bottom of the screen based on the information you want to include in the report:
 - a) Select **Include** to add the current layout or indication table to the report.
 - b) Select **Component**, and select the desired components.
 - c) Select **Edit Notes** or **Edit Header** to open the note pad.
 - d) Enter the note using the virtual keyboard or a USB keyboard. Select **Accept** in the virtual keyboard to proceed to the next line.
 - e) Select **Close** to close the note pad.
 - f) Select **Yes** in the dialog box that appears in order to accept the changes, and save the note.

The note is displayed at the top of the report footer, and the header is displayed at the top of the report header (see Figure 3-2 on page 87).
 - g) Select **Template**, and select the appropriate report template.



OmniScan Report

Text added using FILE > REPORT > EDIT HEADER appears here

Report Date	Report Version	Setup File Name	Inspection Date	Inspection Version	Save Mode
2009 / 09 / 18	MXU-M - 2.2R1	Setup.ops	2009 / 09 / 18	MXU-M - 2.2R1	Inspection Data
OmniScan Type	OmniScan Serial #	Module Type	Module Serial #	Calibration Due	Data File Name
OmniScan.MX	OMNI-	OMNI-M-PA16128	OMNI-2763	2008.04.23	File####

Notes
Text added using FILE > REPORT > EDIT NOTES appears here

Technician Name _____

Technician Signature _____

Contractor _____

Date _____

Figure 3-2 Example of header note (*top*) and footer note (*bottom*) in a report

3. If you chose to include **User Fields** parameters (**File > Report > Category = Format** and **Component = User field**), select **File > Report > Category = User Fields**, and proceed as follows:
 - a) In the **Field** list, select the number of the user field you want to include in the report.
 - b) Select **Enable = On** to make the user field appear in the report.
 - c) Make sure that the **Label** parameter displays the information you require. If not, enter the new information in the edit field (see Figure 3-3 on page 87).
 - d) Make sure that the **Content** parameter displays the information you want to attach to the label. If not, enter the information in the edit field (see Figure 3-3 on page 87).

Contractor	NDT Generics	Technician	John Smith
		Project	The Big Project

Figure 3-3 Example of Label and Content parameters in a report

- e) Repeat steps 3.a through 3.d until all the required user fields have been activated.
4. Select **File > Data Settings > File Name**, and then enter a name for the report. For more information regarding the available file name variables, see “Data Settings Submenu” on page 236.
5. To see the report configuration result, proceed as follows:
 - a) Select **File > Report > Category = Open/Save**.
 - b) Select **Preview**.
The report appears on the screen.
6. In the report viewer, proceed as follows:
 - ◆ Select **Save and Close** to save the report to an HTML file on the destination device, and then return to the **File > Report** submenu.
OR
Select **Close** to return to the **File > Report** submenu without saving the report.

3.3.2 Printing a Report from a Computer

It is possible to transfer a report from the OmniScan instrument to a computer, and then print it from that computer.

To print a report from a computer

1. Create a report on the OmniScan instrument (see “Configuring a Report” on page 86).
2. Save the report on the storage card, and note the name of the report.
3. Remove the storage card from the OmniScan instrument, and insert it into a memory card reader connected to your computer.
4. On the computer, open the \User\Report folder on the storage card.
5. Copy both the report file (.html extension) and the folder with the same name to the computer.
6. Open the HTML report file (.html) with Microsoft Internet Explorer.
7. In Microsoft Internet Explorer:
 - a) On the **File** menu, click **Page Setup**.
 - b) In the **Margins** group box, assuming that measurement units are in millimeters, enter **10** in the **Left** text box, and then enter **10** in the **Right** text box.

- c) Click **OK**.
- d) Under the **File** menu, click **Print** to print the report.

4. Setup Procedures

The setup procedures in this chapter show you how to set up various aspects of the inspection configuration. These procedures must be followed when you need to inspect a new part or when you modify the way you inspect a part.

4.1 Creating the Application Setup with the Wizards

The **Wizard** menu offers step-by-step assistance to quickly and easily create a complete setup for your application (see Figure 4-1 on page 92). The step-by-step approach ensures that you don't miss a parameter change. The help screen of each wizard step provides specific information regarding the parameters to be set.

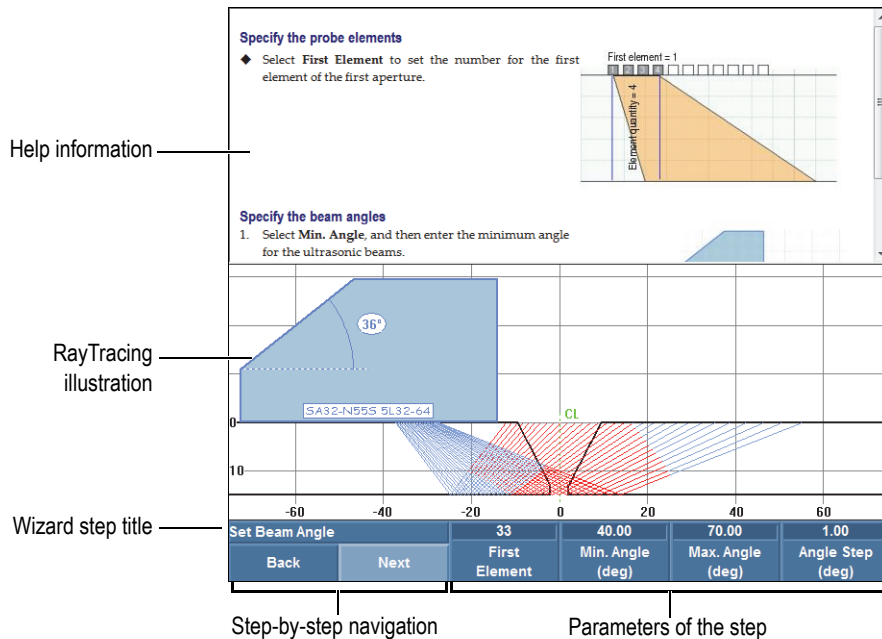


Figure 4-1 Elements of a wizard step

To perform a complete application setup, use the different wizards available from the submenu buttons, in the following order:

- Part and weld (see “Configuring the Part and Weld” on page 92)
- Setup (see “Modifying the Setup” on page 93)

4.1.1 Configuring the Part and Weld

To configure the part and weld

1. Select **Wizard > Part & Weld > Start** to start the **Part & Weld** wizard.
2. Follow the on-screen help information to complete the remaining steps of the wizard.

TIP

You can exit a wizard at any time by pressing the Cancel key.

4.1.2 Modifying the Setup

To modify the setup

1. Select **Wizard > Setup > Start** to start the **Setup** wizard.
2. Select **Modify** from the **Operation** parameter list, and then, in the **Current Group** parameter list, select the Group number to be modified.
3. Follow the on-screen help information to complete the remaining steps of the wizard.

The wizard steps depend on the selection made in step 1. Only the parameters that are relevant to the technology (PA or UT) and the selected application will be configured.

TIP

You can exit a wizard at any time by pressing the Cancel key.

4.2 Selecting the Probe and the Wedge

The OmniScan instrument needs to obtain the necessary information about the probe and wedge models to be used for your inspection.

The OmniScan instrument automatically detects and loads the characteristics of an Olympus PA probe when you connect one to the instrument with an OmniScan connector. When you use an OmniScan connector and a different brand of probe, you need to manually select the probe from the list of predefined probes.

You also need to select a predefined wedge or define the wedge that you want to use for your inspection.

To set up the probe and the wedge, you can use the wizard or navigate directly in the menus and submenus.

To manually select a probe

1. Select **Wizard > Setup > Start**.
2. Select **Next** until you reach the **Select Probe and Wedge** step.
OR
Select **Group/Probe & Part > Probe & Wedge > Select/Edit = Select**.
3. Select **Probe** to choose your probe from the predefined list (see Figure 4-2 on page 94):
 - a) Choose the appropriate probe category in the list on the left side.
 - b) Choose the appropriate probe model in the list on the right side.
The selected probe specifications appear at the bottom of the dialog box.
 - c) Press the Accept key or **Select** to select the probe.

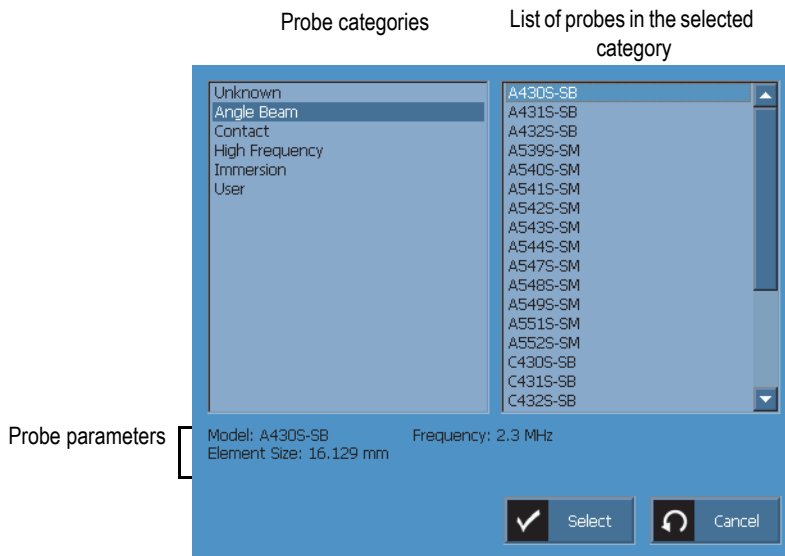


Figure 4-2 The probe selection dialog box

4. When your probe is not available in the predefined list, see “Defining a Probe” on page 96 for the procedure used to define a new probe.

To manually select a wedge

1. Select **Wizard > Setup > Start**.
2. Select **Next** until you reach the **Select Probe and Wedge** step.
OR
Select **Group/Probe & Part > Probe & Wedge > Select/Edit = Select**.
3. Select **Wedge** to open the predefined list of wedges (see Figure 4-3 on page 95):
 - a) Choose the appropriate wedge category in the list on the left side.
 - b) Choose the appropriate wedge model in the list on the right side.
The selected wedge specifications appear at the bottom of the dialog box.
 - c) Press the Accept key or choose **Select** to select the wedge.

NOTE

In Phased Array mode, only the wedges that are dedicated to the probe are available by default. The list of dedicated wedges speeds up the wedge selection process. However, if you need to view the complete list, you can use the **Show Dedicated / Show All** button to toggle between the full and the dedicated lists of wedges.

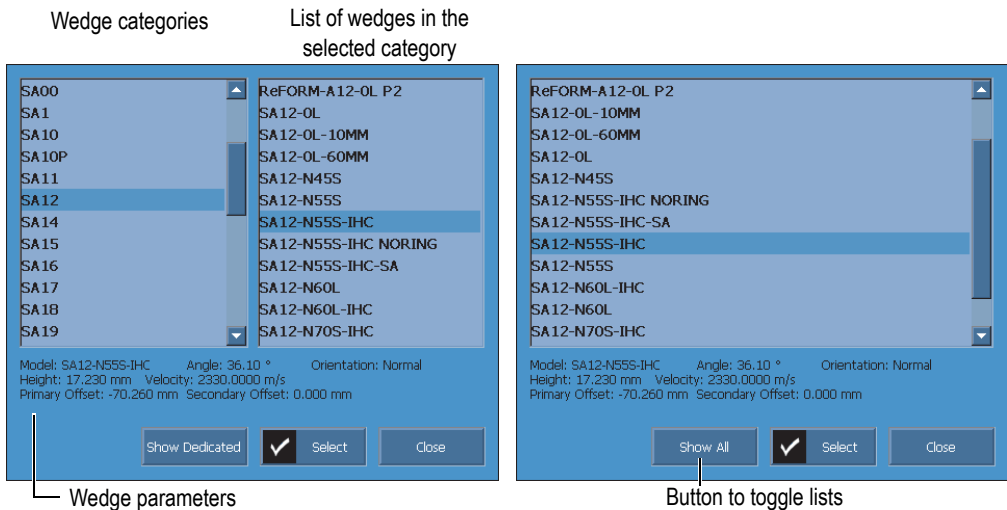


Figure 4-3 The lists of wedges in the wedge selection dialog box

4. If your wedge is not available in the predefined list, see “Defining a Wedge” on page 98 for the procedure used to define a new wedge.

4.3 Defining a Probe

This section describes how to define a probe in the OmniScan instrument.

NOTE

If the probe is not an Olympus PA probe, an adaptor is needed to interface your other brand probe connector with the OmniScan PA connector.

To define a probe

1. Select **Wizard > Setup > Start**.
2. Choose **Next** until you reach the **Select Probe and Wedge** step.
OR
Select **Group/Probe & Part > Probe & Wedge > Select/Edit = Edit**.
This selection gives you access to the **Edit Probe** and **Edit Wedge** parameter buttons.
3. Choose **Select/Edit = Edit**.
4. Select **Edit Probe** to open the Probe Definition Manager.
5. Select **New**.
6. Select **Serial Number**, and then enter the probe serial number.
7. Select **Save**.
A new **Custom / <serial number entered>** probe appears in the probe list.
8. In the **Freq. (MHz)** list, select the probe frequency.
9. If you are using a PA probe, proceed as follows (for conventional UT probes, see step 10):
 - a) In the **Ref. Point** button, enter the probe reference point, which is the distance between the front of the probe and the center of the first probe element (see Figure 4-4 on page 97).

IMPORTANT

By default, the OmniScan MXU software establishes the 0-reference point at the position of the first element. To set the 0-reference point at the front edge of the probe, enter the distance between the front edge of the probe and the position of the first element. The value must be negative.

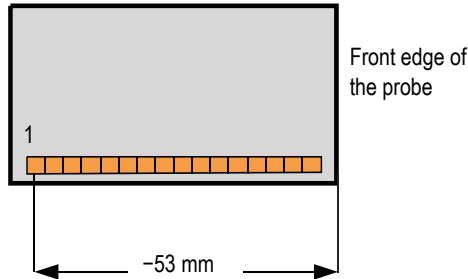


Figure 4-4 The probe reference point measurement

- b) From the **Type** list, select the probe type.
- c) In the **Elements Qty.** button, enter the number of elements in the probe.
- d) In the **Pitch** button, enter the distance between the centers of two adjacent probe elements. This distance is illustrated in Figure 4-5 on page 97.

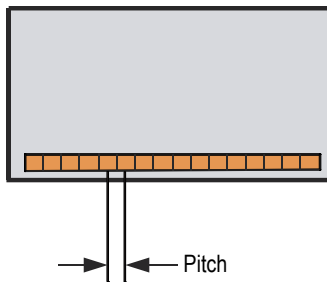


Figure 4-5 Probe-element pitch

10. If you are using a conventional ultrasound (UT) probe, select **Element Diameter** (circular probe), or **Side 1 Length**, or **Side 2 Length** (rectangular probe), and then enter the value (mm) of the probe-element diameter or size.
11. Select **Save**.
12. Choose **Select and Close** to select the probe and quit the Probe Definition Manager.

NOTE

The new probe is now available from the list of defined probes under the **User** category (see “Selecting the Probe and the Wedge” on page 93).

4.4 Defining a Wedge

The OmniScan MXU software requires information about the wedge you are using. If the wedge is not present in the list of predefined wedges, you must create a definition for the wedge.

To define a wedge

1. Select **Wizard > Setup > Start**.
2. Choose **Next** until you reach the **Select Probe and Wedge** step.
OR
Select **Group/Probe & Part > Probe & Wedge > Select/Edit = Edit**.
This selection gives you access to the **Edit Probe** and **Edit Wedge** parameter buttons.
3. Choose **Select/Edit = Edit**.
4. Select **Edit Wedge** to open the Wedge Definition Manager.
5. Select **New**.
6. Select **Serial Number**, and then enter the wedge serial number.
7. Select **Save**.
A new **Custom / <serial number entered>** wedge appears in the wedge list.
8. For a PA wedge, set the following parameters:
 - a) Select **Wedge Angle**, and then enter the angle of the wedge (see Figure 4-6 on page 99).

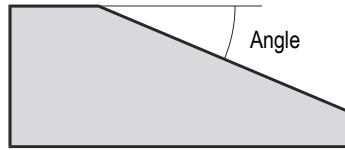


Figure 4-6 Wedge angle

- b) Choose the appropriate value from the **Orientation** list:
 - **Normal**: When the probe is installed on the wedge so that its cable is on the lower side of the wedge slope.
 - **Reverse**: When the probe is installed on the wedge so that its cable is on the higher side of the wedge slope.
- c) Select **Velocity**, and then enter the ultrasonic velocity of the wedge material. For an Olympus wedge, the value is usually 2330 m/s.
- d) Select **Pri. offset**, and then enter the value of the primary offset, which is the distance between the front of the wedge and the center of the first probe element (see Figure 4-7 on page 99).

IMPORTANT

By default, the OmniScan MXU software establishes the 0-reference point at the position of the first element. To set the 0-reference point at the front edge of the wedge, in **Pri. Offset**, enter the distance between the front edge of the wedge and the position of the first element. The value must be negative.

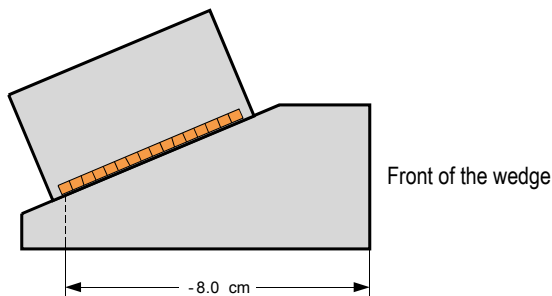


Figure 4-7 The primary offset measurement

- e) Select **Sec. offset**, and then enter **0** to indicate that the probe is centered on the wedge on the secondary axis. If the probe is not centered on the wedge, enter the appropriate value (see Figure 4-8 on page 100).

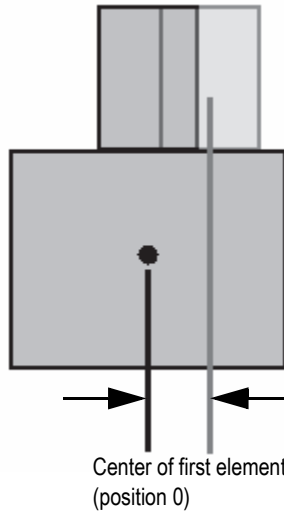


Figure 4-8 The secondary offset

- f) Select **Height**, and then enter the height of the first element (see Figure 4-9 on page 100).
This parameter is usually provided by the wedge manufacturer.

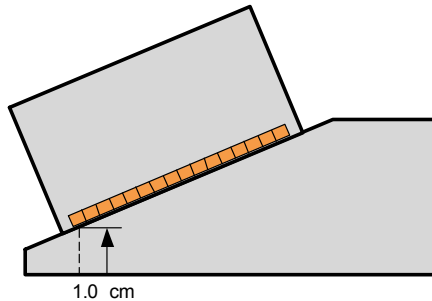


Figure 4-9 The height of the first element

9. For a UT wedge, set the following parameters:
 - a) Select **Refracted Angle**, and then enter the angle of the wedge.
 - b) Select **Probe Delay**, and then enter the appropriate value in μs .
 - c) In the **Wave Type** list, choose Long Wave (**LW**) or Shear Wave (**SW**).
 - d) Select **Ref. Point**, and then enter the value of the reference point.
 10. Select **Save**.
 11. To select the wedge, choose **Select and Close**.
-

NOTE

The new wedge is now available from the list of defined wedges under the **User** category (see "Selecting the Probe and the Wedge" on page 93).

4.5 Defining the Part to Be Inspected

This section describes how to define a part that is to be inspected.

To define the part to be inspected

1. Select **Wizard > Part & Weld > Start**.
2. From the **Specimen Type** list, select the parameter that defines the geometry of the part to be inspected.
Select **Plate** for a flat part, or **Tube** for the inspection of a curved part.
3. Select **Thickness**, and then enter the part thickness. If you have selected **Tube** in step 1, select **Diameter** to enter the part diameter.
4. In the **Material** list, select the type of material the part is composed of.
5. If you have selected **Tube** in step 1, select the **CSC Mode** and **Weld Orientation** for either an external (**OD**) or internal (**ID**) inspection of a curved part and for the orientation of the weld (**Circumferential** or **Axial**).

4.6 Characterizing a Probe Using the FFT

Characterize probes regularly to verify whether their nominal frequency is shifting or spreading. Apart from the probe to be characterized, you need a calibration block from which to get a clean back wall echo. The probe characterization is performed by taking the FFT of the back wall echo signal under gate A.

To characterize a probe using the FFT

1. Connect the probe to be characterized to the OmniScan instrument.
2. To be able to save the probe characterization data to the setup file, select the probe (see “Selecting the Probe and the Wedge” on page 93) or define it (see “Defining a Probe” on page 96).
3. Position the probe on your calibration block to obtain a clean back wall echo.
4. If you only want to see gate A on the A-scan, select **Display > Overlay > Gate = A**, and not **B** or **I**.
5. Select **UT Settings > Receiver > Video Filter = Off**.
6. Select **Rectifier = RF**. The FFT is only available without rectification.
7. Adjust the **UT Settings > General > Start** and **Range** parameters so that only the first back wall echo signal is visible on the A-scan. A minimum range around the back wall echo improves the quality of the FFT results by maximizing the number of points available for the FFT calculation. If the range is too high, a warning message will appear.
8. Select **File > Data Settings > Data Selection = All A & C-Scans**. The FFT is available only when A-scan data can be saved.
9. Adjust the **Preferences > FFT > Gain** parameter so that the back wall echo maximum and minimum amplitudes are at about 80 % of the full-screen height.
10. Adjust the **Start** and **Width** parameters so that gate A is positioned over the back wall echo. Gate A must appear in full on the A-scan.
11. Select **FFT = On**.

The FFT curve and data appear below the A-scan, as in the example in Figure 4-10 on page 103.

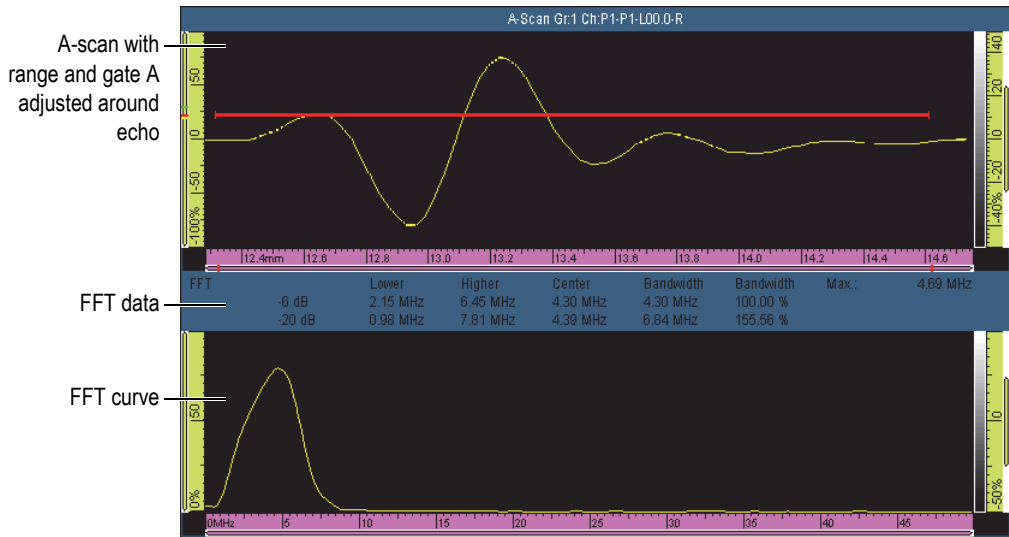


Figure 4-10 Example of FFT curve and data for a 5 MHz probe

12. Compare the FFT data with the one on the specification sheet received with the probe at the time of purchase in order to determine whether the probe nominal frequency has shifted or spread.
13. Optionally, for future reference, do the following:
 - a) Select **Procedure**, and then enter a name for the characterization procedure.
 - b) Select **Block Name**, and then enter a name for the calibration block used.
14. To save the probe characterization data, select **FFT = Off**, and then select **Yes** in the dialog box prompting you to save the characterization.
15. To produce a report that includes the probe characterization information:
 - a) Select **File > Report > Category = Format**.
 - b) Select **Component = Probe**.
 - c) Select **Category = Open/Save**, and then **Preview**.

The report appears on the screen and includes a **Probe Characterization** section (Figure 4-11 on page 104).



Figure 4-11 Example of a Probe Characterization section in a report

NOTE

A high number of A-scan points is recommended in order to maximize the FFT precision. Select **UT Settings > Advanced > Points Qty. = Auto** (default value) in order to maximize the number of points for the selected inspection range.

4.7 Working with Multiple Groups (OmniScan MX2 Only)

This section contains the procedures for adding, removing, and selecting a group.

NOTE

Any parameter changes are applied to the selected group. The screen shows the selected group, except in the case of multiple views.

To add a group

1. Select **Wizard > Setup > Start**.
2. Select **Operation > Add**.
3. In the **Source** parameter list, select **Default** to copy connector default settings in the new group.
OR
Select **Copy From** to copy parameters from an existing group.
4. Select **Next** to create your group, and continue to change the parameters.
After a group has been added, it is automatically selected, and all new parameters will apply to this group.

To remove a group

1. Select **Wizard > Setup > Start**.
2. Select **Operation > Delete**.
3. In the **Current Group** list, select the group you wish to delete.

NOTE

It is not possible to have less than 1 group or more than 8 groups.

To select a group

- ◆ Touch the area on the screen corresponding to the group.
OR

Tap the Group section of the title bar, and then select the desired group.

To display a multiple group layout

- ◆ Tap the Mode section of the title bar, and then select **Multiple**.

4.8 Selecting the Group Mode on the OmniScan MX2

Using the **Group Mode** parameter, you can create conventional ultrasound (UT) groups using the UT connector (if supported by the module), or the PA connector (adaptor required).

To select the Group Mode

- ◆ Select **Wizard > Setup > Start > Group Mode**, and then choose **PA** or **UT**.
If you choose the **UT Conventional Ultrasound (using PA connector)** group mode, be sure to connect the UT probes to the PA connector using one of the available adaptors (see the example in Figure 4-12 on page 106).

Example: EXT-128-1-4L adaptor connects up to four UT groups (or two TOFD groups).

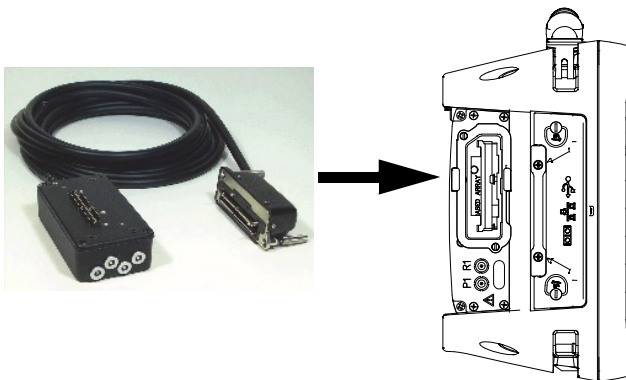


Figure 4-12 Example adaptor to connect UT probes to PA connector

4.9 Selecting the Group Mode on the OmniScan SX

Using the **Group Mode** parameter, you can create a conventional UT group or a PA group.

To select the Group Mode

- ◆ Select **Wizard > Setup > Start > Group Mode**, and then choose **PA** or **UT**.

5. Calibration Procedures

Before starting an inspection, you must perform several calibration procedures using a probe, wedge, and calibration block composed of the same material as the part to be inspected.

5.1 Selecting the Element to Calibrate

The **Wizard > Calibration** submenu provides wizards for calibrating encoders, ultrasonic aspects (**Velocity**, **Wedge Delay**, and **Sensitivity**), and sizing curve functions (**DAC**, **TCG**, and **DGS**).

To select the element to calibrate

1. Select **Wizard > Calibration > Type**.
2. In the **Type** list, choose the type of calibration that you want to perform (see Figure 5-1 on page 110 and Figure 5-2 on page 110).

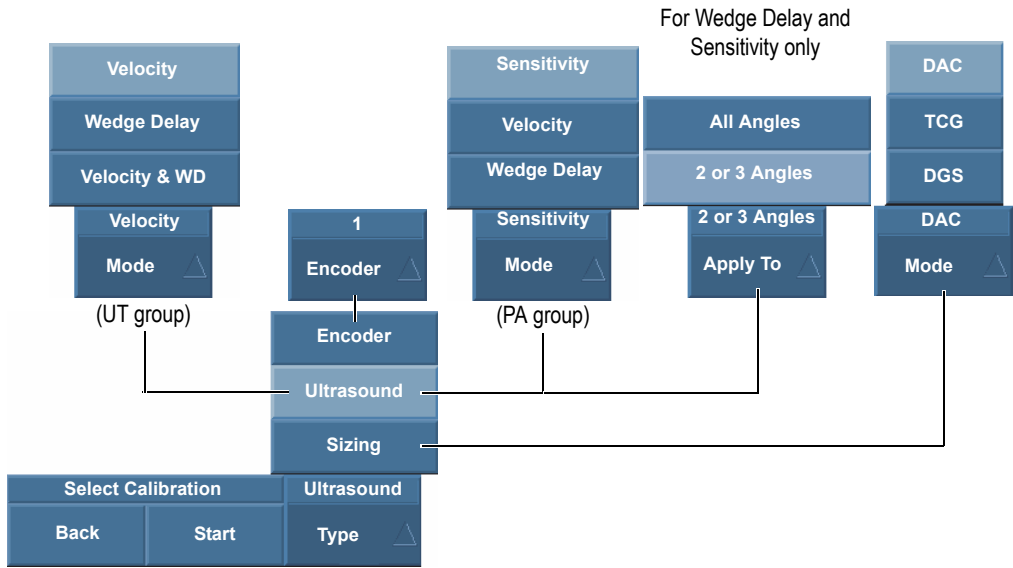


Figure 5-1 Choice of parameters to calibrate in Select Calibration

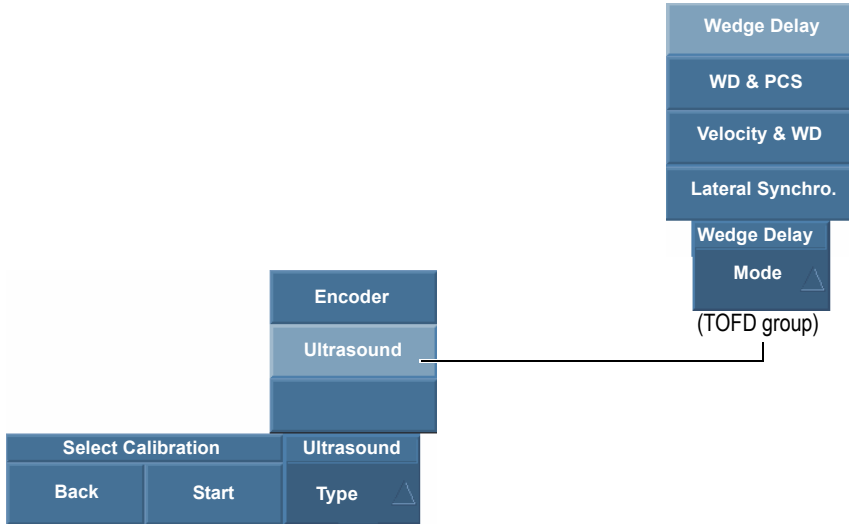


Figure 5-2 Choice of parameters to calibrate – TOFD group

- Depending on the choice you made in the previous step (see Figure 5-1 on page 110), select the item to calibrate in the **Encoder** or **Mode** list.

The **Apply To** parameter appears only for ultrasonic wedge delay and sensitivity calibrations. This parameter allows you to select whether the calibration applies to all focal laws or only to two or three selected angles/VPA.

IMPORTANT

You must perform the ultrasonic velocity calibration before the wedge delay calibration. The OmniScan instrument uses the ultrasonic velocity determination for the wedge delay calibration. If you attempt to calibrate the wedge delay first, you will receive a message warning you that the wedge delay calibration will be lost once you perform the ultrasonic velocity calibration.

- Select **Start** to move to the first step of the selected wizard.
The wizard guides you through the rest of the procedure.
-

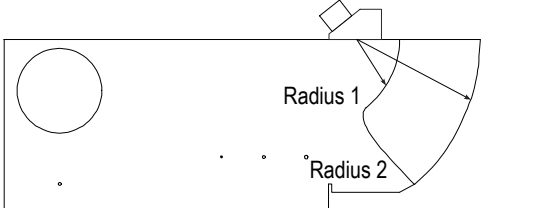
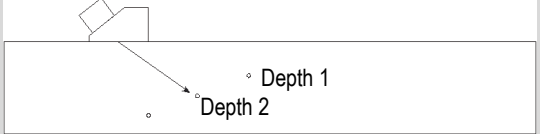
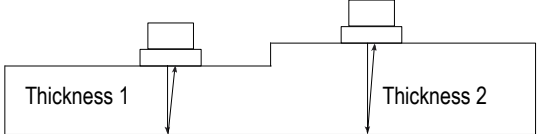
TIP

You can exit a Calibration wizard at any time by pressing the Cancel key. When exiting the wizard, the signal goes back to its original state (as before the calibration).

5.2 Reflector Types

Calibration procedures are performed using calibration blocks with different types of known reflectors. Table 18 on page 112 illustrates the types of probe, wedge, and calibration blocks used for each type of reflector.

Table 18 Reflector, probe, and calibration block types

Reflector type	Probe type	Probe, wedge and calibration block
Radius	Angle-beam	
Depth	Angle-beam	
Thickness	0 degree	

NOTE

In the wizard for ultrasound items, the UT mode is set as follows:

- **Sound Path** when you select **Echo Type = Radius**.
- **True Depth** when you select **Echo Type = Depth** or **Thickness**.

In the wizard, the default echo type depends on the gate geometry selected in the setup:

- If the geometry was set to **Sound Path**, the default **Echo type = Radius**.
- If the geometry was set to **Depth**, the default **Echo type = Depth**.

5.3 Scan Types

With a PA group, the ultrasound calibration wizards can apply to both sectorial and linear scan setups.

For a sectorial scan, the parameter for selecting a focal law is labeled **Angle**, while the parameter for a linear scan is labeled **VPA** (virtual probe aperture). In the procedures, these parameter labels are generically referred to as **Angle/VPA**.

5.4 Ultrasonic Calibration

The OmniScan MXU software features wizards to calibrate several ultrasonic aspects. With **Wizard > Calibration > Type = Ultrasound** selected, the **Mode** parameter provides the following choices:

Sensitivity (PA group only)

Used to calibrate the sensitivity for detecting a reference reflector. See “Sensitivity Calibration for a UT Group” on page 128 and “Sensitivity Calibration for a PA Group” on page 129 for information on the steps required to calibrate the ultrasonic sensitivity.

Velocity

Used to calibrate the velocity of the sound propagation in the material of the inspected part. See “Ultrasonic Velocity Calibration” on page 114 for information on the steps required to calibrate the ultrasonic velocity.

Wedge Delay

Used to calibrate the delay corresponding to the sound propagation within the wedge. You must perform the velocity calibration first, as it is needed to perform the wedge delay calibration. See “Wedge Delay Calibration for a UT Group” on page 117 and “Wedge Delay Calibration for a PA Group” on page 123 for information on the steps required to calibrate the wedge delay.

WD & PCS

Used to calibrate (with a single wizard) both the delay of the sound propagation within the wedge and the probe center separation.

Velocity & WD (UT group only)

Used to calibrate (with a single wizard) both the velocity of the sound propagation in the material of the inspected part and the delay corresponding to the sound propagation within the wedge. See “Velocity and Wedge Delay

Calibration for a UT Group” on page 121 for information on the steps required to calibrate both the velocity and the wedge delay.

5.4.1 Ultrasonic Velocity Calibration

The purpose of the ultrasonic velocity calibration is to measure the real velocity of ultrasonic waves in the material to be inspected. The calibration block must have two known reflectors and be made of the same material as the part to be inspected.

IMPORTANT

If an ultrasonic velocity calibration is required, it must be performed before the wedge delay calibration because the OmniScan instrument uses the ultrasonic velocity calibration result for the wedge delay calibration. If you calibrate the wedge delay first, a message appears, warning you that the wedge delay calibration will be lost if you perform the ultrasonic velocity calibration.

This procedure applies to the following groups, reflectors, and scans:

- UT and PA groups
- All reflector types (radius, depth, or thickness), referred to as the reflector
- Sectorial and linear scans (angle/VPA)

To calibrate the ultrasonic velocity

1. Place the probe on a calibration block, as shown in Table 18 on page 112.
2. Perform the following steps to start the velocity calibration wizard:
 - a) Select **Wizard > Calibration > Type = Ultrasound**.
 - b) Select **Mode = Velocity**.
 - c) Select **Start**.
3. In the **Set Thickness 1 and 2** step, proceed as follows:
 - a) Select the appropriate **Echo Type = Thickness**.
 - b) In **Thickness 1**, enter the known thickness that corresponds to the first signal on the A-scan time base.
 - c) In **Thickness 2**, enter the known thickness value that corresponds to the second signal on the A-scan time base.
 - d) Select **Next**.

4. In the **Select A-Scan** step (PA group), or **Set A-Scan** step (UT group), proceed as follows:
 - a) With a PA group selected, select **Angle/VPA** to set the reference angle/VPA for the calibration. When a UT group is selected, the **Angle/VPA** parameter is read-only.
Typically, use an **Angle/VPA** in the middle of the scan. For example, for a linear scan with 30 VPA, select the 15th in the series of 30.
 - b) Set the **Gain** so that the amplitude of the first signal reaches about 80 % of the full-screen height of the A-scan (see Figure 5-3 on page 115).
 - c) Set the **Start** and **Range** parameters until the two signals from the two reflectors on the A-scan are visible (see the example in Figure 5-3 on page 115).

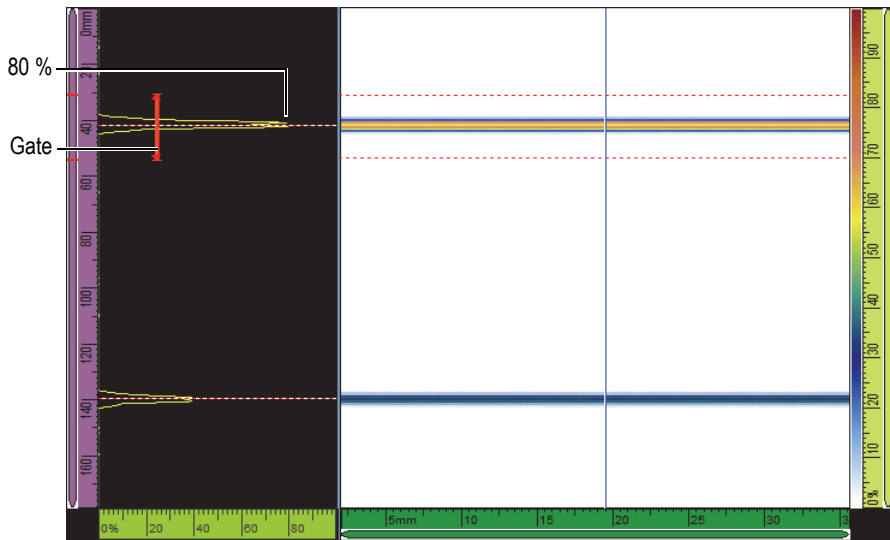


Figure 5-3 Example of reflector signals in a linear scan

- d) Position the probe to maximize the two signals. The probe must not move for the remainder of this procedure.
- e) Select **Next**.
5. In the **Set Gate A on Thickness 1** step, proceed as follows:
 - a) In the **Start** and **Width** parameters, enter the start and width of gate A so that the first signal crosses the gate in the middle, and so that the gate is wide

- enough to contain the complete width of the signal (see Figure 5-3 on page 115).
- b) Set the **Threshold** value between 20 % and 25 %.
 - c) Select **Get Position**.
6. In the **Set gate A on Thickness 2** step, proceed as follows:
 - a) In the **Start** and **Width** parameters, enter values to set gate A on the second signal, according to the same method used for the first signal (see Figure 5-4 on page 116).

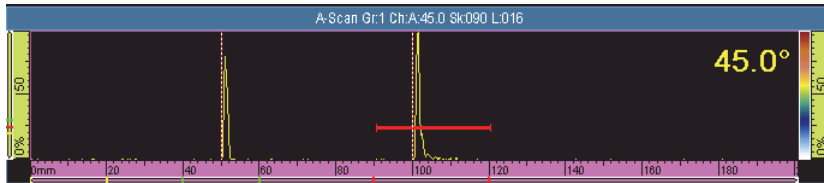


Figure 5-4 Setting the gate on the second signal

- b) Set the **Threshold** value to ensure that the signal crosses the gate.
 - c) Select **Get Position**.
7. In the **Accept** step, proceed as follows:
 - ◆ Select **Accept** if the calculated velocity shown in the **Material Velocity** parameter is reasonable for the calibration block material.

OR

Select **Restart** if you want to repeat the procedure.

The ultrasonic velocity is now calibrated. In the lower left section of the screen, the velocity calibration indicator (**V**) turns green (see Figure 5-5 on page 116).

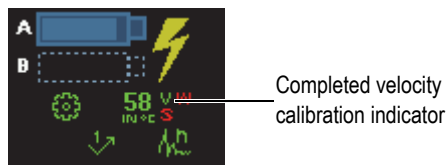


Figure 5-5 Velocity indicator (V) in green after velocity calibration

5.4.2 Wedge Delay Calibration for a UT Group

A wedge delay calibration is a process that enables you to identify the face of the wedge contacting the part, and that establishes a zero position for the entry surface of the part. The calibration procedure requires a calibration block with one known reflector.

IMPORTANT

The ultrasonic velocity calibration must be performed before the wedge delay calibration because the OmniScan instrument uses the ultrasonic velocity calibration result for the wedge delay calibration. If you calibrate the wedge delay first, a message appears, warning you that the wedge delay calibration will be lost if you perform the ultrasonic velocity calibration.

This procedure applies to the following groups and reflectors:

- UT groups only
- All reflector types (radius/depth/thickness), herein referred to as the reflector

To calibrate the wedge delay for a UT group

1. Place the probe on a calibration block, as shown in Table 18 on page 112.
2. Select **Wizard > Calibration**.
3. In the **Select Calibration** step, proceed as follows:
 - a) Select **Type = Ultrasound**.
 - b) Select **Mode = Wedge Delay**.
 - c) Select **Start**.
4. In the **Set Radius/Depth/Thickness A** step, proceed as follows:
 - a) Select **Echo Type**, and then choose the reflector type that you are using.
 - b) Select **Radius/Depth/Thickness A** and enter the radius/depth/thickness of the known reflector.
 - c) Select **Next**.
5. In the **Set Gate A on Radius/Depth/Thickness A** step, proceed as follows:
 - a) In the **Start** and **Width** buttons, enter the start and width of gate A, ensuring that the signal crosses the gate in the middle, and that the gate is wide enough to contain the complete width of the signal.

- b) Set the **Threshold** value to 20 % or 25 %.
- c) Select **Next**.
6. In the **Calibrate** step, proceed as follows:
 - a) Carefully move the probe back and forth along the calibration block and over the reflector to build a smooth signal envelope (see Figure 5-6 on page 118).
 - b) If needed, select **Gain** to adjust the gain applied to the signal.

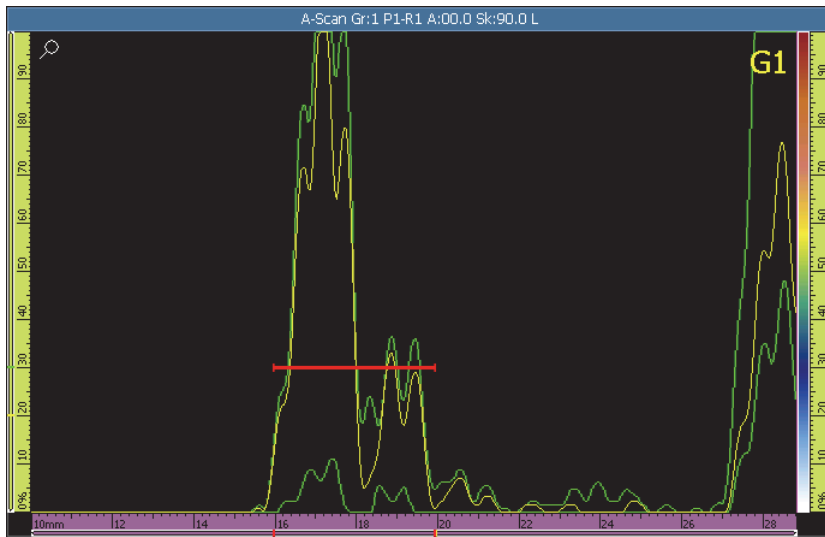


Figure 5-6 Building the envelope for the wedge delay calibration

- c) Select **Calibrate**.
7. In the **Accept** step, proceed as follows:
 - ◆ If the calibration is not satisfactory, select **Restart** to restart the calibration process.
 - OR
 - Select **Accept**.

The wedge delay is now calibrated. In the lower left section of the screen, the wedge delay calibration indicator (**W**) turns green (see Figure 5-7 on page 119).

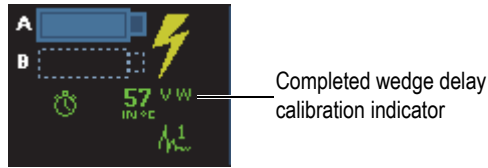


Figure 5-7 Wedge delay indicator (W) after calibration for UT group

5.4.3 Wedge Delay and Probe Center Separation Calibration (TOFD Only)

Calibration is usually done in analysis mode, after the inspection is complete. If the lateral wave has not been straightened or if there is significant distortion from the acquisition, you should perform the **Calibration** wizard for each flaw.

To perform a WD and PCS calibration

1. Display the TOFD group in the single group layout by selecting **A-B** in the Layouts section of the title bar.
2. Select **Wizard > Calibration**.
3. In the **Select Calibration** step, proceed as follows:
 - a) Select **Type = Ultrasound**.
 - b) Select **Mode = WD & PCS**.
 - c) Select **Start**.
4. In the **Select Reference A-Scan** step, proceed as follows:
 - a) Position the blue Data cursor on the B-scan as close to the flaw as possible.
 - b) Select a data point that has a clearly identifiable lateral wave and back wall echo that is not obstructed by the flaw.

NOTE

If the reference A-scan is too far away from the flaw, the sizing accuracy is reduced because the lateral wave and/or back wall appears at a different TOF.

- c) Select **Next**.
 5. In the **Set Part** step, proceed as follows:
 - a) Make sure that the component **Thickness** and **CSC Mode** parameters that were entered during the group wizard creation are correct.
-

b) Select **Next**.

6. In the **Set cursor position and depth** step, proceed as follows:

NOTE

The wizard for the WD and PCS calibration option requires that two targets of known depth be identified with the UT axis cursors. The default targets are the lateral wave and the back wall that were entered in the previous step.

- a) Position the Reference cursor (red) on the lateral wave.
- b) Position the Measurement cursor (green) on the back wall.
- c) Zoom the area of interest for precision cursor placement and most accurate measurements.
- d) Select **Cal. WD & PCS**.
A time ruler (in time [μ s]) and a magenta TOFD ruler (in depth [mm or in.]) appear on the UT axis.

7. In the **Accept** step, proceed as follows:

- a) Observe the calculated **PCS**. It should be very close to the actual measured PCS.
- b) Note the **Wedge Delay** value as a reference for future use with the same probes and wedges.
- c) Make sure that the UT axis cursors are properly positioned on component surface and back wall.
- d) Select **Accept** if the values are correct or **Restart** if the values are not as intended.

NOTE

The **Material Velocity** is a fixed value and is not calculated in this wizard option (WD & PCS).

5.4.4 Quick TOFD Calibration

In Analysis mode, you can perform a quick calibration of the TOFD signal, which avoids several wizard steps. You can use one of the following two methods, depending on your part:

1. A quick calibration on a part that has a uniform thickness.
2. A quick calibration on a part whose thickness varies; in this case, the TOFD calibration must be repeated several times.

To perform a quick TOFD calibration if the part thickness is uniform

1. Position the Data cursor on the B-scan to obtain the desired A-scan signal.
2. Position the Reference cursor on the first reflector.
3. Position the Measurement cursor on the second reflector.
4. Tap and hold the B-scan display area, and then select **Wedge Delay & PCS**.

To perform a quick TOFD calibration if the part thickness varies

1. In the **Wizard > Calibration** menu, select the desired calibration from the **Mode** list.
2. Select the reference A-scan.
3. Enter the required parameters (**Thickness** and/or **PCS**).
4. Position the Reference cursor on the first reflector.
5. Position the Measurement cursor on the second reflector.
6. Select **Calibrate Current**.

NOTE

The quick TOFD calibration methods are designed to calibrate on the lateral wave and the back wall signals. To calibrate on other reflectors, follow the steps in the wizard.

5.4.5 Velocity and Wedge Delay Calibration for a UT Group

For UT groups, the **Velocity & WD** wizard combines the calibration for both the velocity and the wedge delay. The calibration procedure requires a calibration block with two known reflectors.

The purpose of the ultrasonic velocity calibration is to measure the real velocity of ultrasonic waves in the material to be inspected.

A wedge delay calibration is performed to identify the face of the wedge in contact with the part, and to establish a zero position for the entry surface of the part.

This procedure applies to the following groups and reflectors:

- UT groups
- All reflector types (radius/depth/thickness), herein referred to as the reflector

To calibrate both the velocity and the wedge delay for a UT group

1. Place the probe on a calibration block, as shown in Table 18 on page 112.
2. Select **Wizard > Calibration**.
3. In the **Select Calibration** step, proceed as follows:
 - a) Select **Type = Ultrasound**.
 - b) Select **Mode = Velocity & WD**.
 - c) Select **Start**.
4. In the **Set Radius/Depth/Thickness 1 and 2** step, proceed as follows:
 - a) Select the appropriate **Echo Type = Radius, Depth, or Thickness**.
 - b) In **Radius/Depth/Thickness 1**, enter the known radius/depth/thickness value that corresponds to the first reference reflector.
 - c) In **Radius/Depth/Thickness 2**, enter the known radius/depth/thickness value that corresponds to the second reference reflector.
 - d) Select **Next**.
5. In the **Set Gate A on Radius/Depth/Thickness 1** step, proceed as follows:
 - a) Select **Start** and **Width**, and then enter the start and width of gate A so that the signal envelope of the first reference reflector crosses the gate in the middle, and so that the gate is wide enough to contain the complete width of the signal envelope.
 - b) Set the **Threshold** value to 20 %, or 25 %.
 - c) Select **Get Position**.
6. In the **Set Gate A on Radius/Depth/Thickness 2** step, proceed as follows:
 - a) Select **Start** and **Width**, and then enter the start and width of gate A, ensuring that the signal envelope of the second reference reflector crosses the gate in the middle, and that the gate is wide enough to contain the complete width of the signal envelope.
 - b) Set the **Threshold** value to 20 % or 25 %.
 - c) Select **Get Position**.
7. In the **Accept** step, the read-only **Material Velocity** and **Wedge Delay** parameters show the calibrated values.

- ◆ If the calibration is not satisfactory, select **Restart** to restart the calibration process.

OR

Select **Accept**.

The velocity and the wedge delay are now calibrated. At the lower left section of the screen, the velocity (**V**) and the wedge delay (**W**) calibration indicators turn green (see Figure 5-8 on page 123).

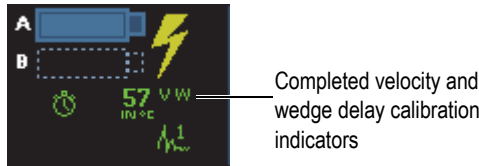


Figure 5-8 The velocity (**V**) and wedge delay (**W**) indicators

5.4.6 Wedge Delay Calibration for a PA Group

A wedge delay calibration is a process that identifies the face of the wedge in contact with the part, and establishes a zero position for the entry surface of the part for all the focal laws. The calibration procedure requires a calibration block with one known reflector.

For PA groups, the wedge delay calibration can be performed for all focal laws, or for two or three selected laws.

IMPORTANT

The ultrasonic velocity calibration must be performed before the wedge delay calibration because the OmniScan instrument uses the ultrasonic velocity calibration result for the wedge delay calibration. If you calibrate the wedge delay first, you will get a message warning you that the wedge delay calibration will be lost if you perform the ultrasonic velocity calibration.

5.4.6.1 Calibrating the Wedge Delay for All Focal Laws

The following procedure describes how to calibrate the wedge delay for all focal laws. This procedure applies to the following groups, reflectors and scans:

- PA groups only
- All reflector types (radius/depth/thickness), herein referred to as the reflector
- Sectorial and linear scans (angle/VPA)

To calibrate the wedge delay for all focal laws

1. Place the probe on a calibration block, as shown in Table 18 on page 112.
2. Select **Wizard > Calibration**.
3. In the **Select Calibration** step, proceed as follows:
 - a) Select **Type = Ultrasound**.
 - b) Select **Mode = Wedge Delay**.
 - c) Select **Apply To = All Angles/VPA**.
 - d) Select **Start**.
4. In the **Set Radius/Depth/Thickness A** step, proceed as follows:
 - a) Select **Echo Type**, and then select the reflector type that you are using.
 - b) Select **Radius/Depth/Thickness A**, and enter the radius/depth/thickness of the known reflector.
 - c) For the **Tolerance** parameter, keep the default value.
 - d) Select **Next**.
5. In the **Set Section** step, proceed as follows:
 - a) If needed, use the **Last Angle/VPA** parameter to define a section smaller than the whole sector, thus enabling the calibration to be performed separately for two or more sections of the sector.
 - b) Select **Next**.
6. In the **Set Gate A on Radius/Depth/Thickness A** step, proceed as follows:
 - a) In the **Start** and **Width** buttons, enter the start and width of gate A, ensuring that the signal crosses the gate in the middle, and that the gate is wide enough to contain the complete width of the signal.
 - b) Set the **Threshold** value to 20 % or 25 %.
 - c) Select **Next**.
7. In the **Calibrate and Accept** step, proceed as follows:

- a) Move the probe forward and backward over the calibration reflector to build an envelope (see Figure 5-9 on page 125).

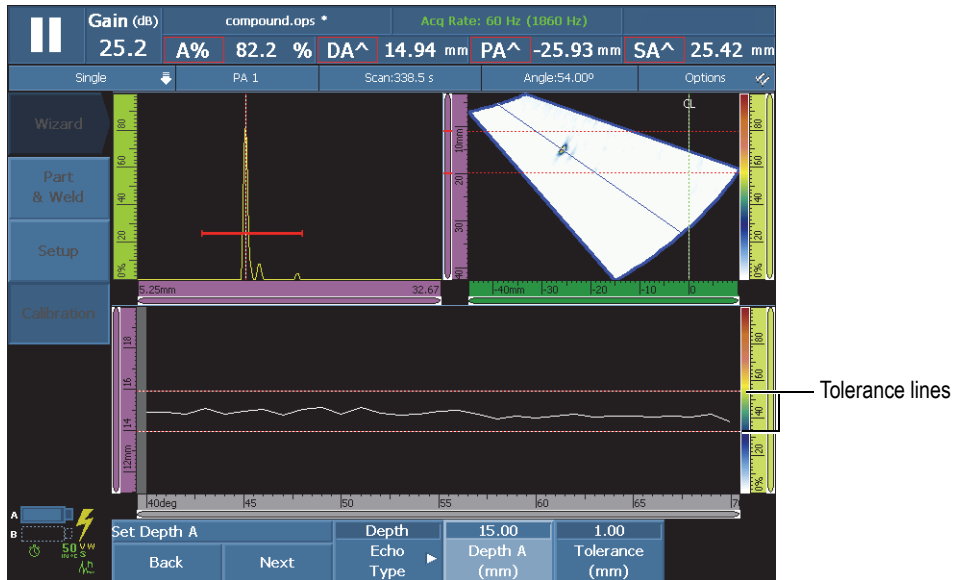


Figure 5-9 Building the envelope for the wedge-delay calibration

- b) Select **Calibrate**.
- c) Again, move the probe forward and backward over the calibration reflector to verify whether the envelope is within tolerances.

When the calibration is correct, the envelope is between the two red dotted lines.

- d) If the calibration is satisfactory, select **Accept**.

OR

If the calibration is incorrect, select **Clear Env.** to clear the envelope, and then either make a new pass over the reflector, or select **Restart** to restart the calibration and, when you have achieved a correct calibration, select **Accept**.

The wedge delay is now calibrated. In the lower left section of the screen, the wedge delay calibration indicator (**W**) turns green (see Figure 5-10 on page 126).

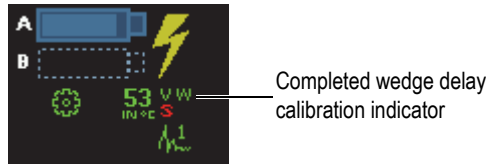


Figure 5-10 Wedge delay indicator (W) after calibration for all focal laws

5.4.6.2 Calibrating the Wedge Delay for Two or Three Focal Laws in a PA Group

This section describes how to calibrate the wedge delay for two or three specific angles/VPA.

This procedure applies to the following groups, reflectors and scans:

- PA groups only
- All reflector types (radius/depth/thickness), herein referred to as the reflector
- Sectorial or linear scans (angle/VPA)

To calibrate the wedge delay for two or three focal laws in a PA group

1. Place the probe on a calibration block, as shown in Table 18 on page 112.
2. Select **Wizard > Calibration**.
3. In the **Select Calibration** step, proceed as follows:
 - a) Select **Type = Ultrasound**.
 - b) Select **Mode = Wedge Delay**.
 - c) Select **Apply To = 2 or 3 Angles/VPA**.
 - d) Select **Start**.
4. In the **Select Angles/VPA to Calibrate** step, proceed as follows:
 - a) Select **Qty**, and then select the number of angles/VPA to calibrate (**2 or 3**).
 - b) Select **Angle/VPA 1** to specify the first angle/VPA to calibrate.
 - c) Select **Angle/VPA 2** to specify the second angle/VPA to calibrate.
 - d) If applicable, select **Angle/VPA 3** to specify the third angle/VPA to calibrate.
 - e) Select **Next**.
5. In the **Set Radius/Depth/Thickness A** step, proceed as follows:
 - a) Select **Echo Type**, and then choose the reflector type that you are using.

- b) Select **Radius/Depth/Thickness A**, and enter the radius/depth/thickness of the known reflector.
 - c) Select **Next**.
6. In the **Set Gate A on Radius/Depth/Thickness A** step, proceed as follows:
 - a) In the **Start** and **Width** buttons, enter the start and width of gate A, ensuring that the signal crosses the gate in the middle, and that the gate is wide enough to contain the complete width of the signal.
 - b) Set the **Threshold** value to 20 % or 25 %.
 - c) Select **Next**.
7. In the **Calibrate** step (first angle/VPA), proceed as follows:
 - a) Move the probe forward and backward over the reference reflector to build an envelope (see Figure 5-11 on page 127).

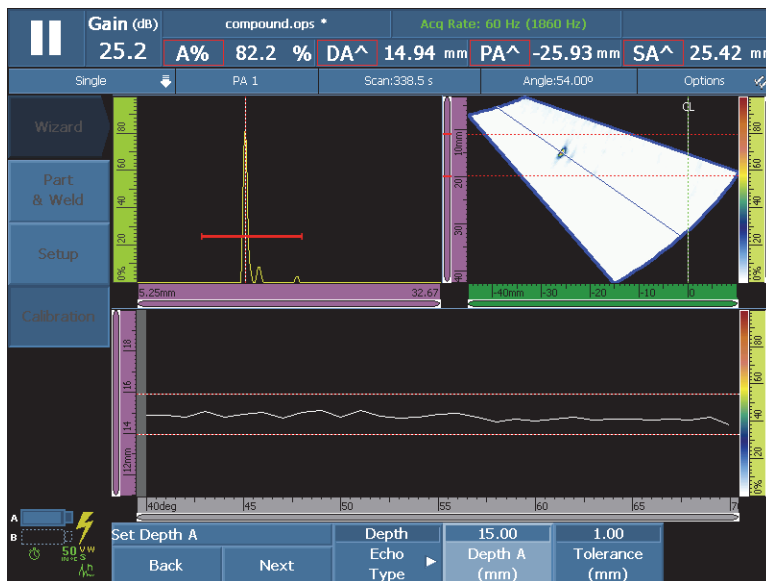


Figure 5-11 Building the envelope for the first angle/VPA

- b) Select **Calibrate**.
8. Repeat step 7 for the second and, if applicable, the third angle/VPA.
9. In the **Accept** step, proceed as follows:

- ◆ If the calibration is not satisfactory, select **Restart** to restart the calibration process.

OR

Select **Accept**.

The wedge delay is now calibrated for two or three focal laws. In the upper-left section of the screen, the wedge delay calibration indicator (W_p) turns green (see Figure 5-12 on page 128).

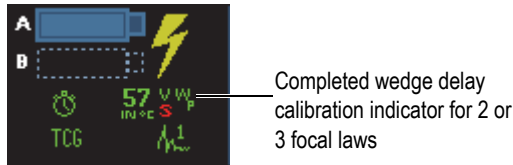


Figure 5-12 Wedge delay indicator (W_p) after calibration for 2 or 3 focal laws

5.4.7 Sensitivity Calibration for a UT Group

The sensitivity calibration for a UT group does not require a wizard.

This procedure applies to the following groups and reflectors:

- UT groups only
- All reflector types (radius/depth/thickness), herein referred to as the reflector

To calibrate a UT probe for sensitivity

1. Position the probe to get a maximum signal from the reference reflector.
2. Position gate A over the signal from the reference reflector.
3. Select **UT Settings > Advanced > Ref. Amplitude**, and then enter the desired reference amplitude value (80 % for the example in Figure 5-13 on page 129).
4. Select **Set to XX.X%** to perform the sensitivity calibration.

Figure 5-13 on page 129 shows the reference reflector signal before and after using the **Set to XX.X%** command.

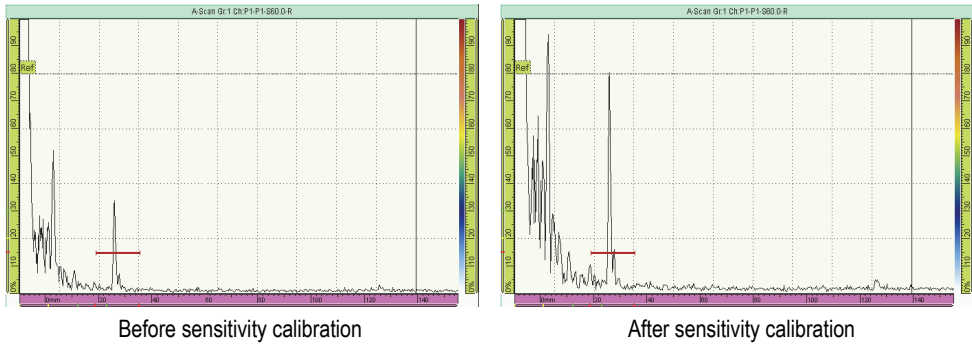


Figure 5-13 Reference reflector signal before and after sensitivity calibration

5.4.8 Sensitivity Calibration for a PA Group

The sensitivity calibration for a PA group normalizes the gain for all focal laws to ensure that all focal laws produce a similar amplitude signal for the reference reflector. The calibration procedure requires a calibration block with one known reflector.

The sensitivity calibration can be performed for all focal laws, or for two or three selected laws.

TIP

It is recommended to perform the sensitivity calibration before performing other types of calibrations. In particular, this may make it easier to perform subsequent ultrasonic velocity and wedge delay calibrations.

5.4.8.1 Calibrating the Sensitivity for All Focal Laws

The following procedure describes how to calibrate the sensitivity for all focal laws.

This procedure applies to the following groups, reflectors and scans:

- PA groups only
- All reflector types (radius/depth/thickness), herein referred to as the reflector

- Sectorial or linear scans (angle/VPA)

To calibrate the sensitivity for all focal laws

1. Place the probe on a calibration block, as shown in Table 18 on page 112.
2. Select **Wizard > Calibration**.
3. In the **Select Calibration** step, proceed as follows:
 - a) Select **Type = Ultrasound**.
 - b) Select **Mode = Sensitivity**.
 - c) Select **Apply To = All Angles/VPA**.
 - d) Select **Start**.
4. In the **Set Reference Amplitude** step, proceed as follows:
 - a) Select **Ref. Amplitude** to specify the level at which the peak amplitude of the signal crossing gate A is plotted when you select the **Set to XX.X%** parameter.
 - b) Select **Tolerance** to specify the tolerance for the measurement of the reference reflector.
 - c) Select **Next**.
5. In the **Set Section** step, proceed as follows:
 - a) If needed, use the **Last Angle/VPA** parameter to define a section smaller than the whole sector, thus enabling the calibration to be performed separately for two or more sections of the sector.
 - b) Select **Next**.
6. In the **Set Gate A on Echo A** step, proceed as follows:
 - a) In the **Start** and **Width** buttons, enter the start and width of gate A, ensuring that the signal crosses the gate in the middle, and that the gate is wide enough to contain the complete width of the signal.
 - b) Set the **Threshold** value to 20 % or 25 %.
 - c) Select **Next**.
7. In the **Set Compensation Gain** step, consider the following information and proceed as necessary:

You need to use the compensation gain in cases where the signal from the focal laws at the periphery of the scan is significantly weaker than that of the focal laws in the middle of the scan. In such case, the amplitude of the focal laws in the periphery of the scan is below the minimum sensitivity in the graph. The calibration wizard generates an error message when you attempt to complete the calibration. The compensation gain flattens the amplitude profile of the focal laws, allowing the profile to be fully visible in the sensitivity graph.

- a) If needed, select **Comp. Gain** to specify a compensation gain value that makes the focal law amplitude profile fully visible in the sensitivity graph.
 - b) Select **Apply** to make the compensation gain value effective.
 - c) If needed, select **Gain** to adjust the gain applied to the signal.
 - d) If needed, select **Clear Env.** to clear the signal envelope and to start building a new one.
 - e) Select **Next**.
8. In the **Calibrate and Accept** step, proceed as follows:
- a) Carefully move the probe back and forth along the calibration block and over the reflector to build a smooth signal envelope with values in the 0–100 % range for all angles/VPA (see Figure 5-14 on page 131).

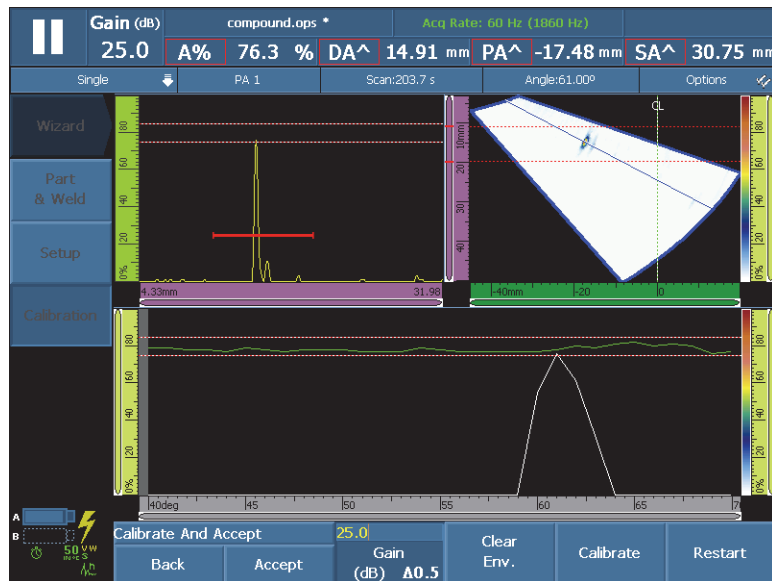


Figure 5-14 Building the envelope for the sensitivity calibration

- b) Select **Calibrate**.
- c) Again, move the probe back and forth over the calibration reflector to determine whether the envelope is within the tolerances.
If the calibration is correct, the envelope is between the two red dotted lines.
- d) If the calibration is satisfactory, select **Accept**.

OR

If the calibration is incorrect, select **Clear Env.** to clear the envelope, and then either make a new pass over the reflector or select **Restart** to restart the calibration and, when you have achieved a correct calibration, select **Accept**.

The sensitivity is now calibrated. In the upper-left section of the screen, the sensitivity calibration indicator (S) turns green (see Figure 5-15 on page 132).

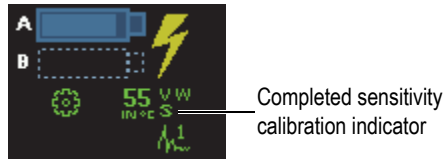


Figure 5-15 Sensitivity indicator (S) in green after calibration

5.4.8.2 Calibrating the Sensitivity for Two or Three Focal Laws

The following procedure describes how to calibrate the sensitivity for two or three specific angles/VPA.

This procedure applies to the following groups, reflectors, and scans:

- PA groups only
- All reflector types (radius/depth/thickness), herein referred to as the reflector
- Sectorial or linear scans (angle/VPA)

To calibrate the sensitivity for two or three focal laws

1. Place the probe on a calibration block, as shown in Table 18 on page 112.
2. Select **Wizard > Calibration**.
3. In the **Select Calibration** step, proceed as follows:
 - a) Select **Type = Ultrasound**.
 - b) Select **Mode = Sensitivity**.
 - c) Select **Apply To = 2 or 3 Angles/VPA**.
 - d) Select **Start**.
4. In the **Set Reference Amplitude** step, proceed as follows:
 - a) Select **Ref. Amplitude** to specify the level at which the peak amplitude of the signal crossing gate A is plotted when you select the **Set to XX.X%** parameter.

- b) Select **Tolerance** to specify the tolerance for the measurement of the reference reflector.
 - c) Select **Next**.
5. In the **Select Angles/VPA to Calibrate** step, proceed as follows:
- a) Select **Qty.**, and then select the number of angles/VPA to calibrate (**2** or **3**).
 - b) Select **Angle/VPA 1** to specify the first angle/VPA to calibrate.
 - c) Select **Angle/VPA 2** to specify the second angle/VPA to calibrate.
 - d) If applicable, select **Angle/VPA 3** to specify the third angle/VPA to calibrate.
 - e) Select **Next**.
6. In the **Set Gate A on Echo A** step, proceed as follows:
- a) In the **Start** and **Width** buttons, enter the start and width of gate A so that the signal crosses the gate in the middle and so that the gate is wide enough to contain the complete width of the signal.
 - b) Set the **Threshold** value to 20 % or 25 %.
 - c) Select **Next**.
7. In the **Calibrate (first angle/VPA)** step, proceed as follows:
- a) Move the probe forward and backward over the reference reflector to build an envelope (see Figure 5-16 on page 134).

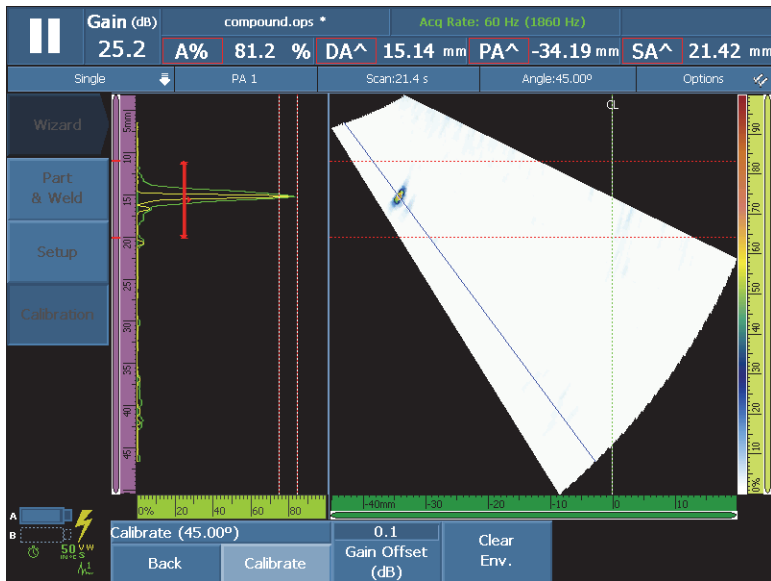


Figure 5-16 Building the envelope for the sensitivity calibration of the first angle

- b) Select **Calibrate**.
8. Repeat step 7 for the second and, if applicable, the third angle/VPA.
 9. In the **Accept** step, proceed as follows:
 - ◆ If the calibration is not satisfactory, select **Restart** to restart the calibration process.
 - OR
 - Select **Accept**.

The sensitivity is now calibrated. In the upper-left section of the screen, the sensitivity calibration indicator (S_i) turns green (see Figure 5-17 on page 134).

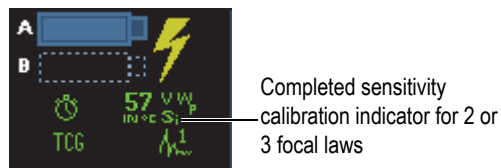


Figure 5-17 Sensitivity indicator (S_i) after calibration for 2 or 3 focal laws

5.5 Sizing Calibration

The OmniScan instrument offers a number of sizing functions (DAC, TCG, DGS, and AWS). A sizing function provides a method to display the real size of a reflector independently of its location in the part.

See “Sizing Menu” on page 286 for more details on the sizing functions.

5.5.1 DAC Calibration

This section describes how to calibrate the DAC (distance-amplitude correction) sizing function. To perform this procedure, you need a calibration block with a number of identical holes at known depths or with multiple known thicknesses.

NOTE

Because the DAC curve(s) will only be seen in the A-scan, they are more appropriate for conventional UT inspection. If using PA, a TCG is more appropriate, because the amplitude correction will be seen in the S-scan as well as the A-scan.

This procedure applies to the following groups, reflectors, and scans:

- UT and PA groups
- Depth or thickness reflector types, herein referred to as the reflector
- Sectorial or linear scans (angle/VPA)

To perform a DAC calibration

1. Place the probe on a calibration block, as shown in Figure 5-18 on page 135.

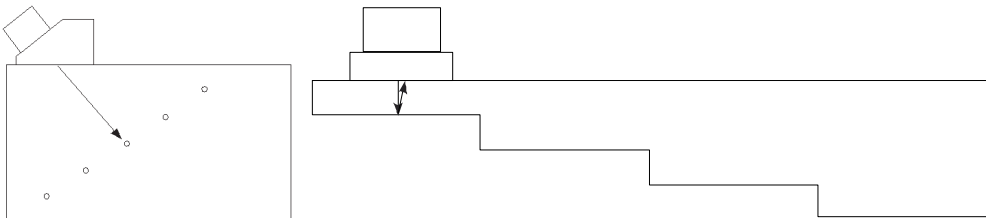


Figure 5-18 Calibration blocks for the DAC calibration

2. Select **UT Settings > General > Range**, and then set the value to see the signal from the farthest reflector (or of the thickest section).
3. To start the wizard, proceed as follows:
 - a) Select **Wizard > Calibration > Type = Sizing**.
 - b) Select **Mode = DAC**.
 - c) Select **Start**.
4. In the **Set Type** step, proceed as follows:
 - a) Select **Compliance** to select a customized sizing function, or a predefined sizing function compliant with the JIS or ASME code (see “Parameters for DAC, Linear DAC, and TCG Curves” on page 290 for more information).
Selecting a preset automatically sets specific sizing function parameters, and sets them as read-only.
 - b) If you selected **Compliance = Custom**, proceed as follows:
 - (1) Select **Curve Type** to specify the type of interpolation between the points of the curve. The following choices are available:
 - Straight**: linear interpolation plotting straight lines between the DAC points.
 - Logarithmic**: logarithmic interpolation between the DAC points. For the DAC, the interpolation is calculated based on two reference points. For the linear DAC, the interpolation is calculated based on the specified attenuation.
 - Polynomial**: third order polynomial interpolation between the DAC points.
 - (2) Select **Curve Qty.** to define the number of sizing curves.
This parameter is read-only for sizing curves compliant with a standard.
 - c) Select **Next**.
5. In the **Set Reference Point** step, proceed as follows:
 - a) Select **Position** to specify the position of the first point on the ultrasound axis.
 - b) Select **Ref. Amplitude** to specify the level at which the peak amplitude of the signal crossing gate A is plotted when you select the **Set to XX.X%** parameter.
 - c) Select **Next**.
6. In the **Set Section** step (PA group only), proceed as follows:
 - a) If you want to perform the calibration separately for two or more sections of the scan sector, edit the values of the **First Angle/VPA** and **Last Angle/VPA** to define the first section.
 - b) Select **Next**.

7. In the **Set Gate A on Echo** step, proceed as follows:
 - a) In the **Start** and **Width** buttons, enter the start and width of gate A, ensuring that the signal from the first reflector signal crosses the gate in the middle, and that the gate is wide enough to contain the complete width of the signal (see the **A-Scan** view in Figure 5-19 on page 137).
 - b) Set the **Threshold** value so that the signal crosses the gate on every aperture, as seen in the **Calibration: DAC** view of Figure 5-19 on page 137.

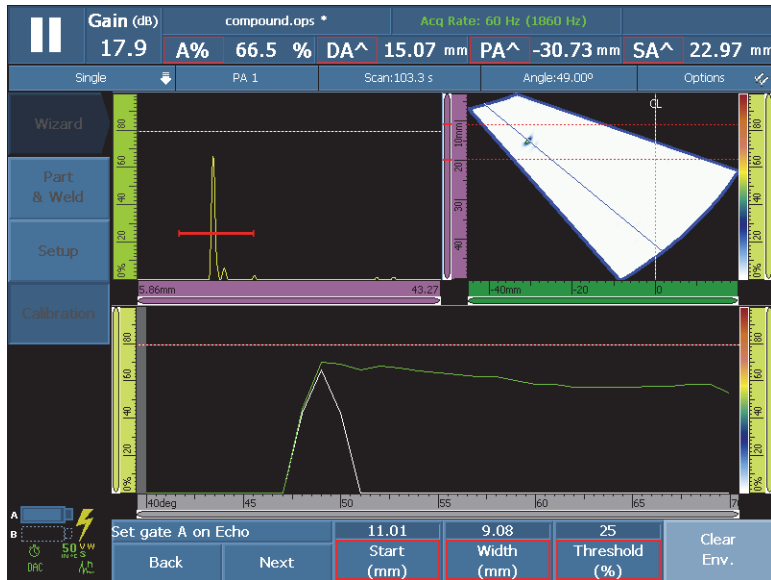


Figure 5-19 Set Gate A on Echo for a PA group

- c) Select **Next**.
8. In the **Add point** step, proceed as follows:
 - a) Carefully move the probe back and forth along the calibration block and over the reflector to build a smooth signal envelope with a maximum amplitude below the **Ref. Amplitude** value.
 - b) If needed, choose **Gain** or **Gain Offset** to modify the gain or the gain offset applied to the signal.
 - c) If needed, select **DAC Gain** to modify the gain applied to the DAC curves.
 - d) Select **Add Point**.

9. In the **Accept DAC** step, proceed as follows:
 - ◆ Select **Next Point** to add other DAC points, and to build a complete DAC curve. The wizard repeats steps 7 through 9, for each new point. The DAC curve can have up to 32 points.
- OR
- Select **Accept DAC** when the DAC curve is finished.

TIP

You can toggle between the DAC and the TCG sizing curve types by selecting **Sizing > Type = TCG**.

5.5.2 TCG Calibration

This section describes how to calibrate the TCG (time-corrected gain) sizing function. To perform this procedure, you need a calibration block with a number of identical holes at known depths or with multiple known thicknesses.

This procedure applies to the following groups, reflectors, and scans:

- UT and PA groups
- Depth or thickness reflector types, herein referred to as the reflector
- Sectorial or linear scans (angle/VPA)

To perform a TCG calibration

1. Place the probe on a calibration block, as shown in Figure 5-20 on page 138.

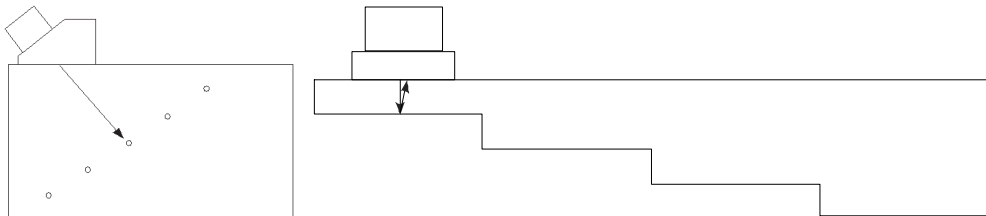


Figure 5-20 Calibration blocks for the TCG calibration

2. Select **UT Settings > General > Range**.
3. Set the value to see the signal of the deepest reflector (or of the thickest section).
4. To start the wizard, proceed as follows:
 - a) Select **Wizard > Calibration > Type = Sizing**.
 - b) Select **Mode = TCG**.
 - c) Select **Start**.
5. In the **Set Type** step, proceed as follows:
 - a) Select **Compliance** to select a customized sizing function, or a predefined sizing function compliant with the JIS or ASME code (see “Parameters for DAC, Linear DAC, and TCG Curves” on page 290 for more information).
Selecting a preset automatically sets specific sizing function parameters, and sets them as read-only.
 - b) If **Compliance = Custom** is selected, choose **Curve Qty.** to define the number of sizing curves.
This parameter is read-only for sizing curves compliant with a standard.
 - c) Select **Next**.
6. In the **Set Curves Offset** step (which only appears if you selected **Compliance = Custom** and a **Curve Qty.** value greater than one), proceed as follows:
 - a) Choose the **Curve n Offset** parameters to set the offset relative to the main curve for each secondary curve.
 - b) Select **Next**.
7. In the **Set Reference Amplitude** step, proceed as follows:
 - a) Select **Start** to adjust the position of gate A, ensuring that the signal of the echo crosses gate A.
 - b) Select **Ref. Amplitude** to specify the level at which the peak amplitude of the signal crossing gate A is plotted when you select the **Set to XX.X%** parameter.
 - c) Select **Set to XX.X%** to set the reference amplitude.
 - d) Select **Tolerance** to specify the tolerance for the measurement of the reference reflector.
 - e) Select **Next**.
8. In the **Set Section** step (PA group only):
 - a) If you want to perform the calibration separately for two or more sections of the scan sector, edit the values of the **First Angle/VPA** and **Last Angle/VPA** to define the first section.
 - b) Select **Next**.
9. In the **Set A-Scan** step, proceed as follows:

- a) Select **Gain** when you need to adjust the gain applied to signal.
 - b) Select **Start** when you need to adjust the A-scan start position.
 - c) Select **Range** when you need to adjust the A-scan range.
 - d) Select **Next**.
10. In the **Set Gate A on Echo** step, proceed as follows:
- a) In the **Start** and **Width** buttons, enter the start and width of gate A, ensuring that the signal from the first reflector signal crosses the gate in the middle, and that the gate is wide enough to contain the complete width of the signal (see the **A-Scan** view in Figure 5-21 on page 140).
 - b) Set the **Threshold** value so that the signal crosses the gate on every aperture, as seen in the **Calibration: TCG** view of Figure 5-21 on page 140.

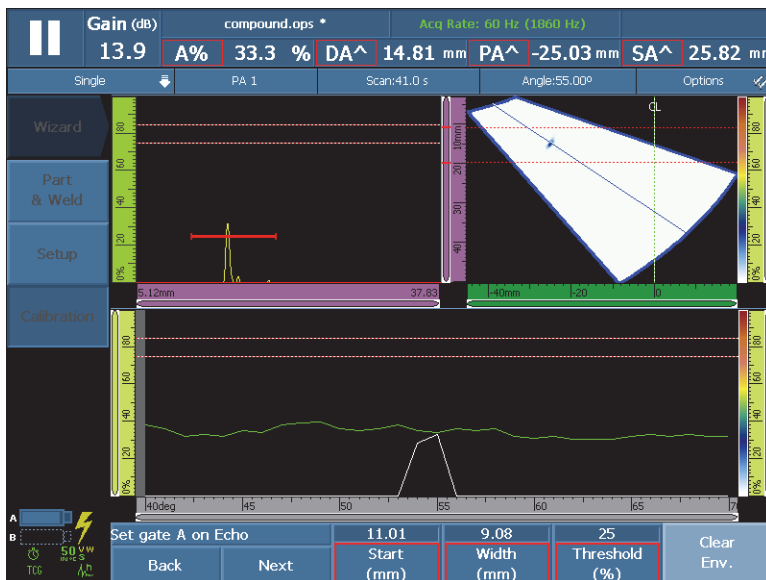


Figure 5-21 Set Gate A on Echo for a PA group

- c) Select **Next**.
11. In the **Add point** step, proceed as follows:
- a) Carefully move the probe back and forth along the calibration block and over the reflector to build a smooth signal envelope with a maximum amplitude below the **Ref. Amplitude** value.

- b) If needed, choose **Gain** or **Gain Offset** to modify the gain or the gain offset applied to the signal.
 - c) If needed, select **TCG Gain** to modify the gain applied to the TCG curves.
 - d) Select **Add Point**.
12. In the **Accept** step, proceed as follows:
- ◆ Select **Next Point** to add other TCG points, and to build a complete TCG curve. The wizard repeats steps 8 through 12 for each new point. The TCG curve can have up to 32 points.
- OR
- Select **Accept TCG** when the TCG curve is finished.
-

TIP

You can toggle between the DAC and the TCG sizing curve types by selecting **Sizing > Type = DAC**.

5.5.3 DGS Calibration

The distance gain size (DGS) method is used to size reflectors based upon a calculated DGS curve for a given transducer, material, and known reflector size.

The main DGS curve represents the signal amplitude of an equivalent flat-bottom hole (FBH) reflector of a specified size. The DGS method only requires a single reference reflector to create a DGS curve for flaw sizing. This is very different from the DAC and TCG methods, which require multiple, representative defects at various depths within a part to create a curve for flaw sizing.

All data needed to build a DGS/AVG curve is obtained from the PA probe ID and wedge information. You can use the DGS calibration wizard to quickly set up and easily evaluate flaw size.

IMPORTANT

The DGS method has been designed to meet ISO 16827 requirements. It is extremely important that you are familiar with this and other standards. You must be qualified according to your local standards to properly use this instrument function. Because the sizing curves are calculated using many variables, a proper instrument setup is

required to ensure accurate results. In a standard sectorial scan, the MX2/SX series instrument applies the DGS curve at 0° to 70° focal laws. For linear scans, the DGS curve is applied at the angle of the scan.

A DGS calibration has the following requirements:

- A DGS-compatible probe and wedge
- A DGS-compatible setup (probe, wedge, angles)
- A calibration block with a known reflector

A DGS calibration applies to the following modes:

- Conventional UT in pulse-echo mode at an angle (45°, 60°, or 70°)
- Conventional UT in pulse-echo mode at 0°
- Conventional UT in pitch-catch mode at 0°
- Phased array in pulse-echo mode at an angle (from 0° to 70°)

To perform the DGS calibration

NOTE

Before using the DGS calibration wizard, the **250 %** mode must be turned off, and the focusing depth should be set to **Unfocused**. Also note that RF mode is not available when DGS mode is activated.

1. Start the DGS calibration wizard:
 - a) Select **Wizard > Calibration > Type = Sizing**.
 - b) Select **Mode = DGS**.
 - c) Select **Start**.
 2. In the **Set Elevation** step, proceed as follows:
 - a) Enter the probe elevation value. (The value is set to 9 mm by default.)
 - b) Select **Next**.
 3. In the **Select Reflector** step, select the type of reference reflector used to build the DGS curve:
 - a) Select **Reflector = SDH, FBH, K1-IIW, or K2-DSC**.
(If **SDH** or **FBH** is selected, the hole diameter must be specified.)
-

- b)* Select **Next**.
- 4. In the **Set Curves Level** step, proceed as follows:
 - a)* Select **Reg. Level**, and then enter the registration level; this value usually equals the critical flaw size for the application.
 - b)* Select **Delta Vt** to set the attenuation of the coupling variation resulting from the surface condition of the calibration block and the inputted part.
 - c)* Select **Warning Level**, and then enter the value of the warning curve offset (dB) relative to the main DGS curve.
Up to three warning curves can be added.
 - d)* Select **Next**.
- 5. In the **Set Attenuations** step, proceed as follows:
 - a)* Select **Cal. Block Att.** to specify the attenuation (dB/mm) for the material of the calibration block.
 - b)* Select **Delta Vk**, and then enter the Delta Vk value (dB).
 - c)* Select **Specimen Att.** to specify the attenuation (dB/mm) for the material of the inspected part.
 - d)* Select **Next**.
- 6. In the **Set Gate A on Echo A** step, proceed as follows (see Figure 5-22 on page 144):

NOTE

This step might be missing if the sensitivity calibration was done prior to starting the DGS calibration wizard.

- a)* Move the probe over the reflector to build a signal envelope.
- b)* In the **Start** and **Width** parameters, enter the start and width of gate A, ensuring that the signal envelope from the reflector crosses the gate in the middle, and that the gate is wide enough to contain the complete width of the signal envelope.

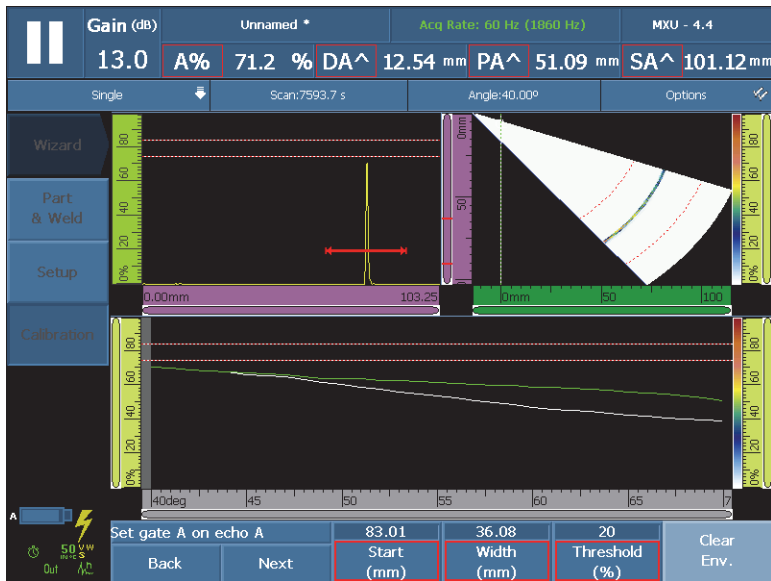


Figure 5-22 Set Gate A over the signal envelope

- c) Set the **Threshold** value between 20 % and 25 %.
 - d) Select **Next**.
7. In the **Calibrate** step, proceed as follows:
- a) Carefully move the probe back and forth along the calibration block and over the reflector to obtain a good-quality signal.
 - b) If needed, select **Clear Env.** to erase the current signal envelope and start building a new one.
 - c) Carefully position the probe to get the maximum signal amplitude in gate A.
 - d) Select **Calibrate**.

As shown in Figure 5-23 on page 145, the DGS main (red) and warning (white) curves appear in the **A-scan** view. Also, the **ERS** (equivalent reflector size for a FBH), **AdBCurve**, **MaxAdBCurve**, and **A%** readings appear at the top of the screen.

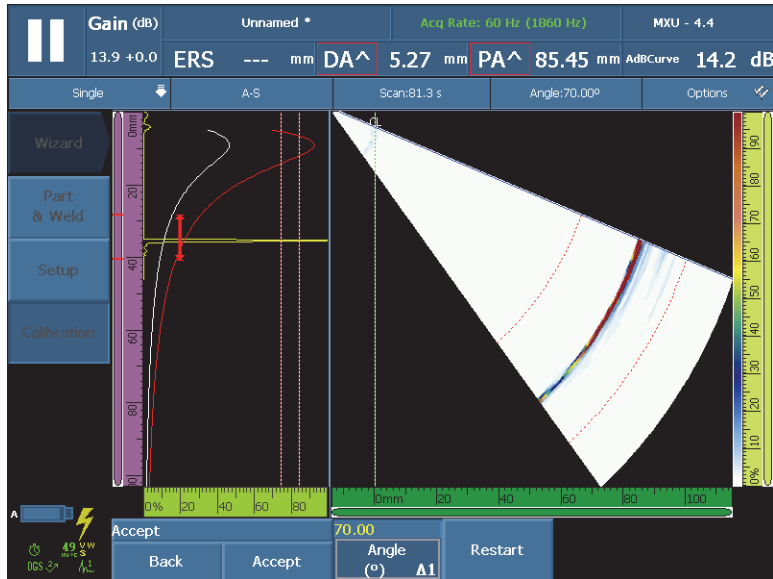


Figure 5-23 Example of DGS sizing curves

8. In the **Accept** step, proceed as follows:
- If needed, select **Angle (°)** to set the desired angle and verify the calculated curve.
 - If the calibration is not satisfactory, select **Restart** to restart the calibration process.
 - Select **Accept** to accept the calibration, and then exit the wizard.
- The DGS label appears in the status indicator (see Figure 5-24 on page 145).



Figure 5-24 Completed DGS calibration status indicator

5.5.4 AWS Calibration

The **AWS** sizing function assists you in performing inspections covered under the American Welding Society D1.1 (or D1.5) Structural Welding Code for steel. This code provides a method for classifying discontinuities found in welds using ultrasonic inspection. This code uses equation (1) on page 146 to develop an indication rating for a reflector found during an inspection. An indication rating is a classification according to grades defined in the AWS code to evaluate the size of defects.

$$A - B - C = D \quad (1)$$

where:

A = Discontinuity Indication Level in dB (see “AWS A” on page 274)

B = Reference Indication Level in dB (see “AWS B” on page 274)

C = Attenuation Factor in dB (see “AWS C” on page 275)

D = Indication Rating in dB (see “AWS D” on page 275)

The **AWS** calibration wizard takes you through the steps to calibrate the instrument for AWS-D1.1/1.5 code-compliant inspections.

Configuring Setups Prior to AWS Calibration

This section describes how to configure the setups before performing an AWS calibration. The first step consists of creating a setup, as outlined below.

The AWS sizing calibration is performed using a side-drilled hole (SDH) reference reflector.

An AWS calibration on a UT group has the following requirements:

- An AWS compatible UT probe and wedge
- A calibration block with a known SDH reflector

This procedure applies to the SDH reflector type: the AWS sizing calibration is performed for 45°, 60°, and 70° angles using a SDH reference reflector.

An AWS calibration on a PA group has the following requirements:

- An AWS compatible PA probe and wedge
- A calibration block with a known SDH reflector

This procedure applies to the SDH reflector type, and to sectorial scans only.

To create a setup

1. Select **Wizard > Part & Weld > Start** to start the **Part & Weld** wizard.
2. Follow the on-screen help to complete the wizard steps.
3. Select **Finish**.
4. Select **Wizard > Setup > Start** to start the **Setup** wizard.
5. Follow the on-screen help to complete the wizard steps.
6. Make sure that the law configuration covers at least one of the following angles: 45°, 60°, or 70°.

After setup creation, the next step consists of calibrating the sensitivity.

To calibrate the sensitivity

1. Select **UT Settings > Advanced**, and then set the **Ref. Amplitude** parameter to the desired value.
2. Select **Wizard > Calibration > Type = Ultrasound**, and then select **Mode = Sensitivity**.
3. In the **Select Calibration** step of the **Calibration** wizard, select **Start**.
4. Follow the on-screen help to complete the wizard steps, but do not change the **Ref. Amplitude** parameter value.
5. When the **Calibration** wizard is complete, select **Sizing > Type > AWS**.

The **AWS A**, **AWS B**, **AWS C**, and **AWS D** readings appear at the top of the screen to provide code-compliant measurements.

AWS appears in the status indicator (see Figure 5-25 on page 147).

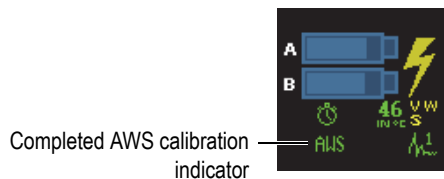


Figure 5-25 Completed AWS calibration status indicator

6. Select the **Standard** to specify which version of the AWS code you want to comply with.
7. Select the **Weld Type** to specify the type of weld to be inspected.

8. Select **Angle** to adjust the real angle entering in the material.
9. Set the **Thickness** of your part.

5.6 Encoder Calibration

This section describes how to calibrate an encoder.

IMPORTANT

The **Wizard > Calibration > Type = Encoder** option only appears when you are using at least one encoder for your scan (defined in **Scan > Inspection**).

NOTE

Before performing this procedure, an X or XY encoder must be properly connected to the alarm and I/O connector.

To calibrate an encoder

1. Select the encoder:
 - a) Select **Wizard > Calibration > Type = Encoder**.
 - b) From the **Encoder** list, select the encoder to be calibrated.
If you are using only one encoder, this parameter is shaded.
 - c) Select **Start**.
2. In the **Set Origin Position** step, proceed as follows:
 - a) Physically move the encoder to a marked point (A).
 - b) Set the **Cur. Position**.
 - c) Select **Next**.
When you select **Next**, the current encoder position is set to the value established in the **Cur. Position** parameter.
3. In the **Set Distance** step, proceed as follows:
 - a) Physically move the encoder to a second marked point (B), and note the distance traveled.

- b) Select **Distance**, and then enter the inspection pass distance (B minus A).
- c) Select **Calibrate**.
The encoder calibration is performed. The resolution is displayed.
- 4. In the **Accept** step, select **Accept** if the resolution is correct. If not, select **Restart** to recalibrate.
- 5. To calibrate the encoder for a second axis, select **2** from the **Encoder** list, and then repeat this procedure.

6. Inspection Procedures

This chapter contains various procedures related to inspections tasks.

6.1 Setting the General Inspection Parameters

The basic inspection parameters are available in the **UT Settings > General** submenu.

To set the general inspection parameters

1. Select **UT Settings > General > Gain**.
2. Enter an appropriate gain value. For an appropriate gain value, you should typically view the peak amplitude in a gate at about 80 % of the full-scale view.
3. Select **Start**, and then enter the start position value of the displayed range.
4. Select **Range**, and then enter the range value to be displayed.
5. Select **Wedge Delay**, and then enter the appropriate delay value for the wedge. For a PA group, use the wedge delay calibration wizard instead to automatically set appropriate wedge delay values for all focal laws.
6. Select **Velocity**, and then enter the appropriate ultrasonic velocity value for the material of the part to be inspected.

6.2 Setting the Tx/Rx Mode (UT Group Only)

The parameters used to configure the emission of ultrasonic pulses are available in the **UT Settings > Pulser** submenu.

To set the pulser and receiver parameters

1. If you are using a UT group on the PA connector, select **UT Settings > Pulser > Pulser**, and then enter the number of the pulser to which the probe is connected.
2. From the **UT Settings > Pulser > Tx/Rx mode** list, choose the desired transmission/reception mode (illustrated in Figure 6-1 on page 152).

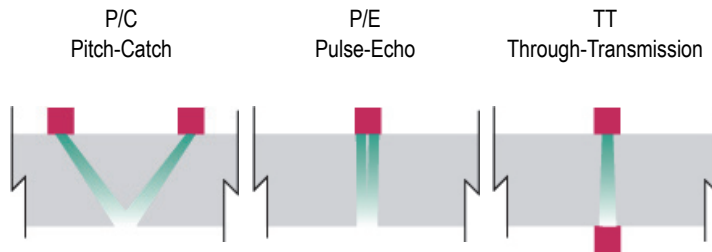


Figure 6-1 Choice of transmission/reception modes

3. Select **UT Settings > Pulser > Freq.**, and then select the probe frequency value.
4. Select **Energy**, and then enter the voltage value.
5. Select **PW**, and then enter the pulse width value. In automatic mode (**Auto**), the pulse width is adjusted according to the probe frequency.
6. In the **Acq Rate** list, select an acquisition-rate calculation method, or enter a custom value.
7. Choose **Optimum** to obtain the best value or **Auto Max** to get the maximum available acquisition rate (according to settings).

6.3 Configuring the Reference Gain

This section describes how to scan using reference gain.

To scan using a reference gain

1. Place the probe on a calibration block containing a known reflector.
2. Adjust a gate on the reflector signal in order to obtain a reading of the signal amplitude in the reading field.
3. Adjust the gain so that the reflector signal reaches the reference level (80 % of full-screen height is the suggested value) as follows:

- ◆ Tap the Gain field in the reading-fields area, and then enter the gain value.
OR
- Tap and hold the Gain field, and then select **Set to Reference (80 %)**.

NOTE

The default reference amplitude value is 80 %. To modify this value, select **UT Settings > Advanced > Ref Amplitude**, and then enter the new reference value.

NOTE

To display the difference between the signal in gate A and the reference signal, select **Measurements > Reading**, and then select the **AdBr** parameter as one of the four reading fields.

6.4 Setting Up a PA-TOFD Inspection (OmniScan MX2 Only)

The OmniScan MX2 allows you to perform phased array (PA) and time-of-flight diffraction (TOFD) scans simultaneously. Setting up a PA-TOFD inspection consists essentially of creating two PA groups, creating one TOFD group, and activating the inspection layout.

To set up a PA-TOFD inspection

1. Configure the part to inspect:
 - a) Select **Wizard > Part & Weld > Start**.
 - b) In the **Part** step, set the parameters to correctly represent the part to inspect, and then select **Next**.
 - c) In the **Select Weld** step, choose the weld type to inspect, and then select **Next**.
 - d) In the **Finish** step, select **Finish**.
2. Select **Wizard > Setup > Start**.
3. In the **Select Operation** step, select **Operation = Modify**, and then select **Group Mode = PA**.
4. Select **Next**.

5. Configure the first PA group by following the **Setup** wizard's step-by-step instructions, and select **Finish** after you have completed the first PA group configuration.
 6. Configure the second PA group:
 - a) Select **Wizard > Setup > Start**.
 - b) In the **Select Operation** step, select **Operation = Add**, and then select **Group Mode = PA**.
 - c) Choose **Source = Copy From** to copy the parameters of a previously created group or **Source = Default** to copy the connector default settings.
 - d) Follow the **Setup** wizard's step-by-step instructions, and select **Finish** after you have completed the second PA group configuration.
-

NOTE

When using OMNI-M-PA1664 and OMNI-M-PA16128 modules, you cannot use two connector types at the same time. However, you can add a PA group and a UT group at the same time, but they must be added on the PA connector using an adaptor. After the connector of the first group is selected, all the other groups must be added on the same connector.

The OMNI-M2 module type does not have this connector-type limitation.

7. Configure the UT group:
 - a) Select **Wizard > Setup > Start**.
 - b) In the **Select Operation** step, select **Operation = Add**.
 - c) Select **Group Mode = UT Conventional Ultrasound** if you connected your UT probes using a PA connector adaptor.
OR
Select **UT Conventional Ultrasound** if you connected your UT probes using a UT connector.
 - d) Select **Application = TOFD Parallel/Non-Parallel**.
 - e) Select **Next**.
 - f) In the **Inspection Mode** step, proceed as follows:
 - (1) Enter appropriate element numbers in the **Pulser** and **Receiver** parameters, if necessary.
 - (2) Select **Next**.

- g) Follow the other **Setup** wizard instructions step-by-step to complete the group configuration for the TOFD group.
8. Choose the appropriate layout in the Layout section of the title bar (see Figure 6-2 on page 155).

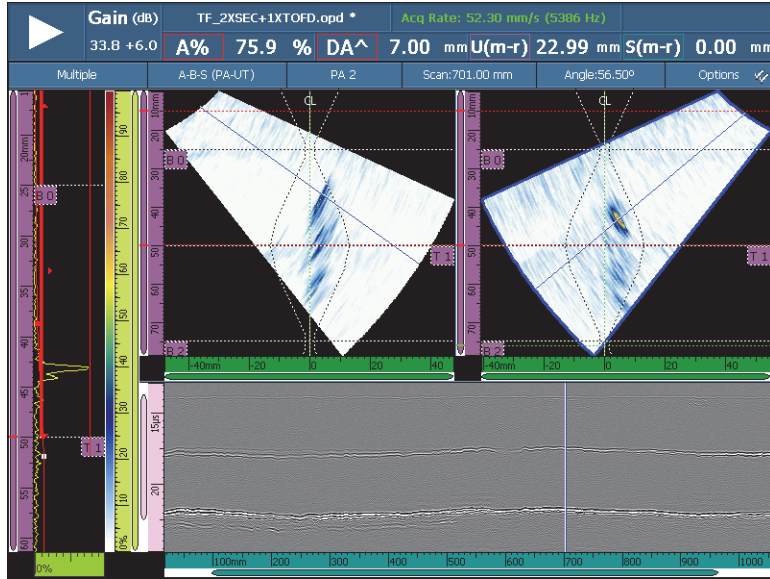


Figure 6-2 An example of the PA-UT Weld (A-B-S) layout

6.5 Configuring a Thickness C-Scan

This section describes how to configure a thickness C-scan. The thickness C-scan is used to monitor part thickness (for example, during corrosion inspections).

To configure a thickness C-scan

1. Place the probe on a calibration block.
2. Select **UT Settings > General > Wedge Delay**, and then enter the value of the wedge delay.

If the value is unknown, see “Wedge Delay Calibration for a PA Group” on page 123.

3. Select **Velocity**, and then enter the velocity of ultrasound in the part to be inspected.
If the value is unknown, see “Ultrasonic Velocity Calibration” on page 114.
4. In the **Part** step of the **Part & Weld** wizard, select **Thickness**, and then enter the thickness of the part to be inspected.
5. In the **Material** list, choose the type of material to be inspected.
6. Select **UT Settings > General > Start**, and then enter the time-base start value.
7. Select **Range**, and then enter the length of the time-base range according to the part thickness.
8. Set gate A on the back wall signal:
 - a) Touch the start point of gate A.
 - b) Use the Scroll knob to adjust the start value, or use the virtual keyboard to directly enter the new start value.
 - c) Touch the end point of gate A, and then use the Scroll knob to include the complete length of the signal according to the gate threshold selected.
9. Set gate I on the interface signal:
 - a) Touch the start point of gate I.
 - b) Use the Scroll knob to adjust the start value, or use the virtual keyboard to directly enter the new start value.
 - c) Touch the end point of gate I, and then use the Scroll knob to include the complete length of the signal according to the gate threshold selected.
 - d) Select **Gates/Alarms > Gates > Gate = I**, and then select **Parameters = Mode**.
 - e) Select **A-Scan Synchro = I/**.

NOTE

The I/ parameter is only available if the following settings have been made:

- In the **Setup** wizard, the **Law Config** parameter is set to **Linear** and the **Angle (deg)** parameter is set to 0°.
- OR
- In the **Setup** wizard, the **Law Config** parameter is set to **Linear at 0°**.

-
10. Select **Measurements > Reading > Measurement Mode = Corrosion 1**.

NOTE

T, the thickness reading, is a dynamic reading. It can appear as **T (A[^])**, **T (B/)**, or **T(A[^]-B/)**, and so on, according to the selections made under **Gates/Alarms > Thickness > Source**.

11. Select the **A-C** layout in the Layout section of the title bar.
12. To choose C-scan sources, tap and hold the C-scan until a contextual menu appears, where you can choose from several available sources.
13. Select **Thickness** to display a thickness C-scan.
The thickness C-scan has been configured.

6.6 Using the Export C-Scan Function

You can use the **Export C-scan** function to export data in a simple text file format that can be read using Microsoft Excel. You can export either the minimum thickness or the maximum amplitude, depending on your C-scan source selection in the OmniScan.

To use the Export C-scan function

1. Tap and hold the C-scan to open the shortcut menu.
Export appears at the bottom of the shortcut menu (see Figure 6-3 on page 158).

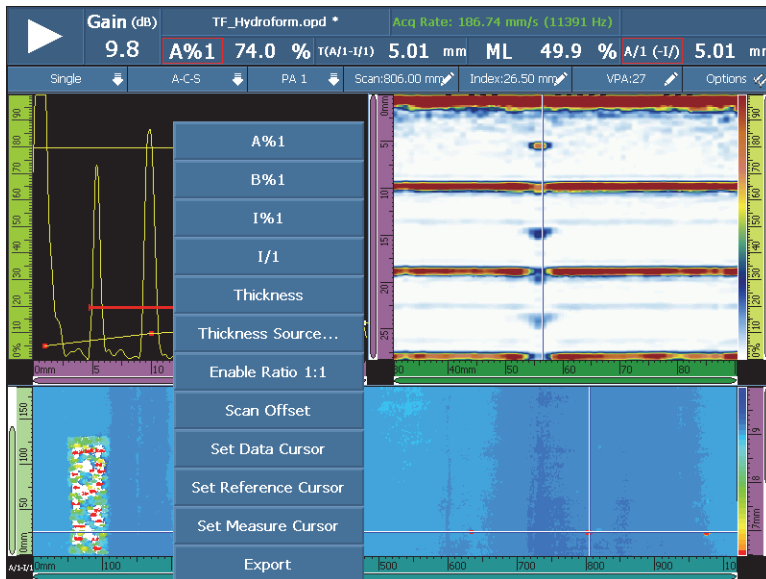


Figure 6-3 The Export C-scan function

2. Choose **Export** to export the thickness or amplitude data from the current C-scan. By default, the exported data files are saved in the **Export** folder, and the data is organized as follows:
 - The export file parameters are detailed in the top left portion of data.
 - The next block of data consists of minimum thickness values (or maximum amplitude values) for each point on the current C-scan. The columns represent scan axis positions and the rows represent index axis positions.

6.7 Using the Profile B-Scan View

The **Profile** B-scan view represents the side profile of the test material, based on thickness or time-of-flight (TOF) measurement. This view, typically used in corrosion scanning applications, is used to verify acquired thickness measurements and to provide a visual reference showing areas on the part with critical thickness values. Additionally, an encoder can be used to provide location information over a scanned distance.

Profile layouts are only available in conventional UT mode for 0° inspections. You activate them through the **Setup** wizard.

To use the Profile B-scan view

1. Open the **Setup** wizard.
2. Select **Conventional UT Group Mode > Zero Degrees Application**.
3. Complete the remaining steps of the wizard, and then exit the wizard.

The **A-B Profile** layout is now available in the Layouts section of the title bar (see Figure 6-4 on page 159). The **A-B-C-D Profile** layout is also available, but only in analysis mode.

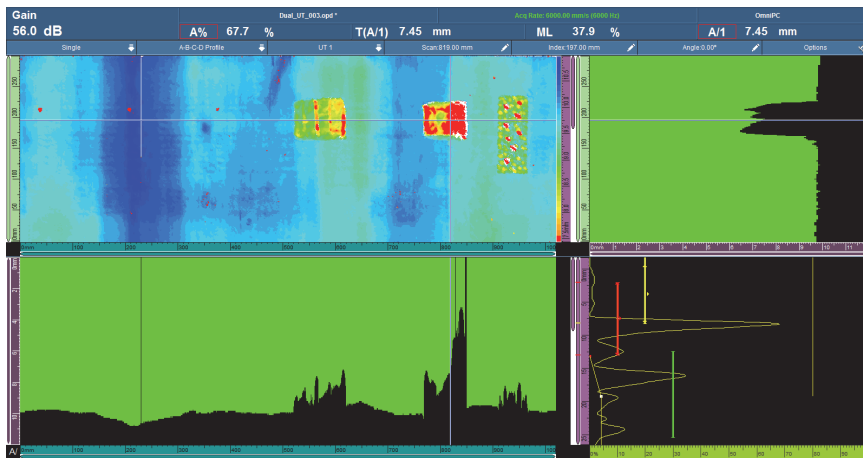


Figure 6-4 The Profile B-scan view

6.8 Using the Export B-Scan Function (OmniPC Only)

You can use the **Export B-scan** function in OmniPC to export data in a simple text file format that can be read using Microsoft Excel.

To use the Export B-scan function

1. In OmniPC, right-click on the B-scan to open the shortcut menu. Depending on the type of scan, **Export** or both **Export** and **Export All** appear as available options (see Figure 6-5 on page 160).

2. Click **Export** to export the A-scan data from the current index position.

OR

Click **Export All** to export the A-scan data from all index positions.

By default, the exported data files are saved in the **Export** folder, and the file contents are organized as follows:

- 1st line: A-scan amplitude data at **Scan Start, Index Start** position
- 2nd line: A-scan amplitude data at **Scan Start + 1, Index Start** position
- 3rd line: A-scan amplitude data at **Scan Start + 2, Index Start** position

IndexQTY + 1st line

A-scan amplitude data at **Scan Start, Index Start + 1** position

IndexQTY + 2nd line

A-scan amplitude data at **Scan Start + 1, Index Start + 1** position

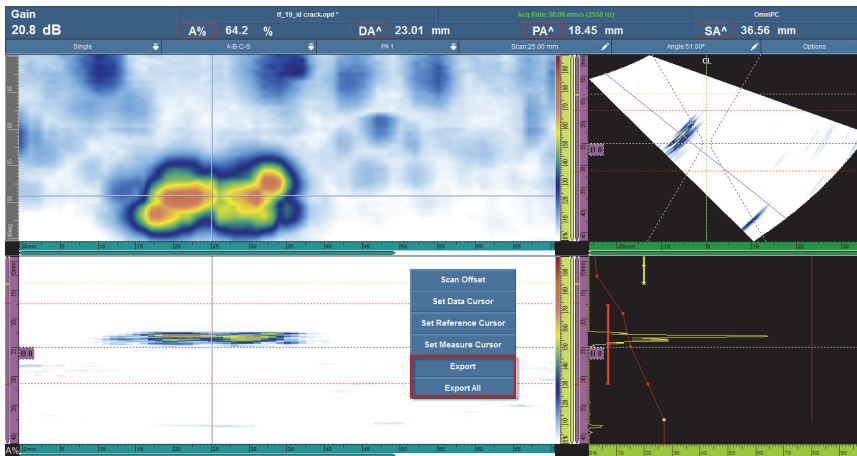


Figure 6-5 The Export function

6.9 Setting Alarms and Outputs

This section explains how to set alarms and outputs.

6.9.1 Setting an Alarm

The alarms on the OmniScan instrument can be output in three different ways:

- The three indicators on the front panel of the instrument
- The buzzer (sound)
- The three TTL outputs of the alarm and I/O connector

To trigger an alarm, at least one gate must be set, because all the alarm conditions are linked to the gates.

To set an alarm

1. In the **Gates/Alarms > Alarm > Alarm** list, choose the alarm number (**Alarm 1** to **Alarm 3**) that you want to configure.
2. In the first **Gates/Alarms > Alarm > Condition** list, choose the condition triggering the alarm. For example, choosing **Gate A** triggers the alarm whenever a signal crosses gate A.
For a more complex alarm, use a second condition. In such case, continue to the next step. If one condition is sufficient, the procedure stops here.
3. In the second **Gates/Alarms > Alarm > Condition** list, choose the parameter triggering the alarm.
4. In the **Gates/Alarms > Alarm > Operator** list, choose one of the following options to link two conditions:
OR: to activate an alarm when either condition is met.
AND: to activate an alarm when both conditions are met.
5. Select **Gates/Alarms > Alarm > State** to activate the alarm.

NOTE

Before activating the alarm, make sure that the output settings in the **Output** submenu are set correctly. After the alarm is turned on, these settings cannot be changed.

6.9.2 Setting an Alarm Output

This section explains how to set an alarm output.

To set an alarm output

1. If you want the signal of the previously configured alarm to be triggered on an output, select **Gates/Alarms > Output**.
Because the output number is related to the alarm number, the **Alarm** parameter is shaded, and displayed for illustration purposes only.
2. To modify the output parameters of the alarm, make sure that you select **Gates/Alarms > Alarm > State = Off**.
3. If you need a condition count, select **Gates/Alarms > Output > Count**, and then enter the number of times the alarm condition must be met before triggering the alarm.
4. If you need an audible alarm, select **Sound = On**.
5. If you need a delay between the occurrence of the alarm condition and the actual alarm trigger, select **Delay**, and then enter the delay value (in milliseconds).
6. If you need a specific duration for the alarm, select **Hold Time**, and then enter the length of time after which the alarm turns off.

6.9.3 Setting an Analog Output (OmniScan MX2 Only)

This section explains how to configure an analog output.

To set an analog output

1. In the **Gates/Alarms > Analog > Analog Output** list, choose **Analog 1** or **Analog 2**.
2. In the **Group** list, select the group from which the signals originate.
3. In the **Data** list, select the information that you want to link to the analog output.
4. Select **State = On** to activate the analog output.

6.10 Setting Sizing Functions

This section describes how to set the various sizing functions.

6.10.1 Manually Configuring the DAC Curve

This section describes how to manually configure the DAC curve. See “DAC Calibration” on page 135 for the procedure on using the calibration wizard.

To manually configure a DAC curve

1. Select **Sizing > Type = DAC**.
2. Select **Type Setup**, and then set the parameters.
For more details, see “Parameters for DAC, Linear DAC, and TCG Curves” on page 290.
3. Position the probe to obtain the maximum signal amplitude for the reference reflector corresponding to the first point to be added on the DAC curve (see Figure 6-6 on page 163).

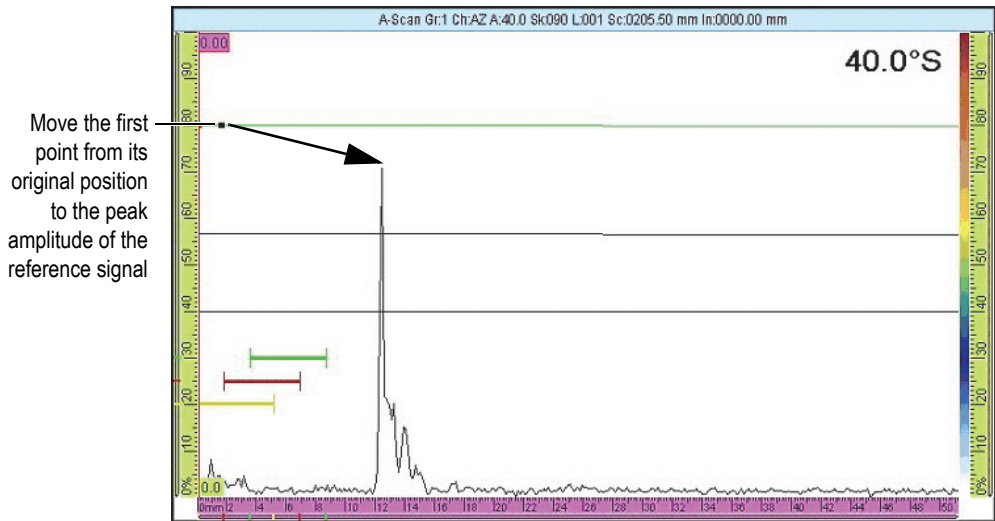


Figure 6-6 The first point of the DAC curve

4. If the signal amplitude is higher or lower than the reference amplitude, select **Sizing > Operator Mode > Reference Gain**, and then adjust the gain value so that it reaches the calibration level.
5. Select **Curves Setup > Apply To > All Laws** or **Current Law** to add the points on all laws or on the current law only. After the points have been added to the current law, you can still select **All Laws** to apply the points of the current law on all laws.
6. Select **Position**, and then move the first point from its original position to the peak amplitude of the reference signal.
7. Move gate A over the signal from the reference reflector.

8. Select **Add**.
9. To modify a particular point, select **Point**, and then choose the point to be modified in the list.

This point of the DAC curve appears as a white dot in the data view. Next, proceed as follows:

- a) Set the value of the **Position** parameter button in order to move the white dot to the time-of-flight value of the reflector.
- b) Set the value of the **Amplitude** parameter button in order to move the DAC curve on the reflector signal peak amplitude (see Figure 6-7 on page 164).

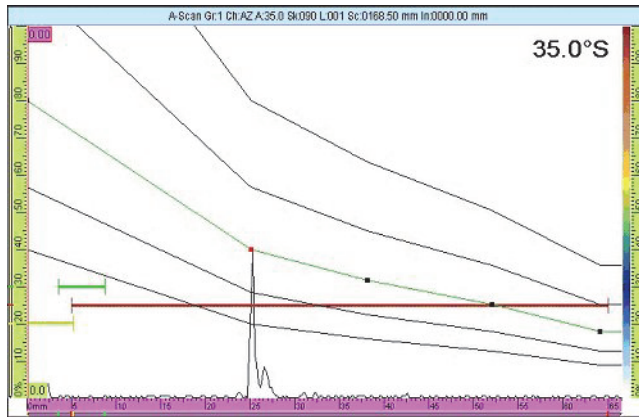


Figure 6-7 The DAC curve on the next signal

10. Move gate A over the signal from the second reference reflector.

To delete a point on the curve

1. In the **Sizing > Curves Setup > Point** list, choose the point you want to delete.
2. Select **Delete**.

To verify the DAC curve

1. Select the A-scan layout in the Layout section of the title bar.
2. Tap and hold the A-scan, and then select **Enable Envelope**.
3. Scan over each reflector, and verify that all reflectors are on the DAC curve (see Figure 6-8 on page 165).

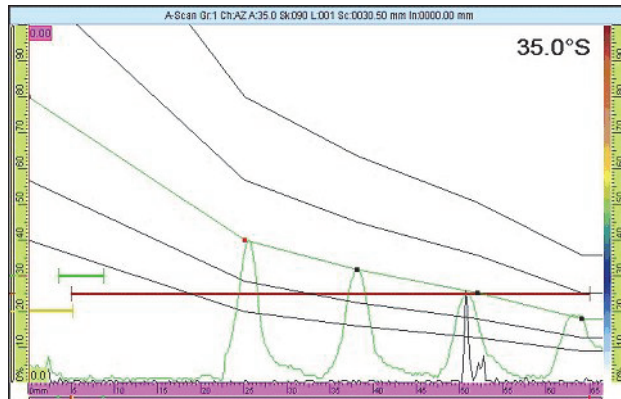


Figure 6-8 Verification of the DAC curve

TIP

To reset the envelope, tap and hold the A-scan, and then tap **Clear Envelope** on the shortcut menu.

6.10.2 Automatically Configuring the Linear DAC Curve

The OmniScan instrument allows you to calculate the DAC curve by specifying the material attenuation slope.

To automatically configure the Linear DAC curve

1. Select the A-scan layout in the Layout section of the title bar.
2. Select **Sizing > Type = Linear DAC**.
3. Select **Sizing > Curves Setup > Mat. Attenuation**, and then enter the attenuation slope (dB/ μ s or dB/mm) for the material of the inspected part.
The OmniScan instrument automatically draws the DAC curves in accordance with this physical property.
4. If needed, select **Sizing > Curves Setup > Delay**, and then enter the delay value to postpone the start of the linear attenuation curve.

6.10.3 Manually Configuring the TCG Curve

This section describes how to manually configure the TCG (time-corrected gain) curve.

To manually configure the TCG curve

1. Select the A-scan layout in the Layout section of the title bar.
2. Select **Sizing > Type = TCG**, and then set the **Type Setup** parameters (see “Parameters for DAC, Linear DAC, and TCG Curves” on page 290 for more information).
3. Select **Sizing > Curves Setup > Add**.
4. In the **Point** parameter list, select **2**.

This point of the TCG curve appears as a white dot in the lower-left corner of the data screen. Next, proceed as follows:

- a) If the white dot is not visible, increase the **Gain** value until the dot is visible.
- b) Set the value of the **Position** button to bring the white dot to the time-of-flight of the reflector.
- c) Under **Gain**, adjust the gain value up or down in order to set the signal at the desired level (80 % is the suggested value).
- d) If you want to see the exact amplitude of the reflector, select **Gates/Alarms > Gates**, and then set the parameter values so that gate A is placed over the reflector.

The amplitude can be displayed in the **A%** reading field at the top of the screen.

5. Move to the next reflector, or look at the next back wall signal.
6. Select **Sizing > Curves Setup > Add** to add a new point to the TCG curve.
7. In the **Point** list, choose the new point (in this case, **3**), and then repeat steps 4 through 6 until you complete the curve. The TCG curve can have up to 32 points.

To delete a point on the curve

1. In the **Sizing > Curves Setup > Point** list, choose the point you want to delete.
2. Select **Delete**.

To verify the TCG curve

1. Select the A-scan layout in the Layout section of the title bar.

2. Tap and hold the A-scan view, and then select **Enable Envelope**.
3. Move the probe to scan over the calibrated reflector, and verify that all the peaks reach 80 % (see Figure 6-9 on page 167).

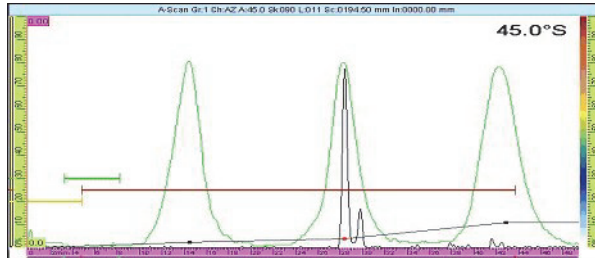


Figure 6-9 An envelope with peaks at 80 %

TIP

To reset the envelope, tap and hold the A-scan view, and then tap **Clear Envelope** on the shortcut menu.

NOTE

You can convert a DAC curve into a TCG curve, and vice-versa, by selecting **Sizing > Type > DAC** or **TCG**.

6.11 Setting Measurements

The procedures in this section describe how to configure reading fields and the indication table.

6.11.1 Configuring Reading Fields

You can select which ultrasonic parameters appear in the four reading fields displayed at the top of the OmniScan instrument user interface (see Figure 2-11 on page 50 for reading-fields explanations).

To configure the reading fields

1. Tap the reading-fields area to toggle between the two available reading-field lists, and choose the one you want to display on the screen (or edit using the other parameters).

Only one list can be displayed on the screen at a time, but readings from both lists appear when you make an entry in a report.

2. Tap and hold the reading-fields area, and then select **Select Reading List**.

In the dialog box that appears, you can choose a reading-field group (see Figure 6-10 on page 168). You can easily choose a group of eight parameters that are appropriate to your application.

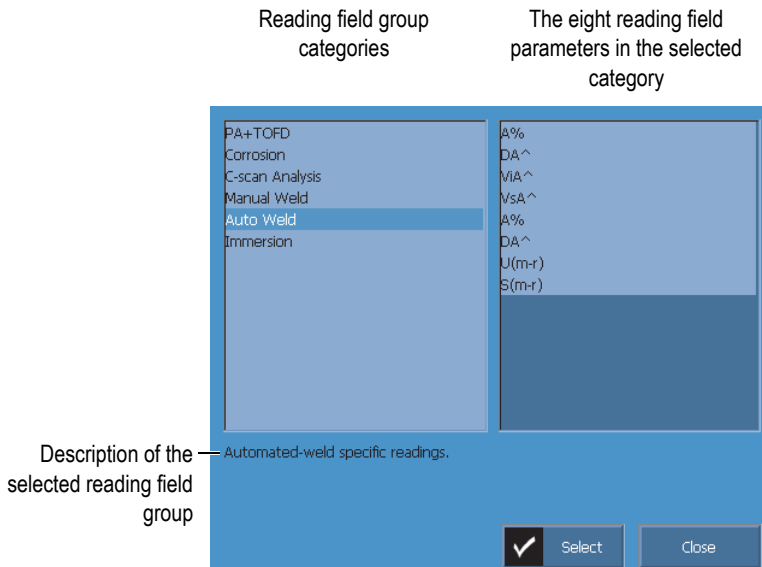


Figure 6-10 Dialog box for choosing predefined readings

3. In the reading list selection dialog box, choose the category that best represents your application from the list on the left, and then choose **Select**.

4. To change a specific reading field, tap and hold the reading zone, and then choose **Select Reading**.
The reading field selection dialog box appears.
5. In the reading field selection dialog box:
 - a) Choose the reading field category from the list on the left.
 - b) Choose the reading field from the list on the right, and then choose **Select**.

6.11.2 Configuring and Building an Indication Table

This section describes how to configure and build an indication table. It also provides information on how to delete and modify entries, and how to create a report that includes an indication table. For more information on indication tables, see “Indication Table Submenu” on page 279.

To configure an indication table

1. In the title bar menu, select **Options > Indication Table** to display an indication table at the bottom of the data view (see Figure 6-11 on page 169).

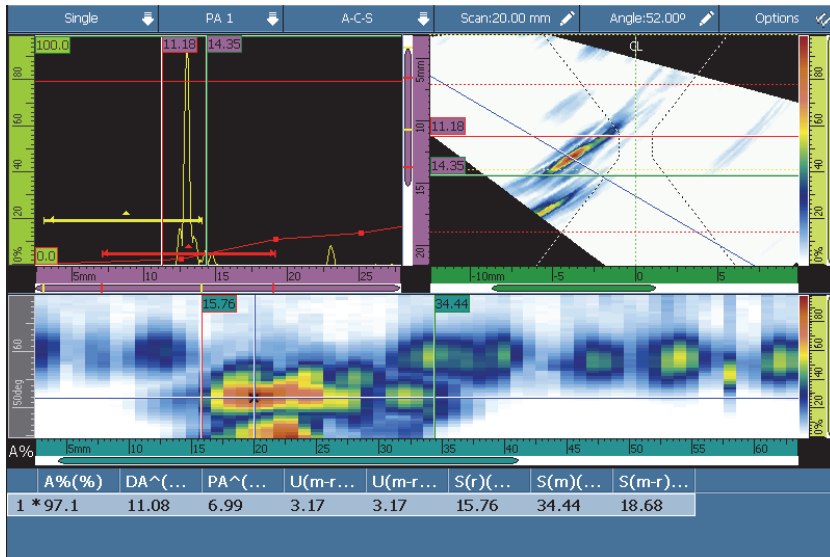


Figure 6-11 Example of an indication table

2. Select **File > Report > Category = Format**, and then select **Component > Images** to include indication images in the report.
3. Choose the desired layout in the Layout section of the title bar (see “Scans, Views, and Layouts” on page 57 for descriptions of available layouts).
4. Tap and hold the reading zone, choose **Select Reading**, and then choose the reading fields that you want to associate with the indication information added to the indication table.

To add an indication in the indication table

- ◆ Tap and hold the reading zone and select **Add Indication** for every indication you want to add to the indication table.

It is possible to modify the information in the entries.

To modify or delete an indication from the indication table

1. Select **Measurements > Indication Table > Mode = Indications**, choose **Select**, and then enter the desired indication number (see Figure 6-12 on page 170).

Entry	Scan mm	Index mm	Group	Channel	A% %	DA* mm	PA* mm	SA* mm
1	0.00	0.00	1	45.0°	54.8	11.79	-33.30	11.79
2	0.00	0.00	1	58.0°	200.0	17.15	-15.20	17.15
3	0.00	0.00	1	60.0°	197.3	16.06	-14.47	16.06
4 *	0.00	0.00	1	62.0°	43.4	8.07	-26.75	8.07
5	0.00	0.00	1	48.0°	58.3	8.22	-35.39	8.22
6 *	0.00	0.00	1	60.0°	75.2	8.94	-26.80	8.94

On		4			
Show Table	Add	Select Δ1	Delete	Comments	...

Figure 6-12 Selecting an entry in the indication table

2. To add a comment for the entry, using either a USB keyboard connected to the OmniScan instrument or the virtual keyboard, select **Comments**, and then type your comment in the **Entry Comment** dialog box.

It is possible to modify the comment by selecting **Measurements > Indication Table > Mode = Indications**, and then selecting **Comments**.

3. To delete an entry, select it as per step 1, and then select **Delete**.

OR

Tap and hold the reading zone, and then select **Delete Last Indication**.

NOTE

An asterisk (*) to the right of an entry number indicates that a comment is attached to the entry, or that an illustration is attached to an indication. Note that the asterisk does not appear immediately after adding a comment. The asterisk only appears after the indication table list has been refreshed, such as when another entry is added to the table (see Figure 6-12 on page 170).

To include the indication table in a report

1. Select **File > Report > Category = Format**, and then select **Include = Table** to include the indication table in the report.
-

NOTE

There must be at least one entry in order for the indication table to be included in the report.

2. Tap and hold the file name zone in the reading-fields area, and then select **Report Preview**.
3. Choose **Save and Close** to save the report and close the preview screen, or choose **Close** to close the preview screen without saving the report.

6.12 Configuring the Screen

The procedures in this section explain how to configure various elements displayed on the screen.

6.12.1 Changing the Current Layout and the Current View

Depending on your inspection requirements, you may need to change your current layout and view.

To change the layout and the current view

1. Select the desired layout in the Layout section of the title bar.
-

For more details on layouts, see “Layout Selection” on page 36 and “Scans, Views, and Layouts” on page 57.

- To access additional A-scan configuration options, select **Display > A-Scan**.

For more details about these options, see “A-Scan Settings Submenu” on page 280.

6.12.2 Displaying the Cursors

You can toggle the visibility of the cursors and the associated parameter values (see Figure 6-13 on page 172).

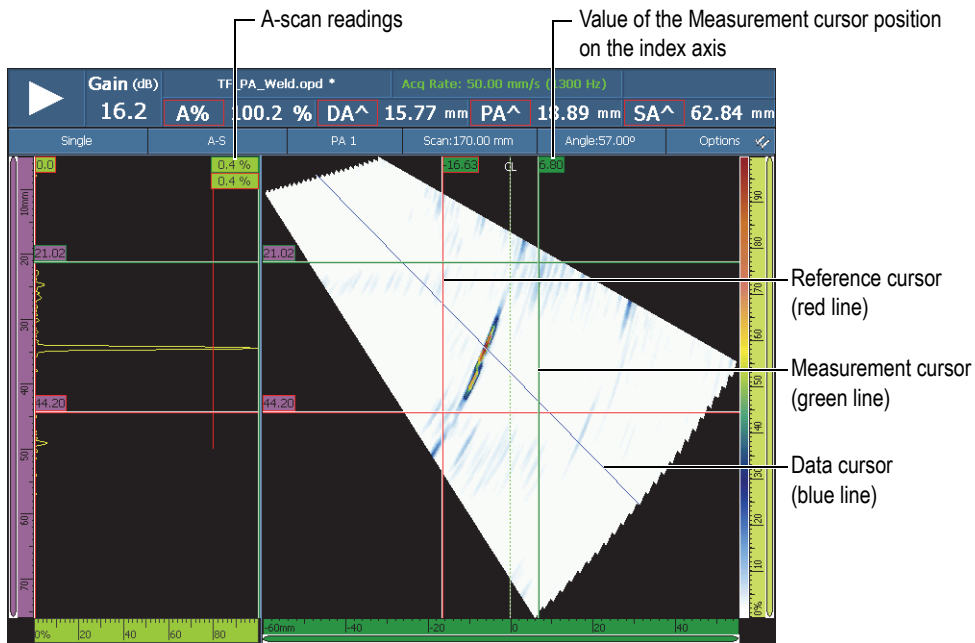


Figure 6-13 Example of cursors

NOTE

In the cursor related data boxes (ex.: **32.98**), the color of the box contour matches the color of the cursor. Similarly, the color of the box background matches the color of the corresponding axis.

To display the cursors

1. Tap the **Options** section of the title bar, and then tap the **Cursor** check box to rapidly activate or deactivate the use of cursors.
The Data cursor (blue line) is always visible.
2. Alternatively, select **Display > Overlay > Cursor**, and then select:
 - a) **Values** to display the position of the cursor on the axis.
 - b) **A-Scan Readings** to display the value of the A-scan signal at the cursor position.

NOTE

The active cursor blinks. To move the active cursor to a particular location, double-click or tap twice on the new location. To change the active cursor, click once on the cursor to be activated.

6.12.3 Displaying the Gates

When the Gate mode is enabled, you can change the gate parameters using the touch screen, the keys (shortcuts), the menus, or a mouse. The visibility of each gate can be toggled separately.

To display the gates

1. Tap the **Options** section in the title bar, and then tap the **Gates** check box to rapidly activate or deactivate the gates display. At least one gate should be activated to enable the display.
2. Make sure that the required inspection gates are activated in **Gates/Alarms > Gates**, then select **A**, **B**, or **I**, and activate them by selecting their gate geometry.

NOTE

The **Options** title bar menu enables or disables the display of the gates, but the gates can still be used for your setup. However, if a gate geometry is set to **Off**, the gates are disabled and cannot be used for your setup.

6.12.4 Displaying Various A-Scan Related Overlays

Overlays are elements displayed on the data views on top of the basic data. You can toggle the element's visibility. A number of overlay elements are related to the A-scan curves.

To display various A-scan-related overlays

- ◆ Select **Display > Overlay > A-Scan**, and then choose the elements you want to make visible.

See "A-Scan" on page 284 for details on the available elements.

6.12.5 Displaying the Weld Geometry Overlay

If your inspected part includes a weld, you can toggle the visibility of the weld geometry overlay. The weld geometry overlay is a drawing of the weld geometry superimposed on the S-scan view in order to help visualize where indications are located relative to the weld. The **Part & Weld** wizard allows you to define the weld geometry, and the position of the probe relative to the weld.

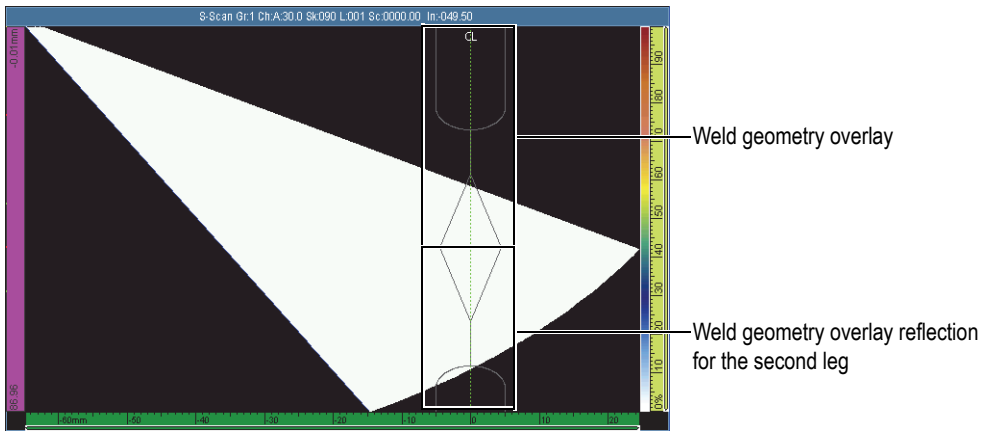


Figure 6-14 Example of a J-type weld geometry overlay

To display the weld geometry overlay

- ◆ Tap the **Options** section in the title bar, and then select **Weld**.

6.12.6 Displaying the Leg Overlay

A leg is the path the shear wave travels in a straight line before being reflected by the opposite surface of the inspected part. The leg overlay is a line drawn on top of a data view to indicate the position of reflection surfaces. The visibility of the leg overlay can be toggled.

To display the leg overlay

- ◆ Tap the **Options** section in the title bar, and then select **Legs**.

6.12.7 Configuring the Grid

To configure the grid

1. Tap the **Options** section in the title bar, and then select **Grid**.
The grid appears on the A-scan view (see Figure 6-15 on page 176).

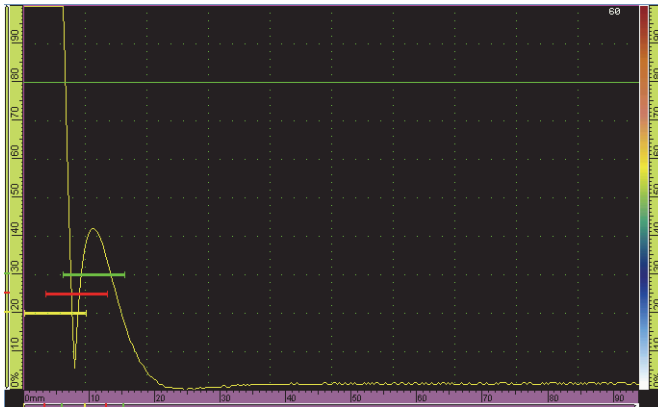


Figure 6-15 A green grid on the A-scan view

2. To set the color of the grid, proceed as follows:
 - a) Select **Display > A-Scan Settings**.
 - b) Select **Grid**, and then choose the desired color for the grid lines.
3. To view data outside of the normal A-scan view, proceed as follows:
 - a) Select **Display > A-Scan Settings**.
 - b) Select **Extra Range**, and then choose the desired setting. See “A-Scan Settings Submenu” on page 280 for details.
4. To change the number of grid cells for the ultrasound and amplitude axes, use the **Ultrasound Cell Qty.** and **Amplitude Cell. Qty.** parameters.

6.12.8 Configuring the View Parameters

To configure the C-scan view parameters

1. Select a layout with a C-scan from the Layouts section of the title bar.
2. To choose the C-scan source, tap and hold the C-scan and then, in the list that appears, choose the desired source (**A%**, **B%**, **I/**, or **Thickness**).
3. Tap and hold the C-scan display, and then select **Enable/Disable Ratio 1:1**.

NOTE

In order for the **Ratio 1:1** parameter button to be available, the inspection must be synchronized on two axes. To perform the synchronization, select **Scan > Inspection > Type = Raster Scan**.

6.12.9 Changing the Color Palettes

The color palettes can be changed for amplitude (for example C-scan or A-scan), RF, or depth views.

To change the amplitude color palette

1. Tap and hold an amplitude color palette (see Figure 6-16 on page 177), and then select **Load**.

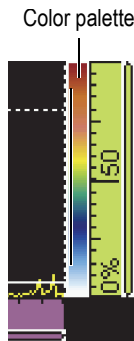


Figure 6-16 The palette on the right side of the A-scan view

2. Choose the color palette in the Color Palette Manager:
 - a) Use the **File Type** button to choose from either the Olympus sample (**System**) or the **User** list.
 - b) Use the Scroll knob to select the color palette file.
3. Select **Open**.

4. Tap the bottom of the color palette, and then enter the signal amplitude start point in the pop-up box (as a percentage of the screen height) that is to be the beginning of the color palette.
5. Tap the top of the color palette, and then enter the signal amplitude end point in the pop-up box (as a percentage of the screen height) that is to be the end of the color palette (red).

You can see the color palette scale on the right side of the A-scan view (see Figure 6-16 on page 177).

To change the RF mode palette or TOFD palette

The RF mode palette is used when the rectification is disabled. To turn rectification off, select **UT Settings > Receiver > Rectifier = RF**.

1. Tap and hold an **RF** color palette, and then select **Load**.
2. Choose the color palette you wish to use.
3. To modify the **Contrast**, tap the bottom of the color palette, and then set the appropriate palette contrast value.
 - When you increase the contrast value, the palette colors are distributed on a larger amplitude scale, which increases the color difference between percentage points.
 - When you decrease the contrast value, the palette colors are distributed on a smaller amplitude scale, which decreases the color difference between percentage points.
4. To modify the **Brightness**, tap the top of the color palette, and then set the appropriate brightness value.
 - If you increase the brightness value, the color palette zoom moves toward positive amplitude values. Any values that are below the start value of the ruler then turn black.
 - If you decrease the brightness value, the values that are above the end value of the ruler turn white.

To change the depth color palette

1. Tap and hold a depth color palette, and then select **Load**.
The Depth palette is used for position C-scans (**I/** and **Thickness**).
2. Tap the bottom of the color palette, and then enter the minimum color palette value in the pop-up box.

3. Tap the top of the color palette, and then enter the maximum color palette value in the pop-up box.

6.13 Performing an Inspection with an Encoder

This section explains how to set up a scan using encoders.

NOTE

Before performing this procedure, an X or XY encoder must be properly connected to the alarm and I/O connector.

To set up a scan using an encoder

1. In the **Scan > Inspection > Type** list, choose the type of scan to be used to scan the part.
2. In the **Scan** list, choose the encoder that is to be used for the scan axis. The other encoder is automatically linked to the index axis.
3. Select **Scan > Encoder**, and then proceed as follows:
 - a) In the **Encoder** list, select **Encoder = 1**.
 - b) In the **Type** list, choose the desired encoder type. **Quad** (quadrature) is set by default. See “Encoder Submenu” on page 320 for a description of the available choices.
 - c) Move the probe and encoder assembly in the scanning direction, and look at the indicator in the title bar to see if the number in the **Scan** or **Index** reading field is positive or negative (see Figure 6-17 on page 179).

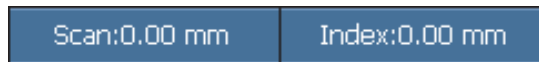


Figure 6-17 Scan and Index reading fields

- d) If the number is negative, select **Polarity = Inverse**.
- e) Select **Resolution**, and then enter the encoder resolution.
- f) If the resolution is unknown, calibrate the encoders for one or two axes.

- g) Select **Origin**, and then enter the value that will be set when the **Preset** parameter is used. The default value is set to **0**.
- h) If only one axis is used, go to step 4.
- i) If a two-axis encoder is connected, then select **Scan > Encoder > Encoder = 2** and repeat steps 3.b to 3.g.

NOTE

The contextual menus (accessed using the **Scan** and **Index** title bar buttons) offer the following options:

- **Inverse Polarity** (see step d above)
- **Define Origin** (see step g above)
- **Set to Origin**

These options act on their respective axes, and can only be used in acquisition mode.

- 4. Select **Scan > Area** and proceed as follows:
 - a) Select **Scan Start**, and then enter the starting position along the scan axis.
 - b) Select **Scan End**, and then enter the ending position along the scan axis.
 - c) Select **Scan Res.**, and then enter the resolution of the scan along the scan axis.
 - d) If you have selected **One-Line Scan** in the **Scan > Inspection > Type**, the procedure is now complete.
 - e) If you have selected two axes in **Scan > Inspection > Type**, continue to the next step.
 - f) Select **Scan > Area > Index Start**, and then enter the starting position along the index axis.
 - g) Select **Index End**, and then enter the ending position along the index axis.
 - h) Select **Index Res.**, and then enter the resolution of the scan along the index axis (Sectorial mode). If in Linear at 0° mode, the **Index Res.** parameter is shaded and the value corresponds to the aperture of the focal laws.
- 5. When you are ready to scan, press the Play key.

6.14 Indexing a Scan

If your scanner encoder setting is defined as **Clicker** or **Clicker + preset**, you can use the **Index Step** function to simulate the physical “clicker” button action that confirms the index distance of your scanner from one inspection scan line to the next (see Figure 6-18 on page 181).

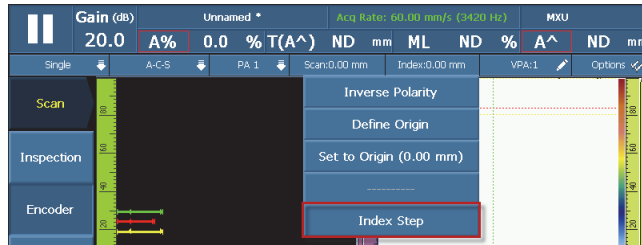


Figure 6-18 The Index Step function

To index a scan

- ◆ After an inspection along a scan line has been completed and the scanner has been indexed (moved) to the next inspection scan position, tap and hold the Index section of the title bar, and then tap **Index Step**.

The scan or index distance value is the same as the distance value determined by the encoder resolution. You can set this value in the scan configuration screen, or alternatively, via the **Scan > Encoder** menu.

7. Complementary Procedures

This chapter contains complementary procedures for various tasks that you may occasionally need to perform.

7.1 Using the File Manager

This section describes the File Manager utility. The File Manager interface is designed to provide as many management options as possible, even if no keyboard or mouse is connected to the instrument.

To open the file manager

- ◆ Select **Preferences > Tools > File Manager** to open the File Manager.

7.1.1 File Manager Interface

There are four areas in the File Manager (see Figure 7-1 on page 184).

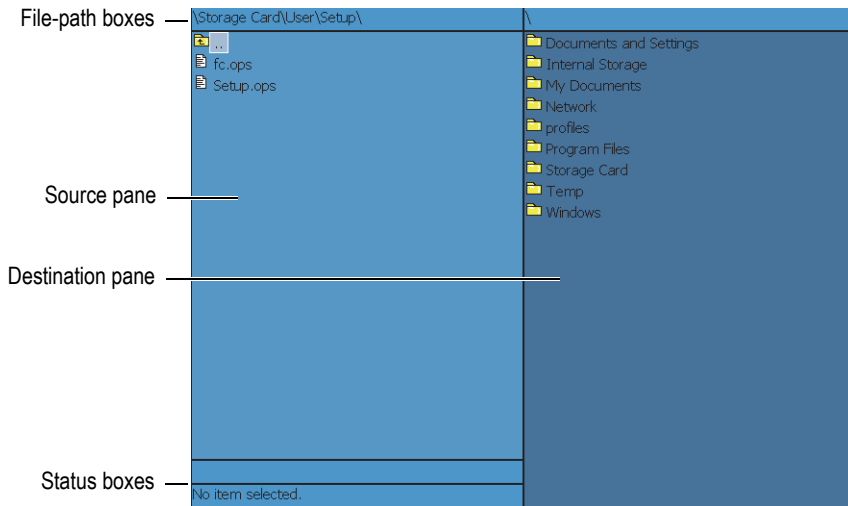


Figure 7-1 The File Manager areas

File-path boxes

These boxes display the path for the destination where the folders and files appearing in the pane are stored.

Source pane

The file management commands apply to the selected item (file or folder) in the source pane. This pane also indicates the source folder when a file or folder is being copied or moved.

Destination pane

This pane is only used with the **Copy** and **Move** parameters. It displays the destination folder for the file that is being copied or moved.

An item cannot be deleted or renamed in this pane, nor can an item be moved or copied to the source pane.

Status boxes

The upper status box displays file size information (when a file is selected), and the lower status box displays the name of the selected item.

Parameter buttons around the File Manager panes:

Close: Used to close the File Manager.

File Type: Used to select the type of file listed on the source pane. Use the Scroll knob to move up and down in the list, and the Accept key to select an item.

Create Folder: Used to create a new folder in the destination pane.

Select: Used to select a highlighted item from the list of files and folders; after an item is selected, the button name changes to **Unselect**.

Select All: Used to select all the items in the pane; after all the items are selected, the button name changes to **Unselect All**.

Copy: Used to copy a selected item from the source pane to the destination pane.

Move: Used to move a selected item from the source pane to the destination pane.

NOTE

The mouse cannot be used to copy or move files by dragging them across two panes, because the drag-and-drop feature is not supported.

Delete: Used to delete a selected item.

Rename: Used to rename a selected item.

7.1.2 File Manager Navigation

This section describes how to use the keys to move around in the File Manager areas.

Moving inside the pane

To move up and down in the list of folders and files, use the Scroll knob.

To select the highlighted folder or file, press the Accept key.

To open the parent folder, select the top folder, and then press the Accept key (see Figure 7-2 on page 185).



Figure 7-2 The parent folder icon

The file-path boxes show you where you are in the folder architecture (see Figure 7-1 on page 184).

Switching between panes

Use the Scroll knob to switch panes (the selected pane is pale blue), and then press the Accept key to select the pane. The source pane then turns pale blue (see Figure 7-3 on page 186).

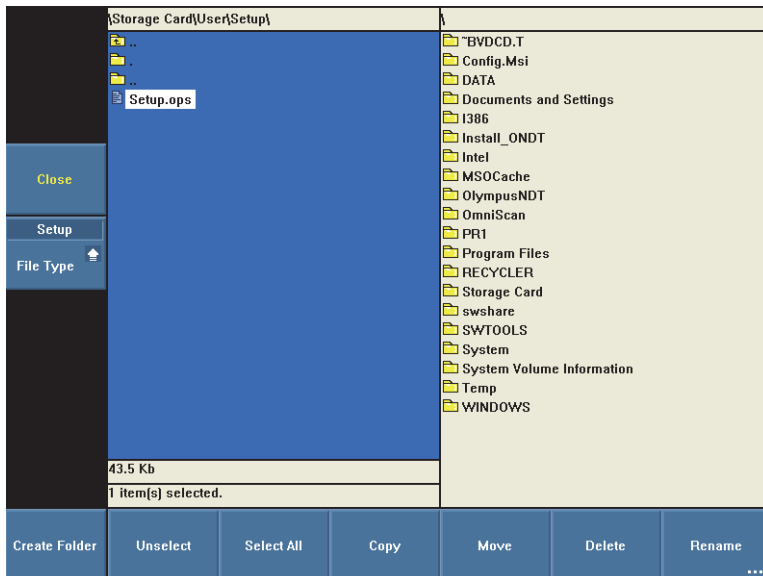


Figure 7-3 The source pane highlighted

7.2 Setting the Administrator Password

The administrator password of the OmniScan instrument is used to access the Microsoft Windows CE operating system and update the OmniScan MXU software.

To set the administrator password

1. On the OmniScan instrument, select **Preferences > Instrument > Category = System**.

2. Select **Preferences > Instrument > Admin. Password**.
3. In the dialog box that appears, enter a numeric password using the virtual keyboard or an external keyboard, and then press the Accept key.
4. In the dialog box used to confirm the password, enter the numeric password again, and then press the Accept key.
5. Keep a record of the entered password in a safe location.

7.3 Creating a Custom Report Template

This section explains how to create a custom report template from the standard OmniScan instrument report template.

A custom template can be created directly on the OmniScan instrument, provided that you have a network link between the OmniScan instrument and a computer for importing files, such as logo image files. However, the following procedure uses a computer to create the template. The procedure consists of copying the files of the standard report to the custom report folder, and customizing the copied files. The procedure makes use of the removable storage card of the OmniScan instrument to transfer the custom template file created on a computer to the OmniScan instrument.

To create a custom report template

1. Remove the storage card from the OmniScan instrument and, using a storage card reader, connect it to your computer.
2. Use Windows Explorer on your computer to open the necessary folder:
 - a) Open the storage card drive.

IMPORTANT

When performing this procedure, you must type the folder names exactly as they are spelled in the procedure. When you choose any other spelling or name, the OmniScan instrument does not recognize the folders and does not generate the custom report template.

- b) If it does not already exist in the storage card, create the following folder:
 \User\Template\Report\ (see Figure 7-4 on page 188).

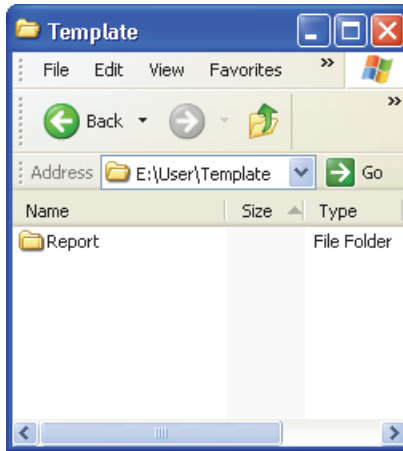


Figure 7-4 The Report folder path

3. Copy the \App\MXU 4.n\Template\Report\Complete\ folder to the \User\Template\Report\ folder that you created.
4. Rename the \User\Template\Report\Complete\ folder, which you just copied, using a name of your choice (renamed to My_Template in the example shown in Figure 7-5 on page 188).

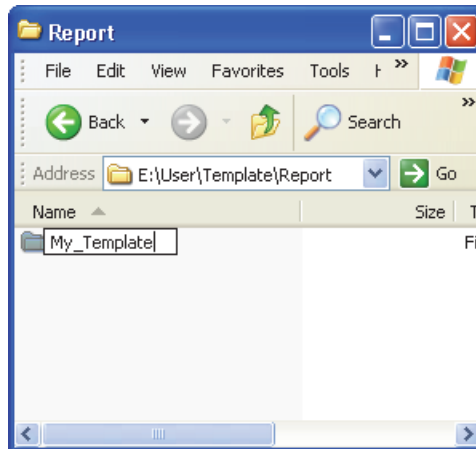


Figure 7-5 The path for the custom report files

**CAUTION**

Do not modify or replace any files in the original \App\MXU 4.n\Template\Report\Complete\ folder. These are the original files used to create the default report. When you replace or modify the original files, you no longer have them to create a custom template.

5. Copy your custom files (for example, your logo image file) to the folder you renamed in step 4.
6. Disconnect the storage card from your computer.
7. Reinstall the storage card in the OmniScan instrument.
8. Restart the OmniScan instrument to bring changes into effect.
9. To select the newly created report template, select **File > Report > Category = Format**.

The newly created report template appears in the **File > Report > Template** parameter under the name you used in step 4.

7.4 Changing the Company Logo

It is possible to replace the report logo with a logo of your choice.

**CAUTION**

Do not modify or replace any files in the Complete folder located in \App\MXU 4.n\Template\Report\. These files are the originals used to create the default report. If you replace or modify them, you will no longer have the original files required to create a custom template.

To change a company logo

1. Create a logo that does not exceed 130 × 60 pixels. A larger image affects the rest of the report page layout.
 2. Save it as **logo.jpg** (the name must be identical).
-

3. Create a custom report template as described in “Creating a Custom Report Template” on page 187.
4. In the renamed copy of the \User\Template\Report\Complete\ folder, copy the logo.jpg file you created in step 1.
The new logo is now included in the custom report template.

TIP

If your logo.jpg file does not appear in the report, try to open and save the logo.jpg file on a computer using the Windows Paint application, and recopy the saved file to the \User\Template\Report\Complete\ folder.

7.5 Creating a Color Palette

The color palettes of the OmniScan instrument (.pal files) provide 256 colors, and the definition for the following three special colors: no-data, no-detection, and no-synchro.

No-data

A zone which is not inspected.

No-detection

An inspected zone with no signal has crossed gate A.

No-synchro

An inspected zone with no signal has crossed gate I; therefore, there is no detection.

In the range of 256 colors, each color is defined as a code in the RGB color scale.

An RGB color code has four components. The first three components each represent the intensity of one of the three basic RGB colors: red (R), green (G), and blue (B). The fourth and last component is an RGB-reserved field that is not used with the Microsoft Windows CE operating system, although it is essential for proper color palette file usage. The number 0 (zero) must always be used as the fourth component.

The different intensity combinations determine the data view colors. Each RGB component has 256 possible values, ranging from 0 through 255.

Table 19 on page 191 explains how to obtain colors using the numerical components.

Table 19 RGB color codes

Color	Red	Green	Blue
Red	255	0	0
Green	0	255	0
Blue	0	0	255
Yellow	255	255	0
Purple	255	0	255
Turquoise	0	255	255
Black	0	0	0
Grey	125	125	125
White	255	255	255

The closer the number is to 0, the darker the color. The closer the number is to 255, the lighter the color. Choosing a color code made from three identical numbers produces black, white, or a shade of grey.

7.5.1 Color Palette File Format

An OmniScan .pal file contains a header that allows the user to identify the XML format version. The file header is as follows:

```
<?xml version='1.0'?>
<!-- This file represents an OmniScan palette colors definition -->
```

The second line of this header is a comment. If you want to modify the comment, it is important to use the following characters:

```
<!-- followed by a space at the beginning of the comment,
--> preceded by a space at the end of the comment.
```

The comment can have many lines.

It is important to include an identifier at the beginning of the palette definition. This identifier is:

<Palette>

The first line of a palette definition must be the version number of the palette definition format. Currently, the version number is 2.0. The line is as follows:

```
<Version>2.0</Version>
```

Next, there are special colors and palette color definitions. The format of the color definition line is as follows:

```
<Color R="RValue" G="GValue" B="BValue" F="FValue"></Color>
```

For example, to define white, the entry should be:

```
<Color R="255" G="255" B="255" F="0"></Color>
```

As mentioned before, the last component should always be 0, and should always be included in the definition.

Therefore, following the version number of the palette definition, you must find the special colors definition:

- The "no-data" color is on the first line;
- The "no-detection" color is on the second line;
- The "no-synchro" color is on the third line.

To differentiate between the special colors, it is important to frame the definition between two identifiers. The identifier located at the beginning of the special colors definition is the following:

```
<SpecialColors>
```

The end identifier of the special colors definition is the following:

```
</SpecialColors>
```

The following is an example that defines the "no-data" to black, the "no-detection" to white, and the "no-synchro" to grey:

```
<SpecialColors>  
<Color R="0" G="0" B="0" F="0"></Color>  
<Color R="255" G="255" B="255" F="0"></Color>  
<Color R="128" G="128" B="128" F="0"></Color>  
</SpecialColors>
```

The definition of the 256 colors of the palette follows. To differentiate between the colors of the palette, it is important to frame their definition between two identifiers. The identifier located at the beginning of the palette color definition is the following:

```
<MainColors>
```

The end identifier of the definition of the palette color is the following:

```
</MainColors>
```

The following is an example of the palette definition in hues of grey:

```
<MainColors>
<Color R="0" G="0" B="0" F="0"></Color>
<Color R="1" G="1" B="1" F="0"></Color>
<Color R="2" G="2" B="2" F="0"></Color>
. . .
<Color R="255" G="255" B="255" F="0"></Color>
</MainColors>
```

To close the file, it is important to add an end identifier at the end of the palette definition:

```
</Palette>
```

7.5.2 Color Palette Rules

The following rules must be observed in order to create an OmniScan-compatible color palette file (.xml):

- The first item needed is the .xml version number. These lines must be written exactly as described in “Color Palette File Format” on page 191.
- The second item needed is the <Palette> identifier at the beginning of the palette definition.
- The first line needed in the palette definition is the version number of the definition. This line must be written as described in “Color Palette File Format” on page 191.
- Always include the special colors definition before the palette colors definition.
- Always include the beginning and end definition identifiers for the colors definition. The identifiers must be written exactly as described in “Color Palette File Format” on page 191.
- Always include three special colors.
- Always include the 256 colors of the palette.
- The order of the special colors is “no-data,” “no-detection,” and “no-synchro.”
- The first line of the palette color definition is the first palette color.
- The last line of the palette color definition is the last palette color.

- An identifier made of a name enclosed in <> signs without spaces starts and ends a definition.
- Always respect the color definition format, as described in “Color Palette File Format” on page 191.
- Always include the end identifier at the end of the palette definition. This line must be written exactly as described in “Color Palette File Format” on page 191.
- An identifier must always be written on one line only.
- An identifier must be alone on the line.
- A color definition must be written on one line only.
- A color definition must be alone on the line.
- In order to ease the file reading, it is possible to add spaces at the beginning of the lines to align the groups of information. It is also possible to add empty lines.
- The file name extension must be .pal.

The following is an example of a palette definition file. Note that not all 256 colors are used in the example.

```
<?xml version='1.0'?>
  <!-- This file represents an OmniScan palette color definition -->

<Palette>
  <Version>2.0</Version>

  <SpecialColors>
    <Color R="0" G="0" B="0" F="0"></Color>
    <Color R="255" G="255" B="255" F="0"></Color>
    <Color R="128" G="128" B="128" F="0"></Color>
  </SpecialColors>
  <MainColors>
    <Color R="255" G="255" B="255" F="0"></Color>
    <Color R="250" G="252" B="254" F="0"></Color>
    <Color R="246" G="250" B="253" F="0"></Color>
    <Color R="142" G="6" B="29" F="0"></Color>
    <Color R="140" G="3" B="29" F="0"></Color>
  </MainColors>
</Palette>
```

7.6 Characterizing a Probe

This section describes how to characterize a probe.

To characterize a probe

1. Position the probe over an object with a spherical reflector. The best configuration is a small metal ball (diameter of 2.5 mm) in water.
2. Select **UT Settings > Receiver**, and then proceed as follows:
 - a) Select **Filter = None**.
 - b) Select **Video Filter = Off**.
 - c) Select **Rectifier = RF**.
3. Select **UT Settings > General**.
4. Using the **Start** and **Range** parameter buttons, adjust the time-base scale on the A-scan so that you only see the signal reflected by the ball in the view.

NOTE

Use a very small range.

5. Select **Gates/Alarms > Gates > Gate = A**.
6. Select **Parameter = Position**.
7. Using the **Start** and **Width** parameter buttons, set gate A across the signal, as shown in Figure 7-6 on page 195.

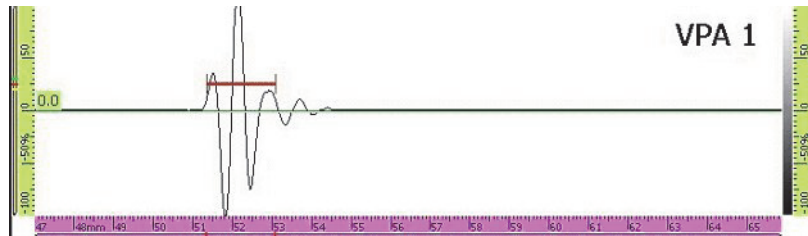


Figure 7-6 Gate A on the ball signal

8. Select **Preferences > FFT**, and proceed as follows:
 - a) If applicable, select **Procedure**, and then enter the name of the characterization procedure.
 - b) Select **Block Name**, and then enter the name of the part used in the characterization.

- c) Select **FFT = ON**.
- d) In the message that appears asking if you want to save the characterization, select **Yes**.

7.7 Connecting the OmniScan MX2 Directly to a Computer

This section explains how to set up a direct network connection between the OmniScan MX2 and a computer. When the network connection between the OmniScan MX2 and the computer has been configured, you can save the OmniScan data directly to a shared folder on the hard disk of the computer.

You need to perform the following procedures to enable data transfer from the OmniScan MX2 to the computer. Use the appropriate procedure for your computer's operating system.

- "Connecting the Hardware" on page 196

For Windows XP:

- "Configuring the Computer Network Connection — Windows XP" on page 197
- "Creating a User Account on a Computer — Windows XP" on page 200
- "Creating a Shared Folder on a Computer — Windows XP" on page 203
- "Configuring OmniScan MX2 Network Connection — Windows XP" on page 209

For Windows 7:

- "Configuring the Computer Network Connection — Windows 7" on page 211
- "Creating a User Account on a Computer — Windows 7" on page 214
- "Creating a Shared Folder on a Computer — Windows 7" on page 217
- "Configuring the OmniScan MX2 — Windows 7" on page 222

Finally, the procedure entitled "Saving OmniScan Data on a Computer" on page 223 describes how to transfer data from the OmniScan MX2 to the computer.

7.7.1 Connecting the Hardware

You need a crosslink Ethernet (RJ-45) cable to interconnect the OmniScan MX2 and the computer. A crosslink Ethernet cable allows direct connection between two computers, as opposed to the usual network cable that allows connection between a computer and a hub or wall connection.

NOTE

You need to have two network ports on your computer if you want to connect your computer concurrently to a local area network and directly to the OmniScan MX2. If your computer is only equipped with one network port, you can use an adaptor.

To connect the hardware under Windows XP or Windows 7

1. Connect a crosslink Ethernet (RJ-45) cable to the network port of the computer.
2. Connect the other end of the cable to the OmniScan MX2 Ethernet port.

7.7.2 Configuring the Computer Network Connection — Windows XP

This section describes how to configure the network connection of your computer under Windows XP in order to ensure communication with the OmniScan MX2.

To configure the network connection on the computer under Windows XP

1. On the computer, click **Control Panel**.
2. Open **Network Connections**.
3. Double-click the connection used to establish the link with the OmniScan MX2. The **Local Area Connection Status** dialog box appears (see Figure 7-7 on page 198).

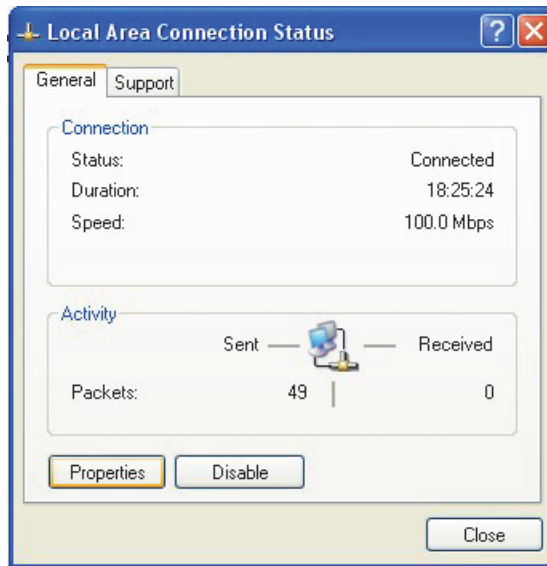


Figure 7-7 The Local Area Connection Status dialog box (Windows XP)

4. On the **General** tab, click **Properties**.

The **Properties** dialog box for the selected connection appears (see Figure 7-8 on page 199).

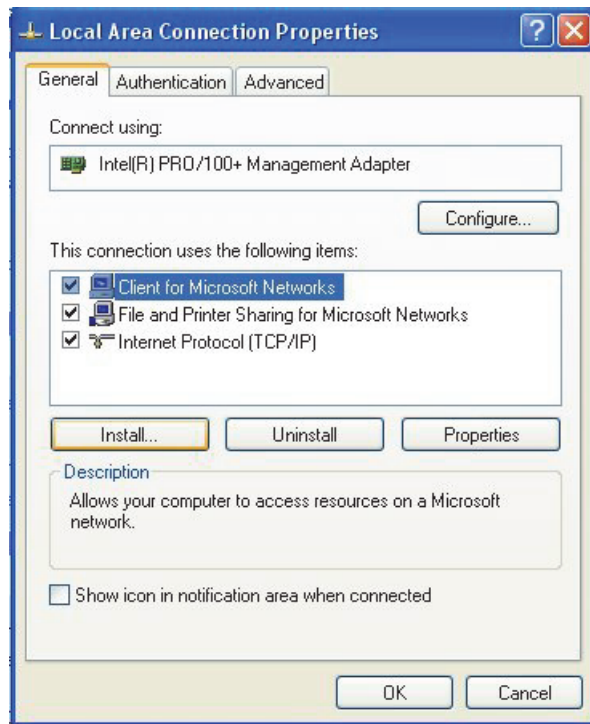


Figure 7-8 The Properties dialog box for selected connection (Windows XP)

5. Make sure that the three following check boxes are selected in the list entitled **This connection uses the following items:**
 - **Client for Microsoft Networks**
 - **File and Printer Sharing for Microsoft Networks**
 - **Internet Protocol (TCP/IP)**
6. Click **Properties**.
7. In the **Internet Protocol (TCP/IP) Properties** dialog box, select **Use the following IP address** (see Figure 7-9 on page 200).

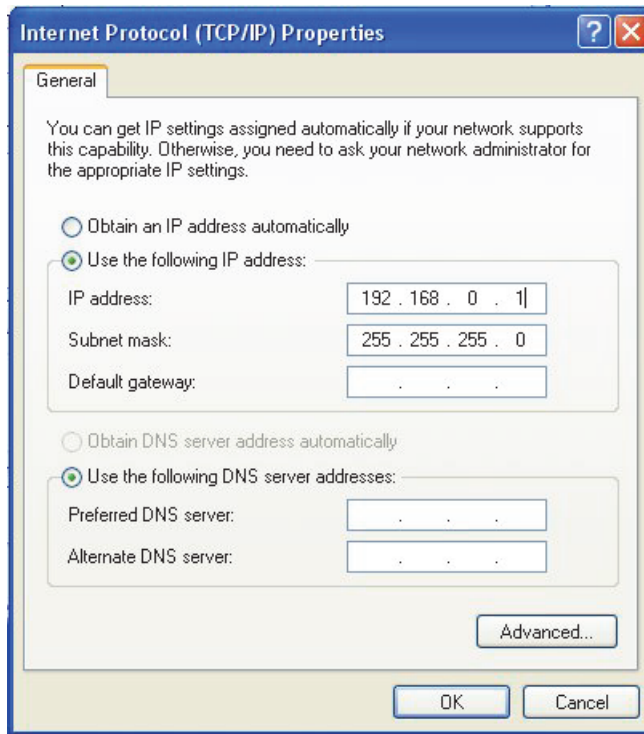


Figure 7-9 The Internet Protocol (TCP/IP) Properties dialog box (Windows XP)

8. In the **IP address** text box, type an OmniScan-compatible IP address (see “Configuring OmniScan MX2 Network Connection — Windows XP” on page 209).
9. In the **Subnet Mask** text box, type an OmniScan-compatible Subnet Mask (see “Configuring OmniScan MX2 Network Connection — Windows XP” on page 209).
10. Click **OK** to confirm the changes, and then close the dialog box.

7.7.3 Creating a User Account on a Computer — Windows XP

This section describes how to create and configure a user account on your computer under Windows XP in order to allow communication between the OmniScan MX2 and the computer.

To create a user account on a computer under Windows XP

1. On the Windows XP desktop, right-click the **My Computer** icon.
2. On the shortcut menu, click **Manage** (see Figure 7-10 on page 201).

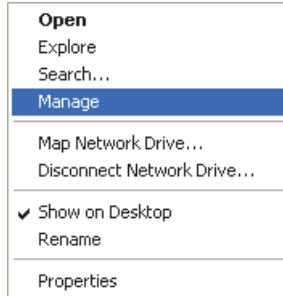


Figure 7-10 The shortcut menu for the My Computer icon

The **Computer Management** dialog box appears (see Figure 7-11 on page 201).

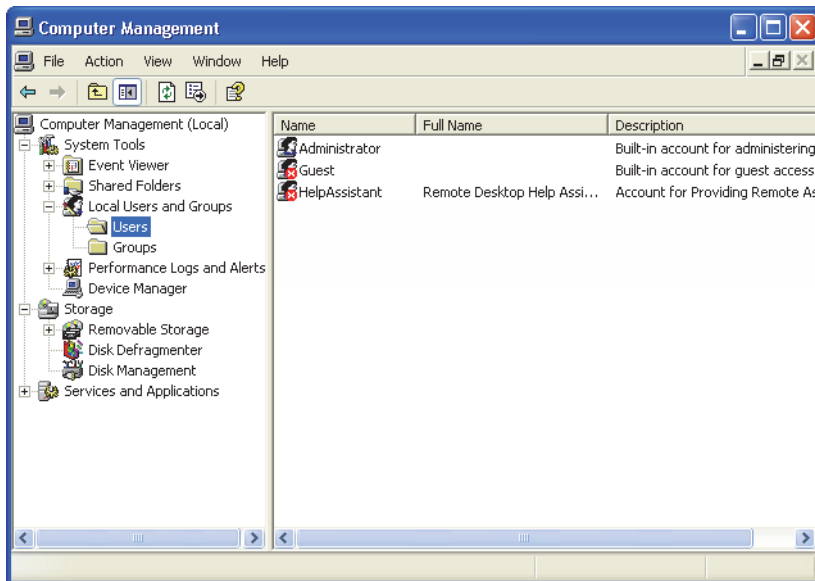


Figure 7-11 The Computer Management dialog box (Windows XP)

3. In the left pane, double-click **Local Users and Groups**, and then click the **Users** folder.
4. On the **Action** menu, select **New User**.
5. In the **User name** text box, in the **New User** dialog box, type **Omniscan**, respecting the case of the letters (see Figure 7-12 on page 202).

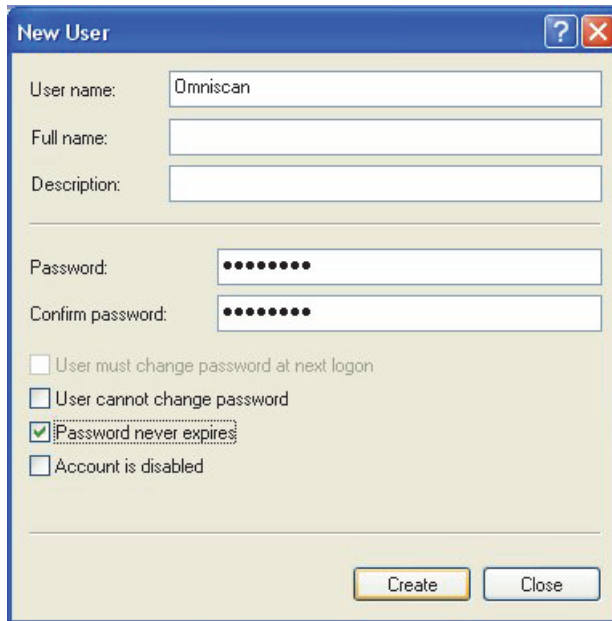


Figure 7-12 The New User dialog box (Windows XP)

6. In the **Password** and **Confirm password** boxes, type **omniscan** (all lowercase letters).
7. Clear the following check boxes:
 - **User must change password at next logon**
 - **User cannot change password**
 - **Account is disabled**
8. Select the **Password never expires** check box.
9. Click **Create**.

The parameters are saved and the **New User** dialog box closes.

10. Close the **Computer Management** window.

7.7.4 Creating a Shared Folder on a Computer — Windows XP

This section describes how to create and configure a shared folder on your computer under Windows XP in order to allow file transfers between the OmniScan MX2 and the computer.

To create a shared folder on the computer under Windows XP

1. Using Windows Explorer, create a folder named “Omnican” on the hard disk drive of the computer.
2. Right-click the **Omnican** folder, and then click **Sharing and Security** on the shortcut menu.
3. In the **Omnican Properties** dialog box, click the **Sharing** tab (see Figure 7-13 on page 203).

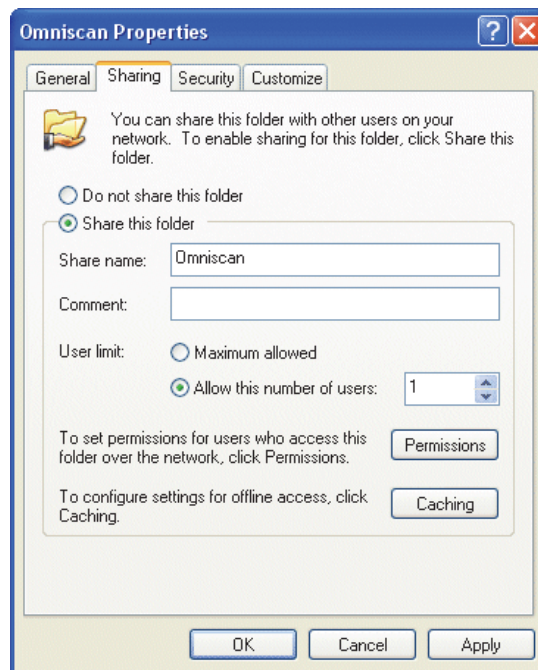


Figure 7-13 The Sharing tab of the Omnican Properties dialog box (Windows XP)

4. Select the **Share this folder** option.
5. If the **Permissions** button does not appear in the **Share this folder** group box, proceed as follows:
 - a) On the computer desktop, double-click the **My Computer** icon.
 - b) On the **Tools** menu, click **Folder Options**.
 - c) In the **Folder Options** dialog box, click the **View** tab (see Figure 7-14 on page 204).

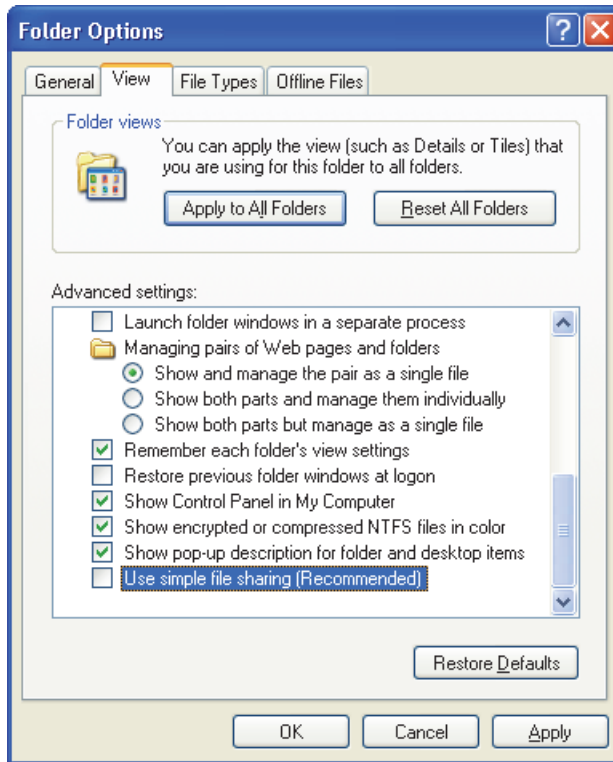


Figure 7-14 The Folder Options dialog box (Windows XP)

- d) Clear the **Use simple file sharing** check box, and then click **OK**.
6. In the **Omniscan Properties** dialog box, click the **Permissions** button.
7. In the **Permissions for Omniscan** dialog box, click **Add** (see Figure 7-15 on page 205).

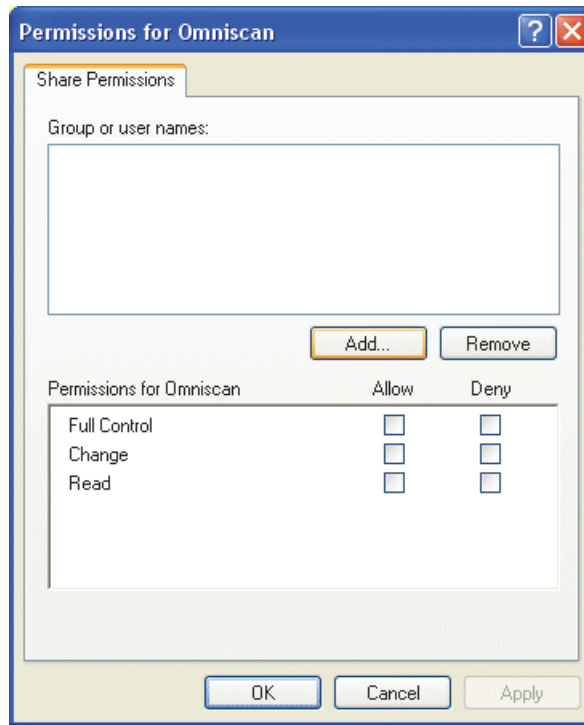


Figure 7-15 The Permissions for Omniscan dialog box (Windows XP)

8. In the **Select Users or Groups** dialog box, click the **Locations** button (see Figure 7-16 on page 206).

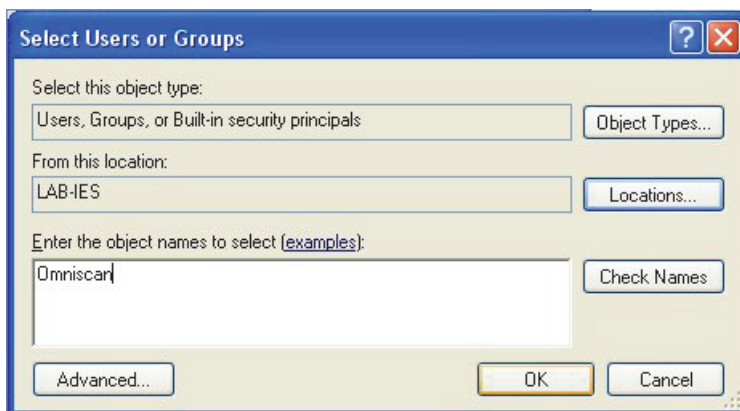


Figure 7-16 The Select Users or Groups dialog box (Windows XP)

9. In the **Locations** dialog box, select the name of your computer, and then click **OK**.
10. Back in the **Select Users or Groups** dialog box, type **Omniscan** (see Figure 7-16 on page 206) in the **Enter the object names to select** text box, and then click **OK**.
11. Back in the **Permissions for Omniscan** dialog box, select the **Change** and **Read** check boxes in the **Allow** column, and then click **OK** (see Figure 7-17 on page 207).

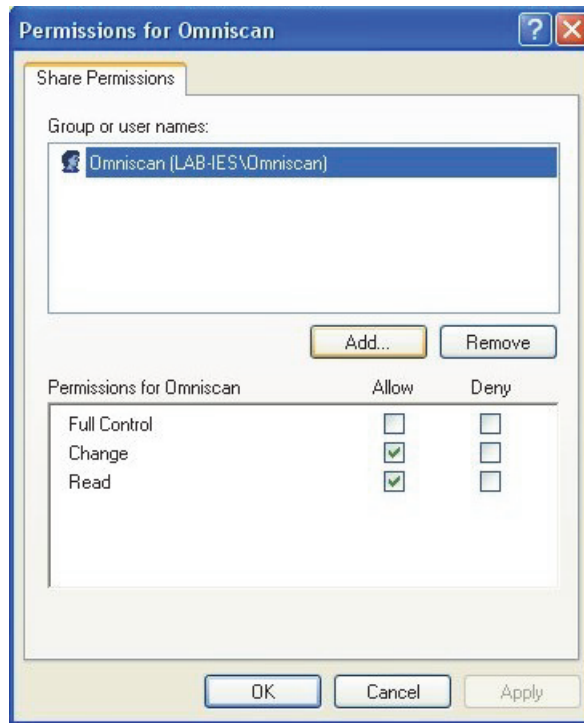


Figure 7-17 The Permissions for Omniscan dialog box (Windows XP)

12. In the **Omniscan Properties** dialog box, click the **Security** tab, and then click **Add** (see Figure 7-18 on page 208).

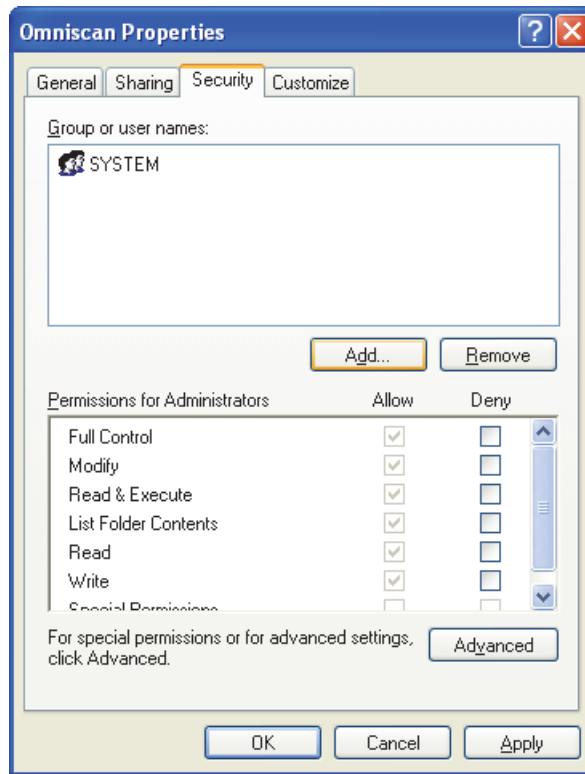


Figure 7-18 The Security tab of the Omniscan Properties dialog box (Windows XP)

13. In the **Select Users or Groups** dialog box, click the **Locations** button (see Figure 7-16 on page 206).
14. In the **Locations** dialog box, select the name of your computer, and then click **OK**.
15. Back in the **Select Users or Groups** dialog box, type **Omniscan** (see Figure 7-16 on page 206) in the **Enter the object names to select** text box, and then click **OK**.
16. Back in the **Security** tab of the **Omniscan Properties** dialog box (see Figure 7-19 on page 209):
 - a) Select the added Omniscan user.
 - b) In the **Permissions for Omniscan** list, select the following check boxes: **Modify**, **Read & Execute**, **List Folder Contents**, **Read**, and **Write**.
 - c) Click **OK**.

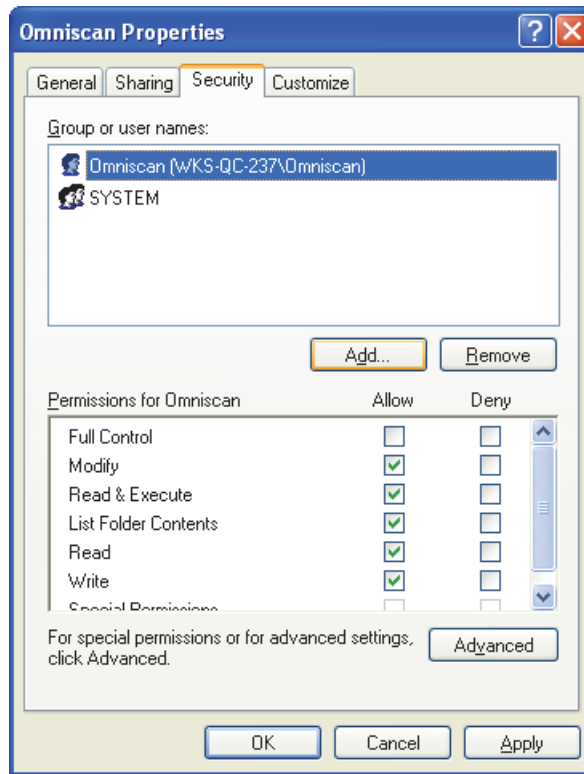


Figure 7-19 Security permissions for the Omniscan user

7.7.5 Configuring OmniScan MX2 Network Connection — Windows XP

This section describes how to configure the OmniScan MX2 to allow communication between the OmniScan MX2 and the computer.

To configure the OmniScan MX2 network connection under Windows XP

1. On the OmniScan MX2, select **Preferences > Instrument > Category = Network Settings**.
2. Select **DHCP = Off**.
3. Select **IP Address**, and then enter an IP address for the OmniScan MX2. It must be in the same range (the same first three blocks), but not identical to the IP address of the computer (for example, 192.168.0.1 and 192.168.0.2).

4. Select **Subnet Mask**, and then enter the subnet mask address for the OmniScan MX2. It must be the same as the subnet mask set on the computer.
5. Select **Apply**.
6. Select **Preferences > Instrument > Category = External Storage**.
7. Select **Remote PC**, and then enter the computer name. To find the computer name, proceed as follows:
 - a) On the computer, click **Start** on the Windows XP taskbar.
 - b) On the Start Menu, right-click **My Computer**.
 - c) On the shortcut menu, click **Properties**.
The **System Properties** dialog box appears.
 - d) Click the **Computer Name** tab.
The computer name is the first part of the **Full computer name**, which excludes the **Domain** name (**wks-qc-150** in the example in Figure 7-20 on page 211).

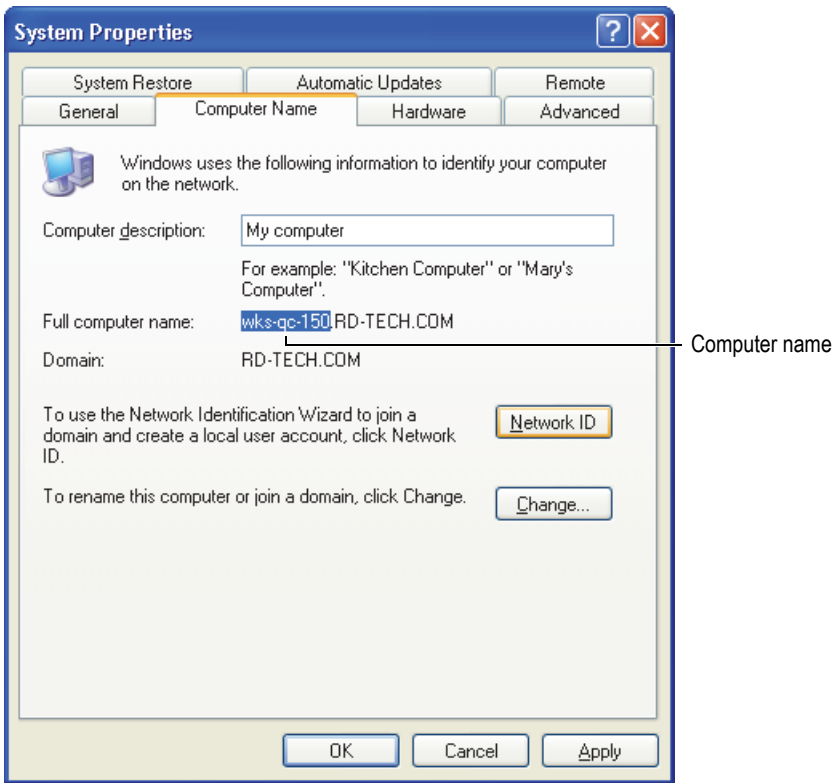


Figure 7-20 The computer name found in the System Properties dialog box

8. Select **Preferences > Instrument > Category = External Storage**, and then select **Connect = On**.

The connection is established between the OmniScan MX2 and the computer when the **Connect** button changes to **On**.

7.7.6 Configuring the Computer Network Connection — Windows 7

To configure the computer under Windows 7

1. On the remote computer, open the **Control Panel** window.
2. Open the **Network and Sharing Center** (in **Network and Internet** if **Control Panel** is viewed by category).

3. Click on the Local Area Connection corresponding to the connection used to establish the link with the OmniScan MX2.

The **Local Area Connection Status** dialog box appears (see Figure 7-21 on page 212).

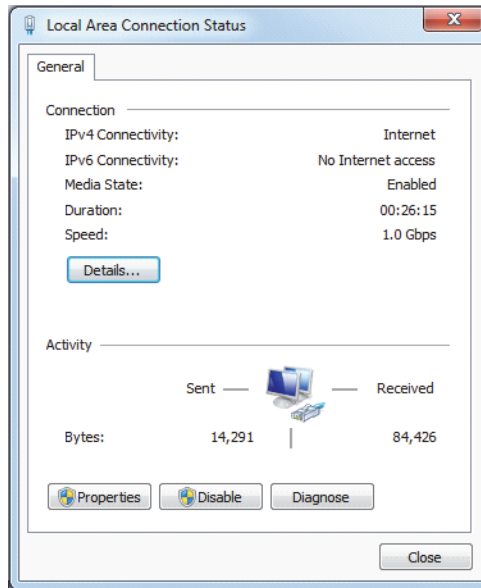


Figure 7-21 The Local Area Connection Status dialog box (Windows 7)

4. On the **General** tab, click **Properties**.

The Properties dialog box for the selected connection appears (see Figure 7-22 on page 213).

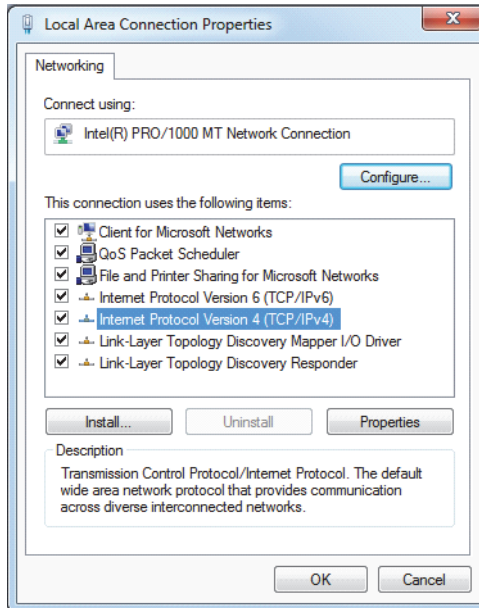


Figure 7-22 The Local Area Connection Properties dialog box (Windows 7)

5. In the list entitled **This connection uses the following items**, make sure the three following check boxes are selected:
 - **Client for Microsoft Networks**
 - **File and Printer Sharing for Microsoft Networks**
 - **Internet Protocol Version 4 (TCP/IPv4)**
6. Select (highlight) **Internet Protocol Version 4 (TCP/IPv4)**, and then click **Properties**.
7. In the **Internet Protocol Version 4 (TCP/IPv4) Properties** dialog box, select **Use the following IP address** (see Figure 7-23 on page 214).

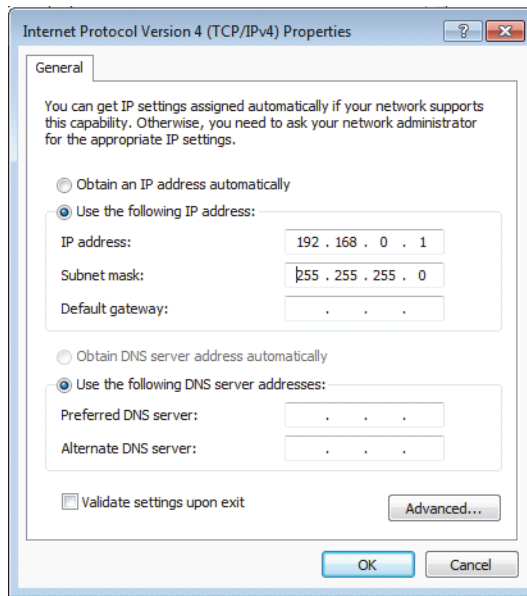


Figure 7-23 The Internet Protocol Version 4 (TCP/IPv4) Properties dialog box (Windows 7)

8. In the **IP address** box, type an OmniScan-compatible IP address (see “Configuring the OmniScan MX2 — Windows 7” on page 222).
9. In the **Subnet Mask** box, type an OmniScan-compatible subnet mask (see “Configuring the OmniScan MX2 — Windows 7” on page 222).
10. Click **OK** to confirm the changes, and then close the dialog box.

7.7.7 Creating a User Account on a Computer — Windows 7

To create a user account on the computer

1. On the Windows 7 desktop (or in the **Start** menu), right-click the **Computer** icon.
2. On the shortcut menu, click **Manage** (see Figure 7-24 on page 215).

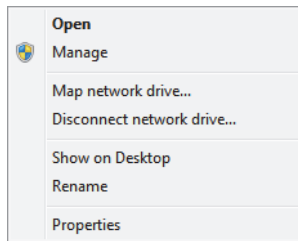


Figure 7-24 The shortcut menu for the Computer icon (Windows 7)

The **Computer Management** dialog box appears (see Figure 7-25 on page 215).

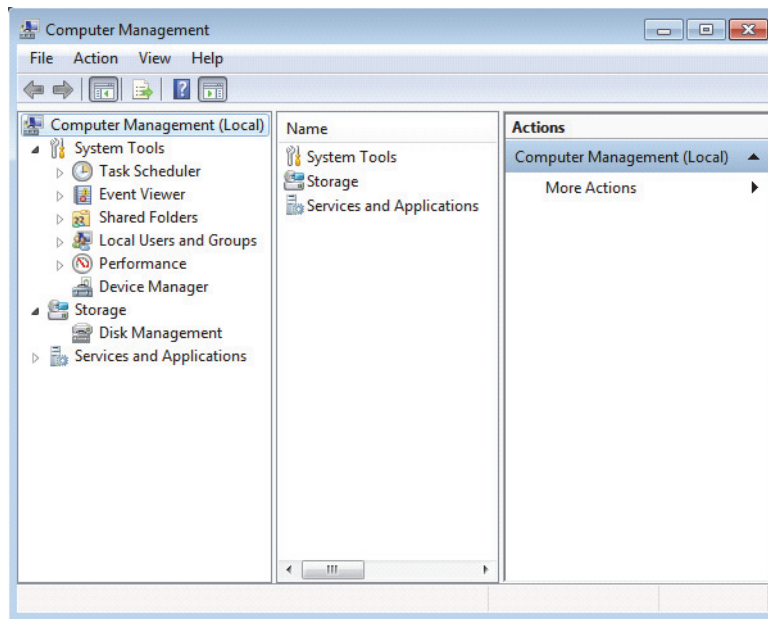


Figure 7-25 The Computer Management dialog box (Windows 7)

3. In the left pane, double-click **Local Users and Groups**, and then click the **Users** folder.
4. In the **Action** menu, select **New User**.

5. In the **User name** box of the **New User** dialog box, type **Omniscan**, respecting the case of the letters (see Figure 7-26 on page 216).

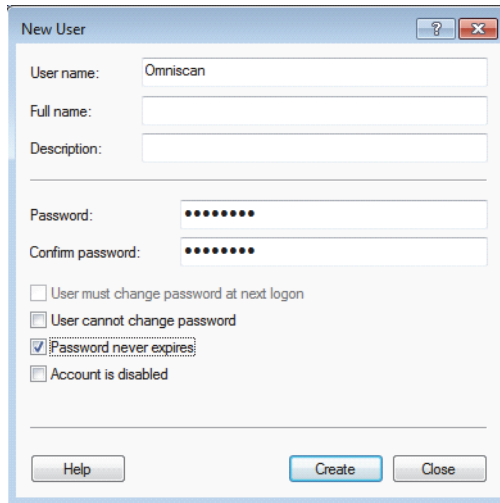


Figure 7-26 The New User dialog box (Windows 7)

6. In the **Password** and **Confirm password** boxes, type **omniscan** (all lowercase letters).
7. Clear the following check boxes:
 - **User must change password at next logon**
 - **User cannot change password**
 - **Account is disabled**
8. Select the **Password never expires** check box.
9. Click **Create**.

The parameters are saved.
10. Close the **New User** dialog box.
11. Close the **Computer Management** dialog box.

7.7.8 Creating a Shared Folder on a Computer — Windows 7

To create a shared folder on the remote computer

1. On the hard disk drive of the remote computer, create a folder named “Omniscan”.
2. Right-click the **Omniscan** folder, and then click **Properties** on the shortcut menu.
3. In the **Omniscan Properties** dialog box, click the **Sharing** tab (see Figure 7-27 on page 217).
4. Click **Share**.

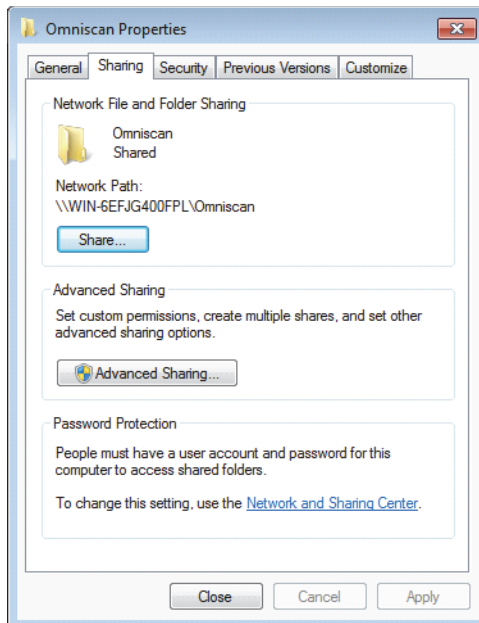


Figure 7-27 The Omniscan Properties dialog box (Windows 7)

5. In the **File Sharing** dialog box (see Figure 7-28 on page 218), select **Omniscan**, and then click **Add**.

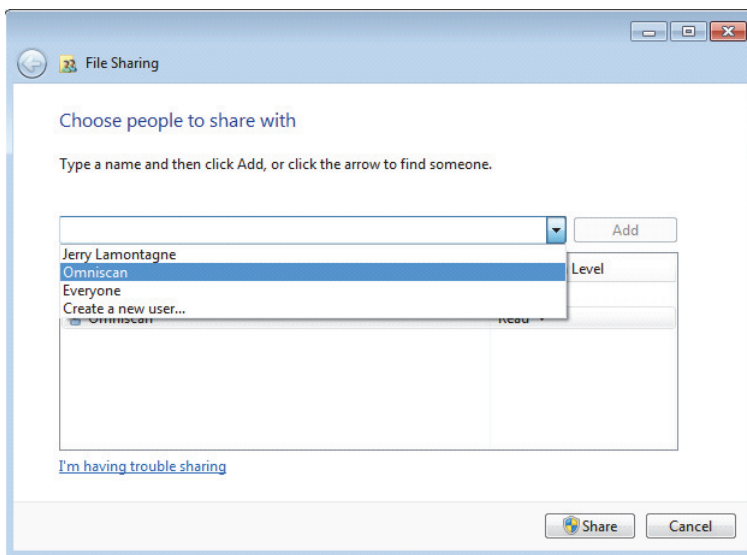


Figure 7-28 The File Sharing dialog box (Windows 7)

6. Under **Permission Level**, select **Read/Write** for **Omniscan** (see Figure 7-29 on page 219).

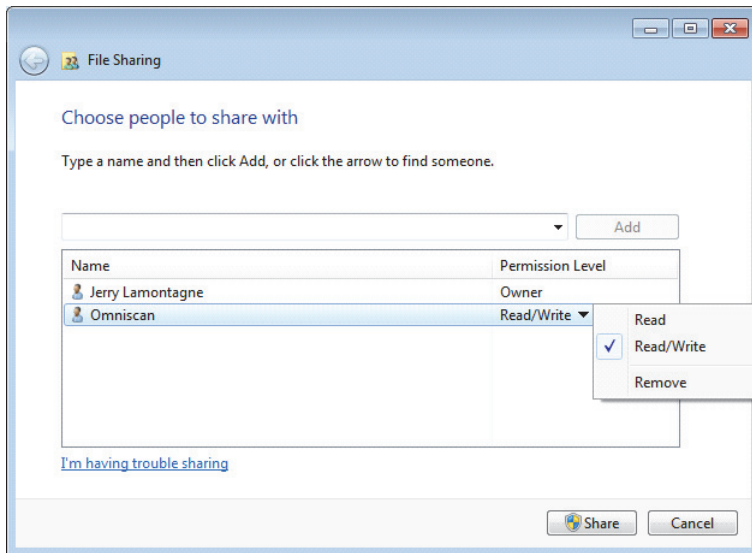


Figure 7-29 Setting Permissions in the File Sharing dialog box (Windows 7)

7. Click **Share**, and then, if asked, select **Yes, turn on network discovery and file sharing for all public networks**. Close the dialog box.
8. In the **Omniscan Properties** dialog box, click **Advanced Sharing**. (see Figure 7-27 on page 217).
The **Advanced Sharing** dialog box appears (see Figure 7-30 on page 220).
9. Click the **Permissions** button.

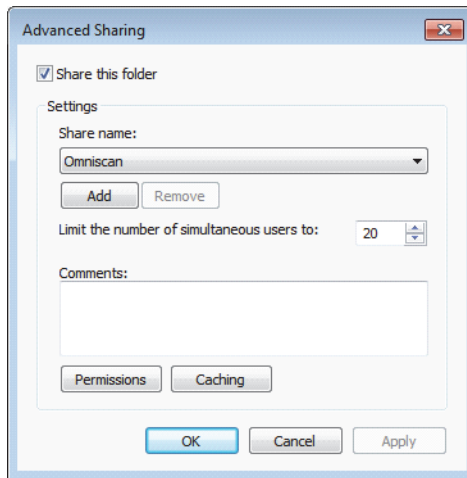


Figure 7-30 The Advanced Sharing dialog box (Windows 7)

10. In the **Permissions for Omniscan** dialog box, click **Add** (see Figure 7-31 on page 220).

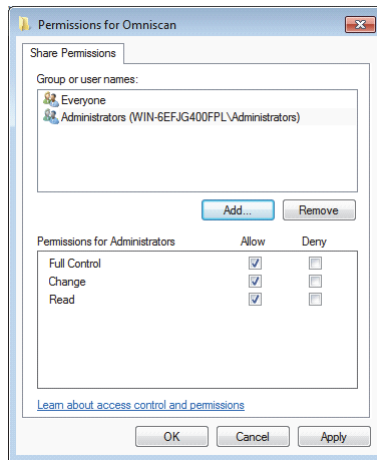


Figure 7-31 The Permissions for Omniscan dialog box (Windows 7)

11. In the **Select Users or Groups** dialog box, click the **Locations** button (see Figure 7-32 on page 221).

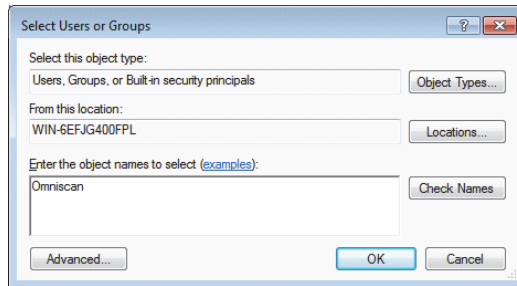


Figure 7-32 The Select Users or Groups dialog box (Windows 7)

12. In the dialog box that appears, select the name of the remote computer, and then click the **OK** button.
13. In the **Enter the object names to select** box of the **Select Users or Groups** dialog box, type **Omniscan** (see Figure 7-32 on page 221), and then click **OK**.
14. In the **Permissions for Omniscan** dialog box, select the **Change** and **Read** check boxes in the **Allow** column for the Omniscan user, and then click **OK** (see Figure 7-33 on page 222).

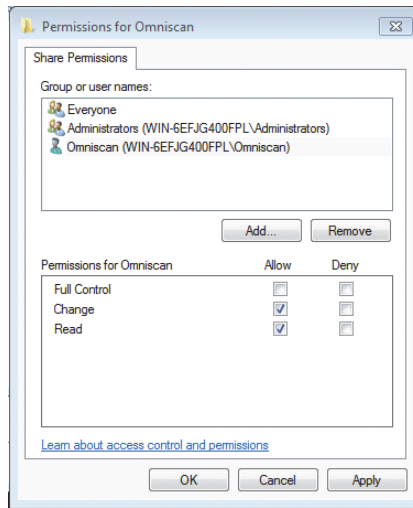


Figure 7-33 The Permissions for Omniscan dialog box (Windows 7)

15. Close all open dialog boxes.

7.7.9 Configuring the OmniScan MX2 — Windows 7

To configure the OmniScan MX2 under Windows 7

1. On the OmniScan MX2, select **Preferences > Instrument > Category = Network Settings**.
2. Select **DHCP = Off**.
3. Select **IP Address**, and then enter an IP address for the OmniScan MX2. It must be in the same range (the same first three blocks), but not identical to the IP address of the remote computer (for example, 192.168.0.1 and 192.168.0.2).
4. Select **Subnet Mask**, and then enter the subnet mask address for the OmniScan MX2. It must be the same as the subnet mask set on the remote computer.
5. Select **Apply**.
6. Select **Preferences > Instrument > Category = External Storage**, select **Remote PC**, and then enter the remote computer name. To get the remote computer name, proceed as follows:

- a) On the remote computer, click **Start** on the Windows taskbar.
- b) On the **Start** menu, right-click **Computer**, and then select **Properties**.

The **System** dialog box appears. The remote computer name is given in the **Computer Name** field (WIN-6EFJG400FPL in the example of Figure 7-34 on page 223).

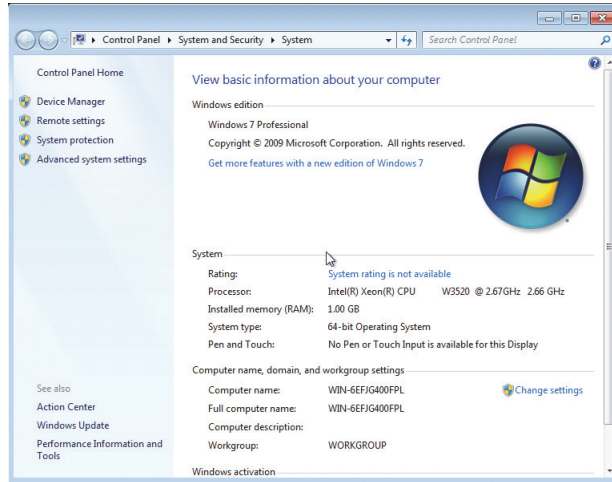


Figure 7-34 Computer name in System Properties dialog box (Windows 7)

7. Select **Preferences > Network > Connect = On**.

The connection is established between the OmniScan MX2 and the remote computer when the **Connect** button changes to **On**.

7.7.10 Saving OmniScan Data on a Computer

This section describes how to save OmniScan data in the shared folder on the hard disk of your computer.

IMPORTANT

You must interconnect the OmniScan MX2 and the computer, configure the computer, and configure the OmniScan MX2 before attempting to save data from the OmniScan MX2 to the computer. See “Connecting the OmniScan MX2 Directly to a Computer” on page 196 for the links to the necessary procedures.

To save OmniScan data on the computer under Windows XP or Windows 7

- ◆ Select **File > Data Settings > Storage = Network**.

From now on, all saved data will be written to the shared folder on the hard disk of your computer.

7.8 Connecting the OmniScan MX2 to a Network

This section describes how to establish a connection between the OmniScan MX2 and a local area network. This procedure works for Window XP and Windows 7 systems.

To connect the OmniScan MX2 to a network

1. Connect a straight Ethernet (RJ-45) cable to a network port (usually a wall socket).
2. Connect the other end of the cable to the OmniScan MX2 Ethernet port.
3. On the OmniScan MX2 instrument, select **Preferences > Instrument > Category = Network Settings**.
4. Select **DHCP = On**.
5. Select **Apply**.
6. Select **Preferences > Instrument > Category = External Storage**.
7. Select **Remote PC**, and then enter the computer name as seen from the network.
8. Select **Connect = On**.

The connection is established between the OmniScan MX2 and the computer when the **Connect** button changes to **On**.


7.9 Importing OmniScan Data into TomoView

This section describes how to import data from the OmniScan MX2 into TomoView.

NOTE

To import data into TomoView, the data must be acquired using either the **All A & C scans** or **C-scans Only** storage mode (select **File > Data Settings > Data Selection**).

To import OmniScan data into TomoView

1. Select **File > Data Settings > Save Key = Data**.
2. Select **File > Data > Save**.
3. Transfer the file to a computer by using a memory card reader or by linking the OmniScan MX2 to the computer through the Ethernet port. For more information about networking the OmniScan MX2, see “Connecting the OmniScan MX2 Directly to a Computer” on page 196.
4. Make sure that the appropriate hardware key is connected to the USB port of the computer.
5. Start the TomoView software.
6. After TomoView has finished loading, click the OmniScan button ()
A dialog box appears.
7. Browse and select the file to be imported.
8. Click **Open**.
TomoView converts the file to the .rdt format.
9. Save the new .rdt file in the desired folder.

NOTE

For more information on configuring and analyzing the file data, consult the *TomoView User's Manual*.

7.10 Transferring Data — OmniScan MX2 with MCDU-02 and TomoView

When you use the OmniScan MX2 with an MCDU-02 and a computer running TomoView, you must connect the three components with network cables. You can maximize the data transfer rate between the OmniScan MX2 and TomoView by interconnecting the three units using an external Ethernet hub (see Figure 7-35 on page 226).

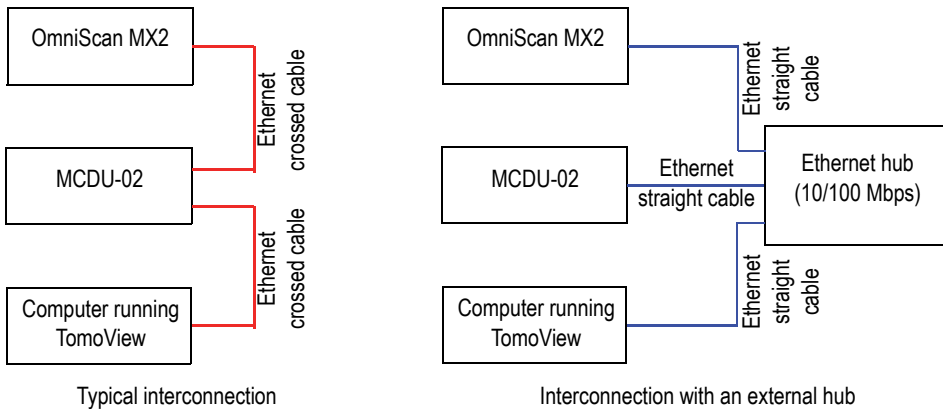


Figure 7-35 Interconnections between OmniScan MX2, MCDU, and computer

7.11 Importing a .law File into the OmniScan Instrument (PA Group Only)

Using the Olympus NDT SetupBuilder or Advanced Calculator application, you can rapidly and efficiently generate and visualize focal laws with various types of phased array probes and wedges. The NDT SetupBuilder and Advanced Calculator applications save focal law data in a text file format with the .law extension. You can take advantage of this data by importing the .law files into the OmniScan instrument.

NOTE

Refer to the *NDT SetupBuilder User's Manual* or the *Advanced Calculator User's Manual* for information on how to use the application and how to create .law files.

To import a .law file into the OmniScan Instrument

1. Copy the .law file from your computer to the OmniScan instrument as follows:
 - a) Remove the OmniScan instrument storage card.
 - b) Connect the storage card to your computer.
 - c) Copy the .law file to the \User\Law\ folder on the storage card.
 - d) Disconnect the storage card from the computer, and reconnect it to the OmniScan instrument.
2. To include the correct probe and wedge names in the OmniScan report, select the probe and wedge models before the .law file import:
 - a) Select **Wizard > Setup > Start**, and then proceed to the **Select Probe and Wedge** step.
 - b) In the **Probe** list, select the appropriate probe.
 - c) In the **Wedge** list, select the appropriate wedge.
 - d) Skip the remaining Wizard steps until the final step, and then select **Generate** to complete the Wizard.
3. To use the .law file with the OmniScan instrument, proceed as follows:
 - a) Select **File > Setup > Load Law File**.
 - b) In the file browser that appears:
 - (1) Choose the .law file that you want from the list.
 - (2) Select **Open**.
4. Tap and hold the S-scan, and then choose the appropriate skew angle from the menu that appears.

7.12 Loading a Custom Color Palette

To load a custom color palette

1. Build a color palette file in .pal format (see “Creating a Color Palette” on page 190).
2. Transfer the .pal color palette file to an OmniScan-compatible storage device in the \User\Palette folder (if this folder does not exist, first create it).
3. Connect the storage device to the OmniScan instrument.

8. Menu Description

This chapter provides reference information for each menu, submenu, and parameter button available in the user interface of the OmniScan instrument. The structure of the sections follows the hierarchical structure of the menu, submenu, and parameter buttons found in the user interface of the software.

The content of this chapter is also available as contextual help on the OmniScan instrument.

TIP

“Online Help” on page 75 describes how to access the online help system of the OmniScan instrument.

8.1 File Menu

The **File** menu contains submenus that relate to the various types of data files.

8.1.1 Setup Submenu


The **Setup** submenu contains parameters for managing setup files (.ops), connectivity files (.ondtsetup), and law files (.law).

Open

Opens the file browser that lists the available setup files.
The files are saved in the \User\Setup folder on the storage device.

TIP

To directly access the **File > Setup > Open** parameter:

- OmniScan MX2: Press the File/Open key () to directly access the **File > Setup > Open** parameter (see Table 2 on page 17 for details).
- OmniScan MX2 and OmniScan SX: In the reading area, tap and hold the file name, and then select **Open** (in the contextual menu that opens) to directly access the **File > Setup > Open** parameter when in inspection mode (see Table 3 on page 20 for details).

Save As

Opens the file browser, enabling you to save the current setup under a different name in the \User\Setup folder on the storage device. The file browser has the following buttons:

File name

Used to enter a name for the file to be saved.

Save

Used to save the setup to the specified file name.

TIP

To directly access the **File > Setup > Save As...** parameter, tap and hold the file name in the reading area, and select **Save As...** (when in acquisition mode).

New

Used to create a new unnamed default setup file.

Import Connectivity

Opens the file browser that lists the available connectivity files. It is not possible to browse based on the file types. The files are saved in the \User\Setup folder on the storage device.

Load Law File

Opens the file browser that lists the available law files. It is not possible to browse based on the file types. The files are saved in the \User\Law folder on the storage device.

8.1.2 Report Submenu

The **Report** submenu contains parameters for managing reports. The reports are generated in HTML format, and can be printed directly from the OmniScan instrument or viewed and printed through a web browser on a computer.

Category

Used to select the category for the other parameters of the submenu. Selecting an item in this list changes the parameter buttons available to the right of the **Category** button. The following parameter categories are available:

- **Open/Save** (see “Open/Save Category” on page 231) to open an existing report or preview and save a new report.
- **Format** (see “Format Category” on page 232) to define the content of the report.
- **User Fields** (see “User Fields Category” on page 234) to define custom user fields to be displayed in the report.

8.1.2.1 Open/Save Category

Select **File > Report > Category = Open/Save** to view the following parameters:

Open

Opens the file browser that lists the available report files.

Preview

Used to display an HTML preview of the report on the display and offers the following parameter buttons:

Save and Close

Saves the report and closes the window.

Close

Closes the window without saving the report.


TIP

To directly access the **File > Report > Open/Save > Preview** parameter, tap and hold the file name in the reading area and select **Report Preview**.

Save

Used to save the HTML files of the report to the \User\Report folder on the storage device. The file name is defined using **File > Data Settings > File Name**.

TIP

When you select **File > Data Settings > Save Key = Report**, the Save/Print key () becomes a shortcut to the **Save** button, appearing when **File > Report > Category = Open/Save**.

8.1.2.2 Format Category

Select **File > Report > Category = Format** to view the following parameters, which enable you to configure the elements generated in the report:

Include

Used to select the type of image to be included in the report.

Table

Includes the image of the indication table in the report. For more information regarding the indication table, see “Configuring and Building an Indication Table” on page 169.

Current Layout

Includes a screen capture of the current layout in the report.

Off

No images are included.

Component

Used to select the sections to be included in the report. Choose the desired items.

User Field

Toggles the inclusion of the user-field section in the report. For more information on user fields, see “User Fields Category” on page 234.

Probe

Toggles the inclusion of the probe-characteristics field in the report.

Setup

Toggles the inclusion of the setup field in the report. This field includes ultrasonic settings, mechanical settings, and TCG and gate information in the report.

Note

Toggles the inclusion of the editable note field in the report.

Images

Includes all images associated with the entries from the indication table.

All

Includes all available sections in the report.

None

Excludes all available sections from the report.

Edit Header

Opens the text editor to edit the header of the report, a setup file, or a data file.

Edit Notes

Opens the text editor to edit a note attached to the report, data file, or setup file.

Template

Used to select a report template. The report type appears in parentheses after the report name.

Complete (Standard)

This is a template delivered with the OmniScan instrument. The report files are located on the storage device in the
\\App\MXU 4.n\Template\Report\Complete folder.

The **Report** folder also contains several other templates included with the instrument. These may be used to display relevant information, depending on the type of inspection being performed.

**CAUTION**

Do not modify or replace any files in the Complete folder located in \App\MXU 4.n\Template\Report. These are the original files used to create the default report. If you replace or modify them, you will no longer have the original files to create a custom template.

Custom report

This is a template that you make using custom files. These files are placed in a folder that you create on the OmniScan instrument storage device. See “Configuring a Report” on page 86 for the procedure for configuring a report.

8.1.2.3 User Fields Category

Select **File > Report > Category = User Field** to access the following parameters, which are used to define user-specified fields, including custom information to be included in the report.

Field

Used to select which user field (1 through 10) to edit.

Enable

Used to activate the currently selected user field.

Label

Used to enter a new label for the selected user field. For example, enter “Inspector.”

Content

Used to change the contents of the selected user field. For example, for the “Inspector” user field, enter “John Smith.”

TIP

To include the user fields in the report, select **File > Report > Category = Format**, and then **File > Report > Component = User field**.

8.1.3 Data Submenu

The **Data** submenu contains parameters for opening and saving inspection data files (.opd).


Open

Used to open the file browser that lists available inspection data files (.opd). The files are saved in the \User\Data folder on the storage device.

Save

Used in analysis mode to save the current inspection data in the \User\Data folder on the storage device. The file name format that is used is defined by selecting **File > Data Settings > File Name**.

TIP

When you select **File > Data Settings > Save Key = Data**, the Save/Print key () becomes a shortcut to **File > Data > Save**.

When in analysis mode, you can also tap and hold the file name and select **Save**, which is equivalent to **File > Data > Save**.

8.1.4 Image Submenu

The **Image** submenu contains parameters for creating and opening new and existing screen captures (.jpg).


Open

Used to open the file browser that lists the available image files (.jpg) in the \User\Screen folder on the storage device.

Save

Used to create a JPEG file (.jpg) for the current data display content. Saves the file in the \User\Screen folder on the selected storage device (defined by selecting **File > Data Settings > Storage**) using the file name format (defined by selecting **File > Data Settings > File Name**).

TIP

- When you select **File > Data Settings > Save Key = Image**, the Save/Print key () becomes a shortcut to **File > Image > Save**. You can also obtain this function by tapping and holding the file name to display the contextual menu, and then selecting **Save Image**.
 - When you plug a keyboard into the USB port, you can use the ALT+P shortcut to capture a screenshot of the full screen. You can also obtain this function by tapping and holding the file name to display the contextual menu, and then selecting **Print Screen**.
-

8.1.5 Data Settings Submenu

The **Data Settings** submenu contains configuration parameters applicable to all file types: setup (.ops), report (.html), data (.opd), and image (.jpg).

Data Selection

Used to specify the type of data recorded when **File > Data > Save** is selected.

The following choices are available:

All A & C-Scans

Saves the complete A-scan wave form and the C-scan data.

C-Scans Only

Saves only the C-scan data (amplitude and position simultaneously).

Storage

Used to select the storage device that you save your data files to.

Storage Card

Saves data, setups, and reports on the default storage device (SD card on the right-hand side of the OmniScan instrument).

Network (OmniScan MX2 only)

Stores data, setups, and reports on a network. The OmniScan MX2 must be connected to a network in order for this parameter to respond.

To connect the OmniScan MX2 to a network, see the following sections:

- “Connecting the OmniScan MX2 Directly to a Computer” on page 196
- “Connecting the OmniScan MX2 to a Network” on page 224

USB Storage

Saves data to an external storage device that you connect to one of the USB ports of the OmniScan instrument. This storage device is represented in the file system as \USB Storage. This item only appears when you connect an external USB storage device.

USB Storage 2

Saves data on a second external storage device that is connected to another USB port of the OmniScan instrument. This storage device is represented in the file system as \USB Storage. This item only appears when you connect a second external USB storage device.

File Name

Used to specify a file name format for all data file types (setup, report, data, and image). Build the file name using a combination of fixed characters and variables. For example: WeldABC###_%D_%T can become WeldABC001_2010-02-29_11.33.00. The following variables are available for building the file name:

#

Represents an automatically incremented number. (For example, data### becomes data001, data002, and so on.) The numbers are incremented separately in the respective folder for each data type.

%D


Represents the date in the international format (yyyy-mm-dd).

%T

Represents the time (hh.mm.ss).

Save Key

Used to specify which data type is saved on the storage device (defined by

selecting **File > Data Settings > Storage**) when you press the Save/Print key (). The following choices are available:

Report

Used to save the report as an HTML file in the \User\Report folder on the selected storage device.

Data (default value)

Used to save the inspection data to a file (.opd) in the \User\Data folder on the selected storage device.

Image

Used to save the screen content as a JPEG file (.jpg) in the \User\Screen folder on the selected storage device.

Data & Image

Used to save the inspection data to a file (.opd) in the \User\Data folder and save the screen content as a JPEG file (.jpg) in the \User\Screen folder on the selected storage device.

NOTE

The selection for the **File > Data Settings > Save Key** parameter is also effective when you save data remotely using the digital input (with **Preferences > Setup > Category = DIN** and **Preferences > Setup > Assign DIN = Save Data**).

8.2 Wizard Menu

The **Wizard** menu consists of submenus that offer step-by-step instructions on configuring specific aspects of your setup. As shown in the example in Figure 8-1 on page 239, each step of a wizard includes online help information and contains a title, navigation buttons, and associated parameter buttons. Using the wizards is the best way to easily and quickly create your setup.

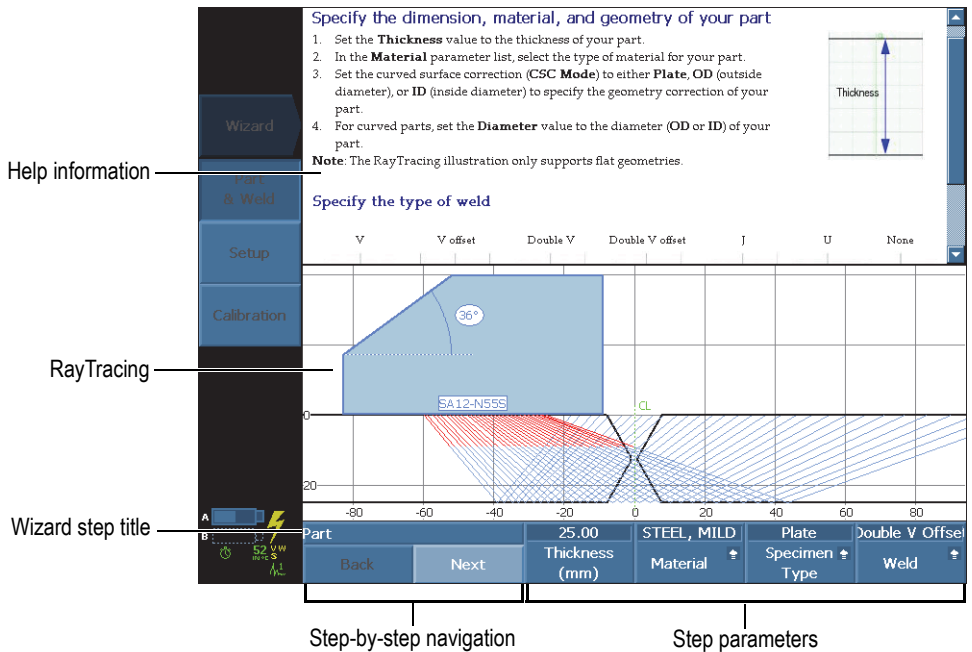


Figure 8-1 Wizard step components

Use the following generic parameter buttons to navigate through the wizard steps:

Start

Starts the first step of the wizard.

Next

Moves to the following step of the wizard.

Back

Returns to the previous step of the wizard.

Restart

Restarts the wizard if, at the end, the result is not within the acceptable tolerances.

Continue

Exits the current wizard and starts the next wizard.

TIP

You can exit a wizard at any time by pressing the Cancel key ().

TIP

When the RayTracing option is enabled, it can hide a portion of the wizard help. Press the Help key to hide the RayTracing view. Press the Help key again to display the RayTracing view.

NOTE

The wizard step control buttons are disabled while the system is in analysis mode. Select **Scan > Start > Pause = Off** to enable the wizard.

8.2.1 Part & Weld Submenu

The **Part & Weld** submenu wizard guides you through the following required steps:

- Define the material and the geometry of the inspected part.
- Set the characteristics of the weld.

8.2.2 Setup Submenu

The **Setup** submenu wizard guides you through the steps required to manage group(s) and inspection techniques.

An inspection setup is a set of parameter values that defines all aspects of an inspection leading to the production of a measurement.

Use the **Setup** wizard to define your probe, your wedge, and their positions relative to the part.

The parameters determine the wedge specifications (if one is used) and the probe specifications. The parameters also determine the connection of the wedge and probe to the instrument, as well as the scan position specifications.

The **Setup** wizard also guides you through the steps required to configure the focal laws.

To add a new group (OmniScan MX2 only)

1. Select **Wizard > Setup > Start** to start the **Setup** wizard.
2. Select **Operation = Add** (**Select Operation** step), and then select **Next**.
3. Follow the on-screen help information to complete the remaining steps of the wizard.

To modify an existing group (OmniScan MX2 only)

1. Select **Wizard > Setup > Start** to start the **Setup** wizard.
2. Select **Operation = Modify** (**Select Operation** step), modify the group, and then select **Next**.
3. Follow the on-screen help information to complete the remaining steps of the wizard.

To delete an existing group (OmniScan MX2 only)

1. Select **Wizard > Setup > Start** to start the **Setup** wizard.
2. Select **Operation = Delete** (**Select Operation** step), and then select **Next**.
3. Follow the on-screen help information to complete the remaining steps of the wizard.

To select an inspection technique

1. Select **Wizard > Setup > Start** to start the **Setup** wizard.
2. Select **Application**, and then choose the required inspection technique (**TOFD**, **Angle Beam**, **Zero Degrees**, or **Custom**).


Continue with the **Focal Law** portion of the wizard to do the following:

- Configure the type of law (**Sectorial**, **Compound**, **Linear**, or **Linear at 0°**)
- Set the wave type (**Longitudinal** or **Shear**)

- Select the probe elements to be used
- Set the beam angle
- Set the focus depth, skew angle, index offset, and scan offset of the probe

If you have more than one group, the focal laws will be applied to the selected group.

TIP

You can exit a wizard at any time by pressing the Cancel key().

8.2.3 Calibration Submenu

The **Calibration** submenu has a wizard that guides you through the necessary steps to perform various calibrations.

You need a calibration block to perform the various calibrations.

Select Calibration Step

The first step of the calibration wizard is to select what item you want to calibrate. The following parameters appear in the **Select Calibration** step:

Type

Used to specify the type of calibration to perform with the **Calibration** wizard. The following choices are available:

Encoder

To select an encoder to calibrate (available when configured to use an encoder [**Scan > Inspection > Scan**]).

Ultrasound

To select one of the ultrasonic aspects to calibrate with the **Mode** parameter.

Sizing

To select one of the sizing functions to calibrate with the **Mode** parameter.

Mode

Used to specify the calibration mode to perform with the **Calibration** wizard.

If you select **Type = Ultrasound**, the following choices are available:

Sensitivity (PA group only)

Used to calibrate the sensitivity to be able to detect a reference reflector.

Velocity

Used to calibrate the velocity of the sound propagation in the material of the inspected part. You must perform the velocity calibration first, because it is needed to perform the wedge delay calibration.

Wedge Delay

Used to calibrate the delay of the sound propagation within the wedge.

Velocity & WD (UT group only)

Used to calibrate, with one wizard, both the velocity of the sound propagation in the material of the inspected part and the delay of the sound propagation within the wedge.

WD & PCS (TOFD group only)

Used to calibrate, with one wizard, both the delay of the sound propagation within the wedge and the exit point-to-exit point distance between the two probes.

Lateral Synchro (analysis mode)

Lateral wave synchronization is used in TOFD to straighten the lateral wave signals on the scan axis.

If you selected **Type = Sizing**, you need to select which type of sizing function you want to use and calibrate. The following choices are available:

DAC

Used to select and calibrate the distance-amplitude correction (DAC) sizing curve.

TCG

Used to select and calibrate the time-corrected gain (TCG) sizing curve.

DGS

Used to select and calibrate the distance gain sizing (DGS) curve.

AWS

Used to select and perform the AWS (American Welding Society) code (D1.1 or D1.5) calibration. When a PA group is selected, **AWS** is applicable and available only for a sectorial scan setup.

Apply To (PA group only)

Used to specify the scope for the **Wedge Delay** and **Sensitivity** calibration wizard. The following choices are available:

All Angles

The calibration applies to all angles of the sectorial scan (available when in sectorial mode).

All VPA

The calibration applies to all VPAs of the linear scan (available when in linear mode).

2 or 3 Angles

The calibration only applies to 2 or 3 selectable angles of the sectorial scan.

2 or 3 VPA

The calibration only applies to 2 or 3 selectable VPAs of the linear scan.

Clear Calib. (sensitivity calibration)

Resets the sensitivity calibration. This parameter is only available with sensitivity calibration.

Clear Calib. (wedge delay calibration)

Resets the wedge delay calibration. This parameter is only available with wedge delay calibration.

Reset All

Resets the sizing function definition (DAC or TCG).

8.3 UT Settings Menu

The **UT Settings** menu contains the OmniScan instrument ultrasound settings.

8.3.1 General Submenu

The **General** submenu contains the main ultrasonic parameters. These parameters apply to the current group.

Gain

Used to set the signal gain value. The gain value always appears at the top of the screen (see Figure 8-2 on page 245).

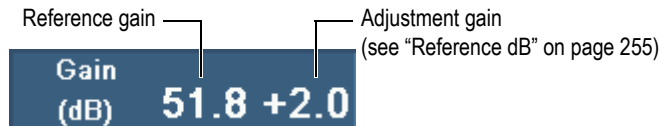


Figure 8-2 Gain value display

Start

Used to set a delay before the acquisition of data on the ultrasound axis.

Range

Used to set the range value.

TIP

- Tap the **Gain** section of the reading-fields area to set the signal gain directly.
- Tap the beginning of the UT axis to directly access the **Start** parameter.
- Tap the end of the UT axis to directly access the **Range** parameter.
- See Table 2 on page 17 for details.

Wedge Delay

When a PA group is selected, this parameter is used to increase the beam delay for all focal laws in the group. When a UT group is selected, it is used to manually adjust the wedge delay to set up a zero offset.

The wedge delay represents the total time of flight in the wedge. For example:

- Pulse-echo (P/E) time of flight, back and forth
- Pitch-catch (P/C) time of flight in pulser wedge, plus time of flight in receiver wedge

Velocity

Used to set the ultrasonic velocity inside a material. A default value is attributed according to the **Material** and **Wave Type** selections in the **Part & Weld** and **Setup** wizards; however, this value can be overridden.

To determine which material is currently selected, refer to the **Part & Weld** wizard.

UT Mode

Used to select the level of correction on the ultrasonic data displayed in the views. Available choices are **True Depth**, **Sound Path**, and **Uncorrected**.

8.3.2 Pulser Submenu

The **Pulser** submenu contains the various pulser parameters:

Pulser (OmniScan MX2 only)

Used to select the pulser for the selected group. The number of available pulsers depends on your probe and on your hardware configuration. Typical values are 1 for a one-probe inspection, and 1 or 65 for a two-probe inspection (using a Y splitter).

Tx/Rx Mode (UT group only)

Used to select the transmitter and receiver modes:

PE

Pulse-echo mode uses the same probe for emitting and receiving.

PC

Pitch-catch (transmit-receive) mode uses one probe for emitting and another probe for receiving.

TT

Through-transmission mode uses one probe for emitting from one side of the part, and another probe for receiving on the other side of the part.

Freq.

Used to set the probe-frequency value. This parameter is only available when the probe type is set to **Unknown**. When a PA group is selected, you also need to select **Auto Detect = Off** in the **Select Probe and Wedge** step of the **Setup** wizard. When the probe type is known, the instrument uses, by default, the frequency of the selected probe.

Energy

Used to set the output voltage for each pulser. Voltage is the same for all groups.

PW

Used to select the pulse-width (PW) value. The pulse width is automatically adjusted according to the probe frequency. You can also change the pulse-width value manually.

Acquisition rate

Used to set the value of the acquisition rate (**Acq Rate**). The **Acq Rate** value is directly linked to the **Scan Speed (Scan > Inspection > Scan = Time)** or to the **Max. Scan Speed (Scan > Inspection > Scan = Encoder)** parameters. The **Max. Scan Speed** appears when the **Scan** parameter is set to **Encoder**. The maximum scan speed represents the maximum encoder speed. The **Max. Scan Speed** parameter may be higher than the actual encoder speed. However, the encoder speed cannot be higher than the **Max. Scan Speed**. If the encoder speed is higher than the **Max. Scan Speed**, certain data could be missed. With encoders, the **Acq Rate** features an energy saving mode, whereby the **Acq Rate** is lowered when the encoder is not moving. Enter a value, which will be the requested value. The software will use this value as the target to be reached. You can also choose one of the following presets:

Auto Max.

Uses the maximum available **Acq Rate** value.

Optimum

Calculates the recommended value for the current configuration (default value). You can enter a value manually.

For more details about the acquisition rate and PRF, see “Acquisition Rate and PRF” on page 247, “Acquisition Rate and PRF for a Multiple Group Configuration (OmniScan MX2 Only)” on page 248, “Acq Rate and PRF Indicator” on page 249, and “Maximizing the PRF value” on page 250.

Acquisition Rate and PRF

The pulse repetition frequency (PRF) is the frequency at which pulses are emitted, whereas the acquisition rate (Acq Rate) is the frequency at which all pulses (the total number of pulses) are emitted. The PRF and Acq Rate are based on the inverse of the time interval between the emission of pulses. Acq Rate is the inverse of T_{Total} and PRF is the inverse of T_{Beam} , which are expressed in the following calculations (illustrated in Figure 8-3 on page 248):

$$\text{Acq Rate} = 1/T_{\text{Total}}$$

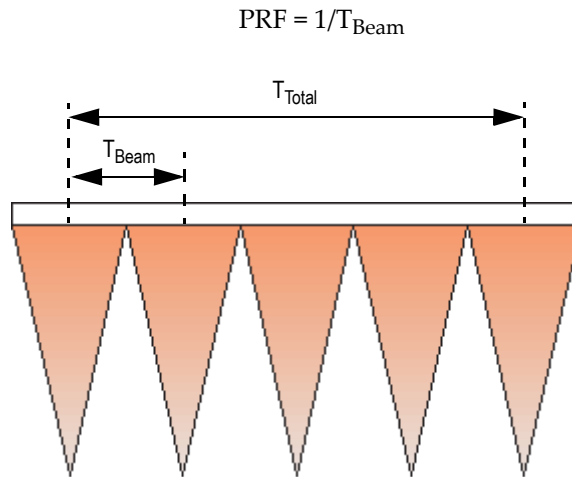


Figure 8-3 The time intervals for the Acq Rate and PRF calculations (PA group)

Acquisition Rate and PRF for a Multiple Group Configuration (OmniScan MX2 Only)

In the case of a multiple group configuration, the acquisition rate takes into account the emission of pulses for all groups. As illustrated in Figure 8-4 on page 248, the acquisition rate for a multiple group configuration is calculated as follows:

$$\text{Acq Rate} = 1 / (T_{\text{Group1}} + T_{\text{Group2}} + \dots + T_{\text{Groupn}})$$

$$\text{PRF}_{\text{Beam}} = 1 / T_{\text{Beam}}$$

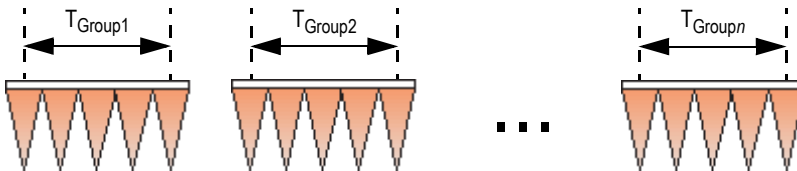


Figure 8-4 The time intervals for Acq Rate in a multiple group configuration

Acq Rate and PRF Indicator

The **Acq Rate** indicator in the reading field section of the user interface displays the Acq Rate—expressed as a frequency [Hz] or speed [distance/s]—followed by the PRF value in parentheses (see Figure 8-5 on page 249). The syntax of the indicator is as follows:

Acq Rate: X (Y)

where:

X = Acquisition rate (Acq Rate), expressed in speed units (mm/s or in./s) or frequency units (Hz)

Y = Pulse repetition frequency (PRF), expressed in Hz



Acq Rate : 60 Hz (1860 Hz)

Figure 8-5 The Acq Rate indicator in the reading field section

For example, if there are 31 beams (focal laws) in the scan, the **Acq Rate** indicator values can be described as follows:

- **Acq Rate: 1** means that 31 beams are scanned one time per second.
- **Acq Rate: 60** means that 31 beams are scanned 60 times per second.

The inspection speed is calculated as follows:

$$V = \text{PRF} \times \text{Scan resolution}$$

The scan resolution can be set in **Scan > Area > Scan Res.**

The characters of the Acq Rate indicator can appear in three colors, as described in Table 20 on page 249. See “Maximizing the PRF value” on page 250 for information on how to prevent PRF warning or error conditions.

Table 20 Color of the Acq Rate indicator

Color	Meaning
Green	Normal condition

Table 20 Color of the Acq Rate indicator (continued)

Color	Meaning
Yellow	Warning condition Not all signals are shown on the screen.
Red	Error condition Data loss and alarms can be missed.

Maximizing the PRF value

The parameters used to maximize the PRF value are listed below. These parameters may be combined in different ways; however, depending on the configuration, some of these parameters may not maximize the PRF value, because its limit may depend on other factors.

- Decrease the number of beams for one or more groups (under the **Focal Law** menu or **Focal Law** wizard).
- Decrease the **Range** value under **UT Settings > General**.
- Decrease the **Point Qty.** value under **UT Settings > Advanced**.
- Decrease the **Averaging** value under **UT Settings > Receiver**.
- Select **File > Data Settings > Data Selection = C-scans Only** rather than **All A & C-scans** to avoid PRF value reduction (and so maximize the PRF value). Saving all A-scans reduces the maximum PRF value to 6000.
- Reduce the number of groups (selecting **Group/Probe & Part > Group Management > Remove Last Group**). The PRF maximum value is divided by the number of active groups, regardless of whether or not they are displayed.
- To increase the maximum scan speed, you can also increase the scan resolution (**Scan > Area > Scan Res.**).

Preventing Abnormal PRF Conditions

If many specific features are enabled simultaneously, the OmniScan instrument might not be able to produce the requested Acq Rate value. If this occurs, the Acq Rate indicator value turns yellow (warning condition) or red (error condition) [see Table 20 on page 249 for details].

To prevent the occurrence of Acq Rate warnings or error conditions, consider the following suggestions:

- Decrease the value of the **UT Settings > Pulser > PRF** parameter.
- Do not select **Display > Overlay > A-scan = Envelope**.
- Do not use a layout that includes a B-scan (**Display > Selection > Layout**).
- Reduce the ultrasonic range with the **UT Settings > General > Start and Range** parameters.
- Reduce the value of the **UT Settings > Pulser > PW** parameter.
- Do not select the analog outputs using **Gates/Alarms > Analog > State = OFF**. The analog output for thickness (**Gates/Alarms > Analog > Data = Thickness**) calculated as a subtraction of variables (for example: **Gates/Alarms > Thickness > Source = A[^] - I[^]**) has a greater impact on the PRF value.
- Reduce the value of the **UT Settings > Advanced > Points Qty.** parameter. Reducing the A-scan points quantity helps maximize the PRF value.
- If the PRF abnormal condition persists, select **File > Setup > New** to create a new blank setup, and then rebuild your setup.

8.3.3 Receiver Submenu

The **Receiver** submenu is used to set the waveform of the A-scan view, in addition to the different filters available.

Receiver (OmniScan MX2 only)

A read-only parameter for viewing the selected receiver for a given channel.

- If you use the UT connector, the receiver will be P1 in P/E mode, and R1 in P/C mode. These parameters cannot be changed.
- However, when selecting a UT group on the PA connector, you can select one of the connector elements as a receiver. To use this feature, you should connect an adaptor on the PA connector and select a pitch-catch configuration (**UT Settings > Pulser > TX/RX mode = PC, TT, or TOFD**).

Filter

Used to select the appropriate filter value.

Rectifier

Used to select the rectification mode.

RF (radio frequency)

There is no rectification (see Figure 8-6 on page 252). Only available when **250 % mode** is turned off (see “250 % Mode Category” on page 328).

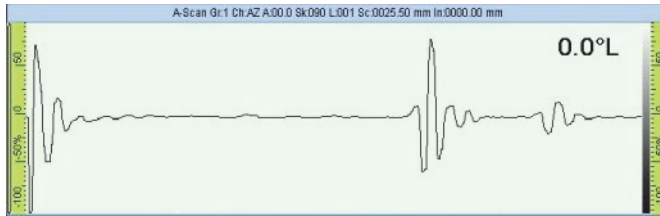


Figure 8-6 RF mode

HW+ (half wave positive)

Only the positive values are rectified (see Figure 8-7 on page 252).

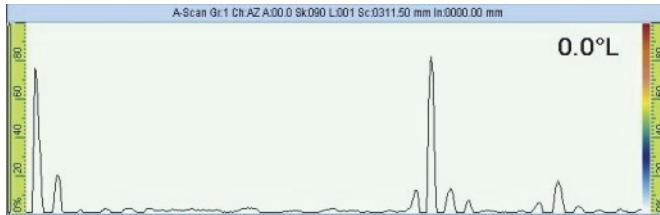


Figure 8-7 HW+ rectification

HW- (half wave negative)

Only the negative values are rectified (see Figure 8-8 on page 252).

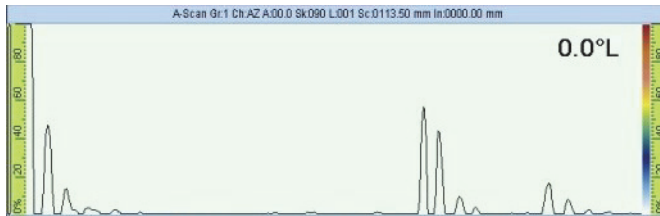


Figure 8-8 HW- rectification

FW (full wave)

The rectification values are all changed into absolute values (see Figure 8-9 on page 253).

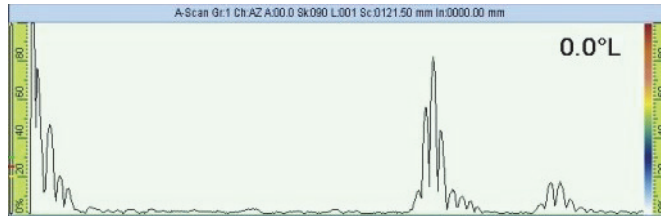


Figure 8-9 FW rectification

Video Filter

When activated, this parameter enables the video-smoothing filter. It is set according to the probe frequency and the rectification mode. The video filter is always available, except in RF mode.

Averaging

Used to select an averaging value (1, 2, 4, 8, 16, 32, or 64) for the current group. The averaging value divides the PRF value. For example, changing the averaging value from 1 to 4 causes an original PRF value of 1 kHz to drop to 250 Hz. The hardware still pulses at 1 kHz, but echo signals from all four pulses are averaged to produce a unique signal. Averaging is useful for reducing the noise on the echo signals. An averaging value of 1 corresponds to no averaging.

Reject

The signal amplitude inferior to the specified value is forced to 0 %. The default value is set to 0 %. It is typically used with a UT group only. It can cause excessive PRF reduction with PA groups.

8.3.4 Beam Submenu

The **Beam** submenu contains the parameters related to the beam.

Scan Offset

The **Scan Offset** is the difference between the 0 position marked on the part for inspection, and the real start position for the probe center on the scan axis (see Figure 8-10 on page 254).

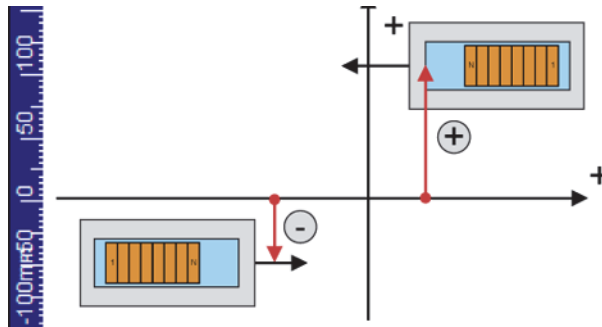


Figure 8-10 The scan offset

Index Offset

The beam **Index Offset** (see Figure 8-11 on page 254) is the difference between the 0 position marked on the part for inspection, and the real start position for the probe front edge on the index axis. The beam index offset is negative for a probe in the Skew 90 position, and positive for a probe in the Skew 270 position.

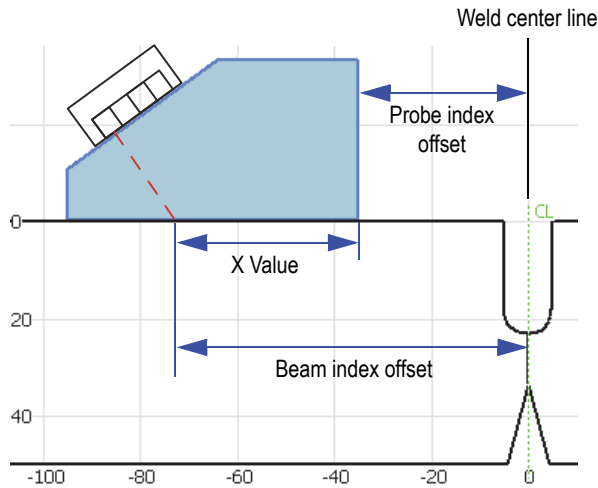


Figure 8-11 The beam index offset in a weld inspection example

Refracted Angle

Ultrasonic beam angle in the material. It only displays information for PA groups. It can be manually adjusted for a UT group if no wedge is selected.

Skew

Orientation of the ultrasonic beam relative to the scan axis. Skew 90 and skew 270 are typically used to define a two-sided two-probe inspection.

Beam Delay (PA group only)

Used to set the wedge delay for the selected focal law. Use the wedge delay calibration wizard to calculate the beam delay value for all beams. This parameter should only be used if you need to fine-tune the beam delay for the current focal law.

Gain Offset (PA group only)

Used to read the calculated gain offset applied to the current focal law. Values are typically created through the sensitivity calibration wizard, and they can be adjusted manually if needed.

8.3.5 Advanced Submenu

The **Advanced** submenu is used to set a reference signal.

Ref. Amplitude

Used to specify the A-scan full-screen height of the reference amplitude. The value is expressed as a percentage of the A-scan full-screen height. The default value is 80.0 %. The value modifies the **UT Settings > Advanced > Set to XX.X%** parameter. For example, when you set **Ref. Amplitude = 75%**, the other parameter changes to **Set to 75.0%**.

Set to XX.X %

Adjusts the gain of the currently selected group so that the peak amplitude of the signal in gate A reaches the full-screen height (FSH) value (XX.X %), as specified in the **Ref. Amplitude** parameter. This parameter can also be accessed by tapping and holding (or right-clicking) the **Gain** value field.

Reference dB

When turned on, this function freezes the current gain as the reference gain and adds an adjustment gain value (initially 0.0) to the **Gain** value field (see Figure 8-12 on page 256). The gain applied (to all focal laws in PA) is the total of the reference gain and the adjustment gain. The **Reference dB** parameter is useful

for inspections requiring the establishment of a reference gain, and the addition or subtraction of an adjustment gain.

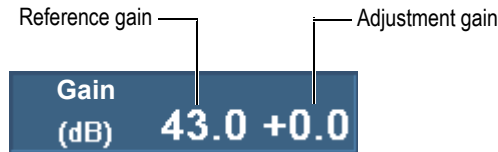


Figure 8-12 The Gain value field with an active reference

When turned off, this function removes the reference set, eliminating the adjustment gain from the **Gain** value field.

Points Qty.

Used to set the number of A-scan points to be stored. This parameter can also be set to the automatic mode (**Auto**). The parameter automatically adjusts the point quantity and compression factor according to the inspection range in order to ensure the number of points remains between 320 and 640. The inspection range is determined in **UT Settings > General > Range**.

The number of points in the A-scan, and the scale factor or compression, are directly related to file size.

Scale Factor

Displays the compression factor of the current group. A scale factor of 10 means that the most relevant point for every 10 points is produced by the acquisition hardware. The scale factor is directly related to the A-scan range and point quantity.

NOTE

Compression is not available on a TOFD channel, and is automatically set to 1.

Sum Gain

Used to set a gain on the sum of all channels, and automatically calculated by the group wizard based on the number of elements in the focal law. Does not typically require adjustment. Available for a PA group, or for a UT group when using the OmniScan MX2 PA connector.

NOTE

When you change the sum gain manually, you can lose the vertical linearity of the device. This parameter should only be changed by advanced users.

Attenuation

Turns the 20 dB signal attenuation on or off. Available only when a UT group using a UT connector is selected with a first-generation PA module.

8.4 Measurements Menu

The **Measurements** menu provides measurement tools and statistical parameters.

8.4.1 Cursors Submenu

The **Cursors** submenu contains cursor selection and positioning parameters. Cursors are reference lines that can be moved relative to axes. The Reference (red line) and Measurement (green line) cursors allow you to take precise measurements, either directly on the views, or by using cursor category readings (see “Cursor Category Readings” on page 268 for the list of available cursor category readings). The cursors need to be made visible by checking the **Cursor** box in the title bar shortcut menu.

Category

Used to select the view for the current layout in which you want to move the cursor positions. The choice of available views (**A-scan**, **B-scan**, **C-scan**, **D-scan**, **End View**, **S-scan**, or **Data**) depends on the current layout selected using **Display > Selection > Layout**. The parameters appearing to the right of **Category** apply to the selected view.

Moving Mode

Used to determine whether the Reference and Measurement cursors are moved individually (**Single**) or simultaneously (**Both**). This parameter affects the parameters in the **Measurements > Cursors** submenu and the pop-up button for the cursor parameter.

[Delta] <Axis>(<Cursor>)

The cursor parameters allow you to move one or both cursors on one axis. The available cursor parameters depend on the selected view. [Delta] is only available when you select **Cursor > Moving Mode = Both**.

The cursor parameter uses the [Delta] <Axis>(<Cursor>) naming convention where:

[Delta] is an optional prefix used only when <Cursor> = (r&m), which indicates that the parameter controls the distance between the two cursors.

<Axis> can be represented as follows:

%: representing the amplitude axis

U: representing the ultrasound axis

S: representing the scan axis

I: representing the index axis

Angle: representing the angle

VPA: representing the virtual probe aperture

<Cursor> can be represented as follows:

r: representing the Reference cursor

m: representing the Measurement cursor

r&m: representing the Reference and the Measurement cursors

For example, the U(r) parameter allows you to position the Reference cursor on the ultrasound axis.

Scan

Used to set the position of the Data cursor on the scan axis. Available when **Measurements > Cursors > Category = Data** is selected.

VPA or Angle

Used to set the current angle of the Data cursor. Available in **Measurements > Cursors > Category = Data**.

Index

Used to set the position of the Data cursor on the index axis. Available in **Measurements > Cursors > Category = Data** and when **Scan > Inspection > Type = Raster Scan**.

8.4.2 Reading Submenu

The **Reading** submenu provides parameters for selecting the values that are to appear in the four **s** at the top of the screen (see Figure 8-13 on page 259).

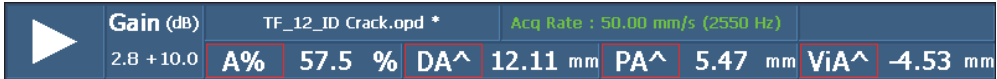


Figure 8-13 The four **s** at the top of the screen

Measurement Mode

Opens a dialog box which can be used to easily select the most useful readings for your application (see Figure 8-14 on page 260). The dialog box contains a list on the left of measurement modes associated with eight predefined readings that appear in a list on the right. Select one measurement mode, and then choose **Select** to display the eight associated readings at the top of the display (see Figure 8-13 on page 259). To select a measurement mode, you can also tap and hold the reading-field area, and then choose **Select Reading List** in the reading list shortcut menu.

Because only four readings can appear at the top of the display at one time, the eight readings are divided into two sets of four labeled **List 1/2** and **List 2/2**.

To select the reading list to be displayed

- ◆ Select **Measurement > Reading > Displayed Readings = List 1/2** or **List 2/2**.

OR

Tap or click on the readings.

All eight readings are generated in the report and saved with the setup file.

The **Custom** value appears when you customize the list of eight readings using the **Reading *n*** parameters.

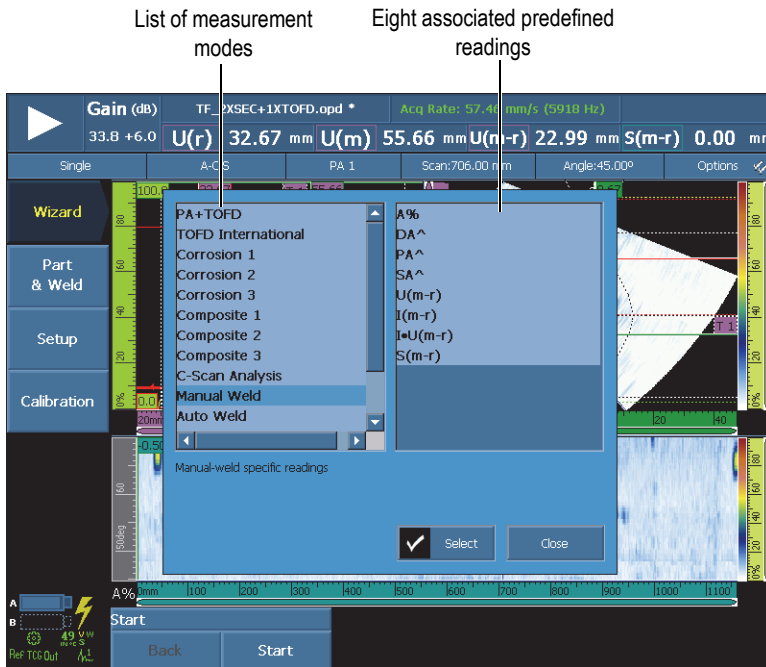


Figure 8-14 Dialog box for selecting measurement mode predefined readings

Displayed Readings

Used to select the list (**List 1/2** or **List 2/2**) of four readings you want to display at the top of the screen. The **Reading *n*** parameters on the right apply to the selected list.

Reading 1, 2, 3, or 4

Used to select and modify one reading field. Choosing a **Reading *n*** button opens the reading selection dialog box shown in Figure 8-15 on page 261. Choose the desired reading category from the list on the left-hand side of the dialog box, and then choose the desired reading from the list on the right-hand side of the dialog box.

TIP

To modify a particular reading field, you can also tap and hold the reading field to be modified, choose **Select Reading** in the readings shortcut menu, and then make your selection in the reading selection dialog box.

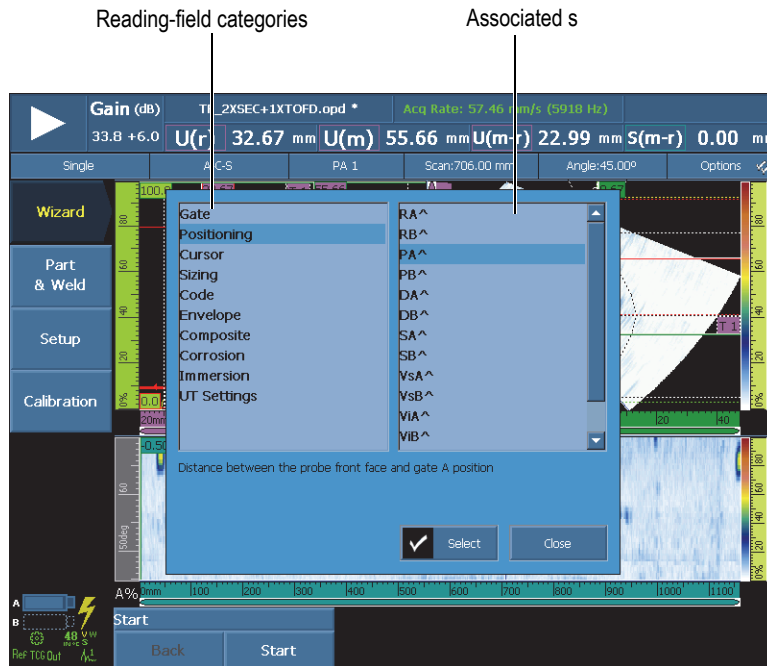


Figure 8-15 Reading field selection dialog box

NOTE

A colored contour around the reading code means that the reading field is displaying the information from the gate, the cursor, or the ultrasound unit corresponding to the color of the contour. See “Reading-Field Colors” on page 68 for details on outline colors.

NOTE

See the following sections for a description of available readings in each category.

8.4.2.1 Generic Reading Codes

Generic reading codes appear when abnormal conditions occur and no value can be displayed.

ND

No signal detected. This code appears when no signal has crossed the gate.

No data acquired. This code appears if any portion of the scan area was not covered during the inspection.

NS

No synchronization. This code appears when a gate is synchronized with another gate (or synchronization gate), but the synchronization cannot be established because no signal has crossed the synchronization gate.

8.4.2.2 Gate Category Reading

The following is the list of the **Gate** category reading codes and their descriptions.

A%

Peak amplitude of the signal detected in gate A (see Figure 8-16 on page 263).

B%

Peak amplitude of the signal detected in gate B.

I%

Peak amplitude of the signal detected in gate I.

AdBA

Amplitude (dB) of the signal detected in gate A minus the gate level.

The example presented in Figure 8-16 on page 263 shows an amplitude value for the peak signal crossing gate A at 49.9 % FSH (full-screen height). This value appears in the **A%** reading field on the left. The gate FSH is set to 25 %. The ratio

of the two FSH values is one half. Consequently, a difference of 6 dB appears in the **AdBA** reading field.

BdBB

Amplitude (dB) of the signal detected in gate B minus the gate level (see the **AdBA** definition).

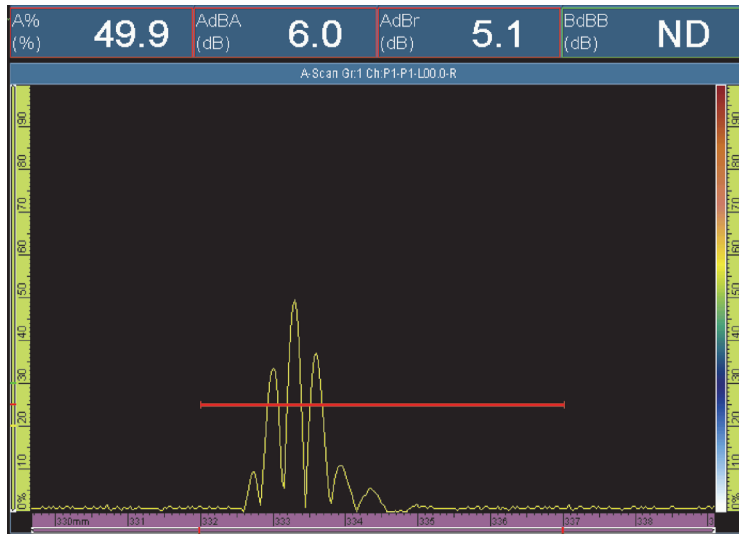


Figure 8-16 Example for the AdBA reading

AdBr

Difference between the current signal amplitude in dB (*right* in Figure 8-17 on page 264) and the reference signal amplitude (*left* in Figure 8-17 on page 264) detected in gate A. You can set the reference by selecting **UT Settings > Advanced > Reference = On**, and then selecting on **Set to XX.X%**.



Figure 8-17 Example for the AdBr reading

In Figure 8-17 on page 264 (*left*), the reference signal is at 100 % FSH. The reference signal value is indicated in the **A%** reading field. The **A%** reading field for the current signal (*right*) is 49.6. The signal loss is about a half, which represents a 6 dB drop, as indicated by the **-6.1** value in the **AdBr** reading field (*right*).

BdBr

Difference between the current signal amplitude (dB) and the reference signal amplitude detected in gate B (see the **AdBr** definition). The reference can be set by selecting **UT Settings > Advanced > Reference = On**, and then selecting on **Set to XX.X%**.

A^

Position of the signal peak value in gate A (see Figure 8-18 on page 265). The measurement taken depends on the gate mode that is selected (see “Gates Submenu” on page 295).

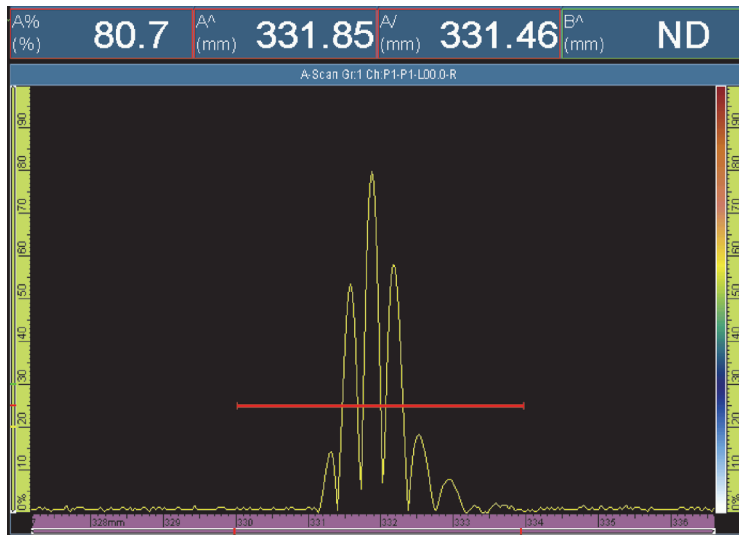


Figure 8-18 Example for the A%, A[^], and A/ readings

B[^]

Position of the signal peak value in gate B. The measurement taken depends on the gate mode that is selected (see “Gates Submenu” on page 295).

I/1

Position of the signal when it crosses gate I. The measurement taken depends on the gate mode that is selected (see “Gates Submenu” on page 295).

A%_r

Difference between the current signal amplitude (%) and the reference signal amplitude detected in gate A.

B%_r

Difference between the current signal amplitude (%) and the reference signal amplitude detected in gate B.

8.4.2.3 Positioning Category Reading

The following list outlines the **Positioning** category reading codes and their descriptions.

RA[^]

Distance on the part surface between the beam exit point and the indication detected in gate A (see Figure 8-20 on page 267).

RB[^]

Distance on the part surface between the beam exit point and the indication detected in gate B (see the **RA[^]** definition).

PA[^]

Distance on the part surface between the wedge (or probe) front face and the indication detected in gate A (see Figure 8-20 on page 267).

PB[^]

Distance on the part surface between the wedge (or probe) front face and the indication detected in gate B (see the **PA[^]** definition).

DA[^]

Depth in the part of the reflector producing the indication detected in gate A (see Figure 8-20 on page 267).

DB[^]

Depth in the part of the reflector producing the indication detected in gate B (see the **DA[^]** definition).

SA[^]

Sound path from the part entry point to the indication detected in gate A (see Figure 8-19 on page 266).

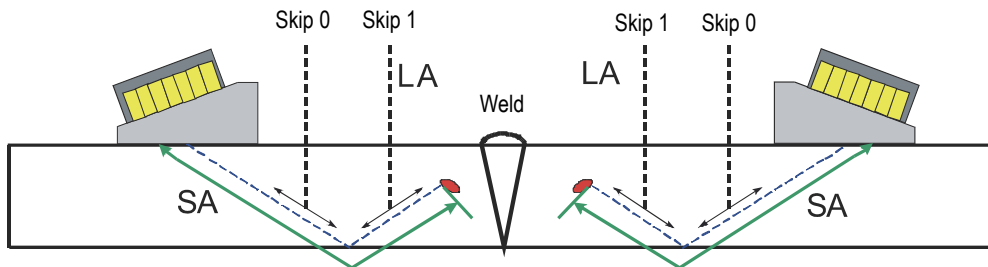


Figure 8-19 Diagram of the LA and SA readings

SB[^]

Sound path from the part entry point to the indication detected in gate B (see the **SA[^]** definition).

VsA[^]

Volumetric position of the indication detected in gate A in relation to the scan axis (see Figure 8-20 on page 267).

VsB[^]

Volumetric position of the indication detected in gate B in relation to the scan axis (see the **VsA[^]** definition).

ViA[^]

Volumetric position of the indication detected in gate A on the index axis (see Figure 8-20 on page 267).

ViB[^]

Volumetric position of the indication detected in gate B on the index axis.

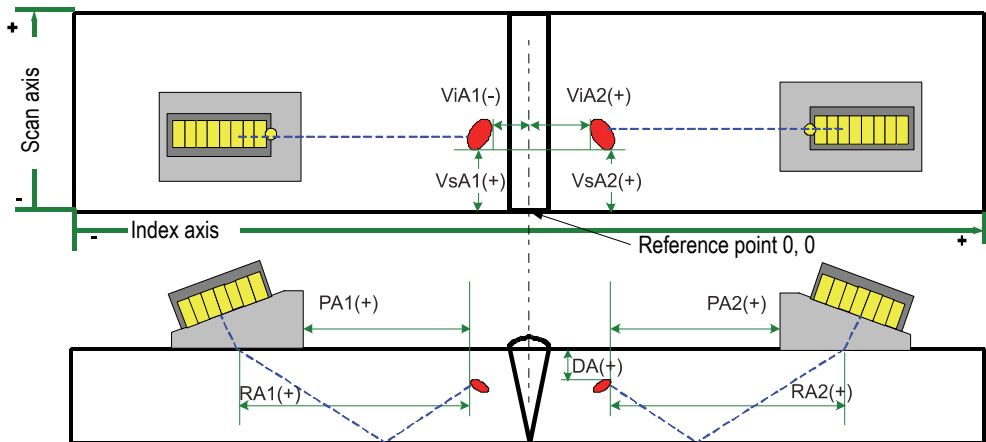


Figure 8-20 Diagram of the RA, PA, DA, ViA, and VsA readings

LA[^]

Number of skips from the beam exit point to the reflector indication detected in gate A. In Figure 8-19 on page 266, the **LA[^]** value is 1, because the reflector is seen after one skip.

LB[^]

Number of skips from the beam exit point to the reflector indication detected in gate B (see the **LA[^]** definition).

NOTE

When **Display > Overlay > Indicators = Legs** is checked, dotted lines appear on the views, spaced at intervals that correspond to the specified thickness value.

8.4.2.4 Cursor Category Readings

The following list outlines the **Cursor** category reading codes and their descriptions.

%(r)

Amplitude value at the Reference-cursor position (see Figure 8-21 on page 269).

%(m)

Amplitude value at the Measurement-cursor position (see Figure 8-21 on page 269).

%(m-r)

Amplitude value obtained by subtracting the amplitude of the Reference cursor from the amplitude of the Measurement cursor (see Figure 8-21 on page 269).



Figure 8-21 Example for the % (r), % (m), and % (m-r) readings

U(r)

Position of the Reference cursor on the ultrasound axis (see Figure 8-22 on page 270).

U(m)

Position of the Measurement cursor on the ultrasound axis (see Figure 8-22 on page 270).

U(m-r)

Distance along the ultrasound axis, obtained by subtracting the position of the Reference cursor from the position of the Measurement cursor (see Figure 8-22 on page 270).



Figure 8-22 Example for the U(r), U(m), and U(m-r) readings

P(r)

Distance between the Reference cursor and the probe front.

P(m)

Distance between the Measurement cursor and the probe front.

P(m-r)

Probe position obtained by subtracting the position of the Reference cursor from the position of the Measurement cursor.

S(r)

Position of the Reference cursor on the scan axis.

S(m)

Position of the Measurement cursor on the scan axis.

S(m-r)

Distance along the scan axis, obtained by subtracting the position of the Reference cursor from the position of the Measurement cursor.

I(r)

Position of the Reference cursor on the index axis.

I(m)

Position of the Measurement cursor on the index axis.

I(m-r)

Distance along the index axis, obtained by subtracting the position of the Reference cursor from the position of the Measurement cursor.

I•U(m-r)

Distance along the diagonal of the rectangle formed by the intersection of the Measurement and the Reference cursors.

%(U(m))

Signal amplitude at the Measurement cursor position on the ultrasound axis. This value is not computed in analysis mode (see the **U(m)** definition).

%(U(r))

Signal amplitude at the position of the Reference cursor on the ultrasound axis. This value is not computed in analysis mode (see Figure 8-23 on page 271).



Figure 8-23 Example for the %(U(r)) reading

TOFD(r)

Corresponding depth in the part along the ultrasound axis for the Reference cursor (calibrated TOFD group only).

TOFD(m)

Corresponding depth in the part along the ultrasound axis for the Measurement cursor (calibrated TOFD group only).

TOFD(m-r)

Corresponding depth in the part along the ultrasound axis, obtained by subtracting the depth of the Reference cursor from the Measurement cursor (calibrated TOFD group only).

D(r)

Corresponding depth in the part along the ultrasound axis for the Reference cursor.

D(m)

Corresponding depth in the part along the ultrasound axis for the Measurement cursor.

D(m-r)

Corresponding depth in the part along the ultrasound axis, obtained by subtracting the depth of the Reference cursor from the Measurement cursor.

S(m-r)CSC

Scan distance between the Reference and Measurement cursors, corrected for the part curvature and flaw depth.

TOF:U(r)

Position of the Reference cursor in microseconds (Halfpath), excluding the beam delay in the wedge.

TOF:U(m)

Position of the Measurement cursor in microseconds (Halfpath), excluding the beam delay in the wedge.

TOF:U(m-r)

Difference between the Measurement and cursor in microseconds (Halfpath), excluding the beam delay in the wedge.

8.4.2.5 Sizing Category Readings

The following is the list of the **Sizing** category reading codes and their descriptions (see Figure 8-24 on page 273).

A%Curve

Difference, in percentage, between the peak amplitude of the signal detected in gate A, and the corresponding amplitude of the selected sizing curve.

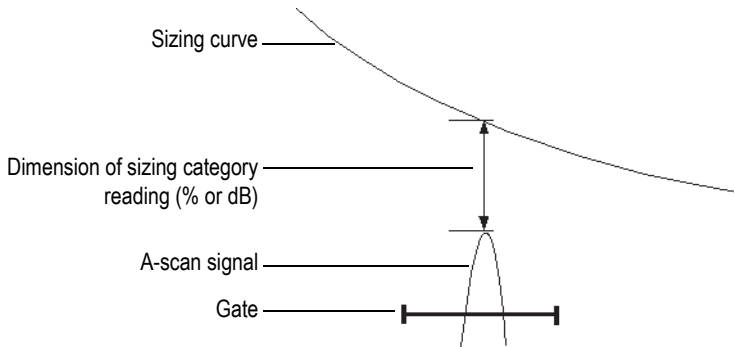


Figure 8-24 Illustration of the sizing category readings

AdBCurve

Difference, in dB, between the peak amplitude of the signal detected in gate A, and the corresponding amplitude of the selected sizing curve.

MaxA%Curve

Maximum difference, in percentage, between the peak amplitude of the signal detected in gate A, and the corresponding amplitude of the selected sizing curve from the last curve selection or the last data reset.

MaxAdBCurve

Maximum difference, in dB, between the peak amplitude of the signal detected in gate A, and the corresponding amplitude of the selected sizing curve from the last curve selection or the last data reset.

B%Curve

Difference, in percentage, between the peak amplitude of the signal detected in gate B and the corresponding amplitude of the selected sizing curve.

BdBCurve

Difference, in dB, between the peak amplitude of the signal detected in gate B and the corresponding amplitude of the selected sizing curve.

TIP

Press the Play key () to reset the memory of the peak values of the **MaxA%Curve**, **MaxAdBCurve**, **MaxB%Curve**, and **MaxBdBCurve** readings.

MaxB%Curve

Maximum difference, in percentage, between the peak amplitude of the signal detected in gate B and the corresponding amplitude of the selected sizing curve from the last curve selection or the last data reset.

MaxBdBCurve

Maximum difference, in dB, between the peak amplitude of the signal detected in gate B and the corresponding amplitude of the selected sizing curve from the last curve selection or the last data reset.

ERS

Equivalent reflector size, in mm, used with the DGS.

h/l Ratio

Aspect ratio of defect height to defect length as measured by the Reference and Measurement cursors.

h/t Ratio

Aspect ratio of the defect height to the defined part thickness as measured by the Reference and Measurement cursors.

8.4.2.6 Code Category Reading

The following is the list of the **Code** category reading codes and their descriptions.

AWS A

Gain needed to set the signal amplitude to the reference amplitude, as per the AWS (American Welding Society) code, for the signal in gate A.

AWS B

Zero reference indication level, as per the AWS code, for the signal in gate A.

AWS C

Attenuation factor, as per the AWS code, for the signal in gate A.

AWS D

Indication rating, as per the AWS code, for the signal in gate A.

AWS CL

Discontinuity severity class, as per the AWS code, for the signal in gate A.

8.4.2.7 Envelope Category Readings

The following is the list of the **Envelope** category reading codes and their descriptions.

E%

Peak amplitude of the envelope in gate A.

The envelope is set by selecting **Display > Overlay > A-Scan**, and then selecting **Envelope**.

TIP

To reset the envelope, tap and hold the A-scan view, and then tap **Clear Envelope** on the shortcut menu.

E^

Position of the envelope peak value in gate A (see Figure 8-25 on page 276). This code appears when **Gates/Alarms > Gates > Parameters = Mode and Measure = Peak**.

E/

Position of the envelope edge value in gate A. This code appears when **Gates/Alarms > Gates > Parameters = Mode and Measure = Edge**.

E-3dB

Width of the signal envelope at -3 dB for the signal detected in gate A.

E-6dB

Width of the signal envelope at -6 dB for the signal detected in gate A (see Figure 8-25 on page 276).



Figure 8-25 Example for the E%, E[^], and E-6dB readings

E-12dB

Width of the signal envelope at -12 dB for the signal detected in gate A.

E-20dB

Width of the signal envelope at -20 dB for the signal detected in gate A.

8.4.2.8 Corrosion and Composite Category Readings

The following is the list of the **Corrosion** and **Composite** category reading codes and their descriptions.

NOTE

The C-scan gate time and amplitude measurements are taken at a digitization sampling frequency of 100 MHz.

T

T is a dynamic reading used to measure thickness. Thickness can be measured using one gate, or by subtracting two gate values (see Figure 8-26 on page 278).

To calculate thickness, select **Gates/Alarms > Thickness > Source**. The reading appears as **T (A[^])**, **T (B/)**, or **T(B[^]-A/)**, etc., according to the selection made (see “Thickness Submenu” on page 304). The thickness value (T) is always measured in millimeters.

ML

Material loss, expressed as a percentage (%), is the result of the part thickness (configured in **Group/Probe & Part > Part > Thickness**) minus the value in the **T** reading field divided by the part thickness. The example shown in Figure 8-26 on page 278 illustrates the inspection result of a plate in which each letter of the word CORROSION is engraved progressively deeper.

Tmin

Tmin is the thinnest reading recorded during the current acquisition.

Tmin and its associated readings [**S(Tmin)**, **I(Tmin)** and **Angle(Tmin)**] are typically used for corrosion mapping and composite inspection applications, and they are designed to monitor and update the minimum thickness that is recorded during the current acquisition. The **Tmin** reading only displays readings within the thickness range defined in **Gate/Alarms > Thickness > Source, Min.** and **Max.** The **Tmin** reading is reset when a new acquisition is initiated.

S(Tmin)

S(Tmin) is the scan axis position of the **Tmin** reading.

I(Tmin)

I(Tmin) is the index position of the **Tmin** reading.

Angle(Tmin)

Angle(Tmin) is the relative focal law or virtual probe aperture (VPA) of the **Tmin** reading.

TminZ

TminZ is the thinnest reading recorded within the zone box created by the Reference and Measurement cursors on the thickness C-scan display.

TminZ and its associated readings [**S(TminZ)**, **I(TminZ)** and **Angle(TminZ)**] are typically used for corrosion mapping and composite inspection applications, and they are designed to display the minimum thickness within a rectangular zone created with the Reference and Measurement cursors on the thickness C-scan

display. The **TminZ** reading only displays readings within the thickness range defined in **Gate/Alarms > Thickness = Source, Min.** and **Max.**

S(TminZ)

S(TminZ) is the scan axis position of the **TminZ** reading.

I(TminZ)

I(TminZ) is the index axis position of the **TminZ** reading.

Angle(TminZ)

Angle(TminZ) is the relative focal law or virtual probe aperture (VPA) of the **TminZ** reading.

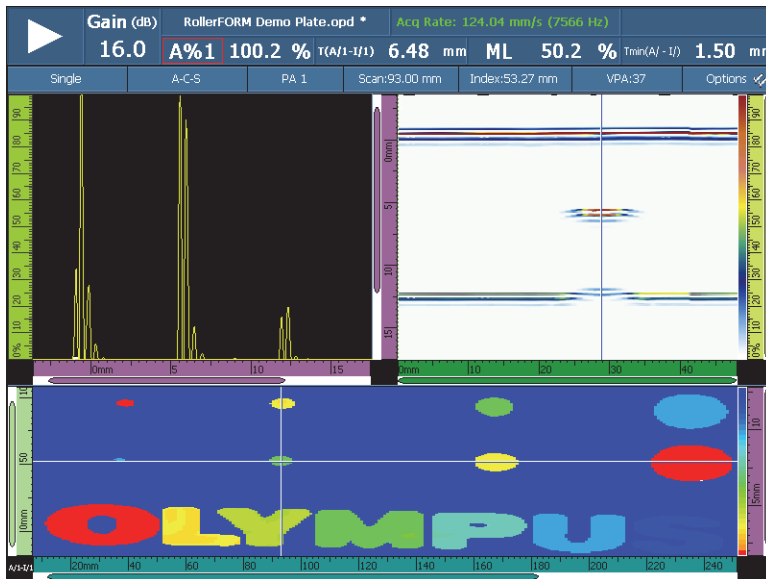


Figure 8-26 Example of corrosion inspection results, and T and ML readings

8.4.2.9 Immersion Category Readings

The following list outlines the **Immersion** category reading codes and their descriptions.

I/1

Position of the signal when crossing gate I.

I(w)/1

Position of the signal when crossing gate I using the velocity of water.

8.4.2.10 UT Settings Category Readings**Scale**

Scale factor, which is the signal compression factor with regard to the digitizing frequency.

8.4.3 Indication Table Submenu

The **Indication Table** submenu contains the parameters needed to configure the indication table. The indication table (see Figure 8-27 on page 279) presents detailed information related to the reflectors identified and recorded during the inspection, and it is used to create the inspection report.

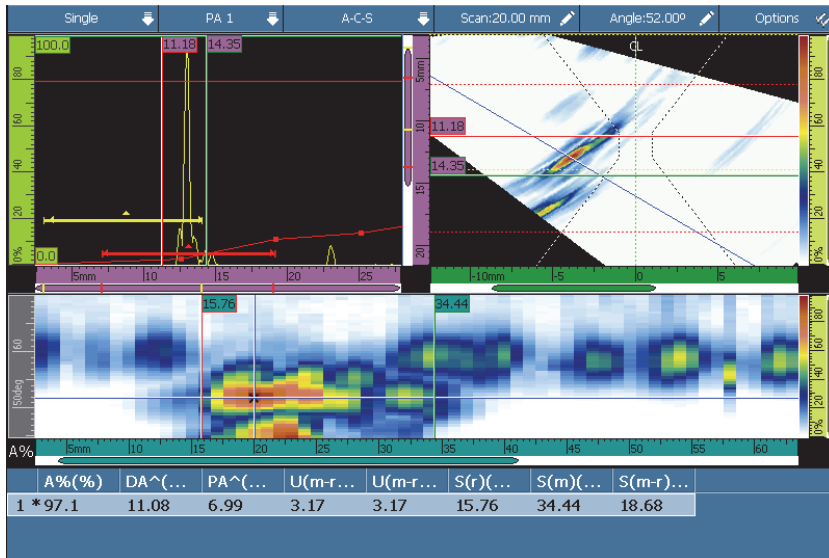


Figure 8-27 Example of an indication table

Select

Used to select a particular indication in the table. Displays the data related to the selected indication. In the RayTracing view, the corresponding dot turns red.

Add

Used to add the current data as an indication to the indication table.

Delete

Used to delete a selected indication from the indication table.

Comments

Used to add or edit a comment related to the currently selected indication.

TIP

Tap and hold the reading zone to quickly access the most-frequently used options for **Indication** table management.

8.5 Display Menu

The **Display** menu contains parameters related to A-scans (**A-Scan Settings** submenu) and graphical elements in views (**Overlay** submenu).

8.5.1 A-Scan Settings Submenu

The **A-Scan Settings** submenu contains multiple options for the A-scan view to help you display data more clearly.

Appearance

Used to select an appearance for the signal on the A-scan view.

Hollow

The area below the A-scan signal is not colored.

Fill

The area under the A-scan signal is filled with the A-scan signal color.

Hollow 2 Colors

The area under the A-scan signal is not colored. The A-scan signal crossing gate A appears in red.

Fill 2 Colors

The area below the A-scan signal is filled with the A-scan signal color. The part of the A-scan signal crossing gate A and the area above it is filled in with red.

Data Source

Used to choose which A-scan or combination of A-scans (the source data) to display on the A-scan view. When the PRF value is above the display refresh rate, not all acquired A-scans are displayed on the screen. This may cause you to miss an important signal. The following options enable you to choose how the A-scans appear on the display.

Normal

When a PA group is selected, displays the A-scan of the current focal law. When a UT group is selected, displays all A-scans when the PRF value is below the display refresh rate, and displays a sample of A-scans in all other cases.

Highest (%)

Displays the A-scan of the focal law with the highest signal in gate A (PA group only). In the S-scan, the Data cursor linked to the current law tracks the highest value.

Thinnest

Displays the A-scan of the focal law with the smallest thickness value (PA group only).

All Laws

Displays the A-scan built from the highest values of all focal laws (PA group only).

A% Max.

When the PRF value is above the display refresh rate, displays the A-scan built from the peak amplitude signal in gate A of all A-scans, including the ones that are not displayed (UT group only).

Min. Thickness

When the PRF value is above the display refresh rate, displays the A-scan built from the minimum thickness of all A-scans, including the ones that are not displayed (UT group only).

Max. Thickness

When the PRF value is above the display refresh rate, displays the A-scan built from the maximum thickness of all A-scans, including the ones that are not displayed (UT group only).

Extra Range

Used to enlarge the A-scan view by a specific percentage at the beginning and end of the ultrasound axis. When the grid is active (with **Display > Overlay > A-scan = Grid** selected), solid grid lines mark the beginning and end of the range specified by the **UT Settings > General > Start** and **Range** parameter settings. As shown in the example in Figure 8-28 on page 282, selecting **Extra Range = 5% - 5%** enlarges the A-scan view by 5 % at both ends of the range.

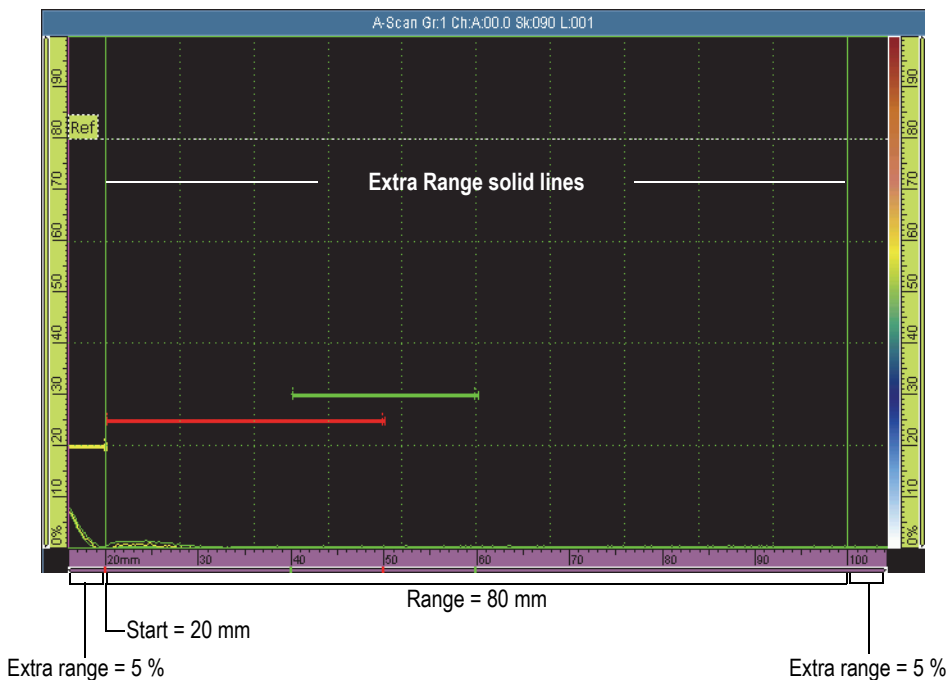


Figure 8-28 Example of an extra range set to "5% - 5%"

Grid

Used to select the color of the grid lines. Select **Display > Overlay > A-scan = Grid** to make the grid visible.

Ultrasound Cell Qty.

Used to set the number of grid cells for the ultrasound axis.

Amplitude Cell Qty.

Used to set the number of grid cells for the amplitude axis.

8.5.2 Overlay Submenu

The **Overlay** submenu contains parameters needed to display various graphical elements (such as the gates, cursors, A-scan elements, and indicators) in the views.

Cursor

Used to select which cursor-related overlay elements appear on the screen (see Figure 8-29 on page 284). Choose the desired items in the list.

Cursor

Displays the Reference (red lines), and the Measurement (green lines) cursors.

Values

With the cursors visible, displays the cursor-position values at the end of the cursor line.

A-Scan Readings

Displays the A-scan values at the cursor positions in the upper-right corner of the A-scan view.

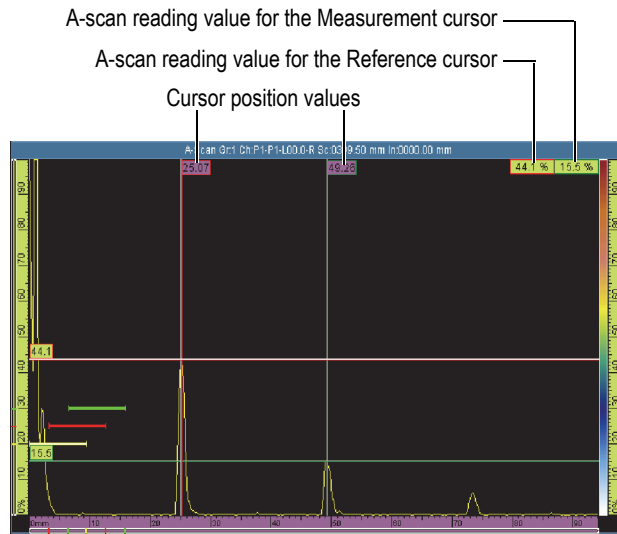


Figure 8-29 Cursor overlay example

A-Scan

Used to select which A-scan-related overlay elements appear on the screen. Choose the desired items in the list.

Envelope

Displays a curve built with the maximum and minimum amplitudes on the A-scan view (see Figure 8-30 on page 285).

TIP

To reset the envelope, tap and hold the A-scan view, and then tap **Clear Envelope** on the shortcut menu.

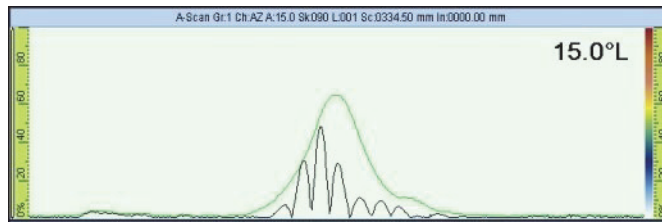


Figure 8-30 A-scan envelope curve

Sizing curves

Displays the sizing curve (DAC [distance-amplitude correction], linear DAC, TCG [time-corrected gain], or DGS [distance gain size]) that was configured using the **Sizing** menu.

TCG Gain Curve

Displays horizontal lines at an amplitude equal to the gain of each TCG curve.

Ref. Amplitude

Displays a horizontal dotted line on the A-scan at the height of the reference-amplitude value.

Grid

Displays grid lines on the A-scan view. Select **Display > Properties > Category = Grid Settings** to configure other aspects of the grid.

Indicators

Used to toggle the visibility of overlay elements in the views. An overlay element is visible when a check mark appears next to it in the list. The following elements are available:

Weld

Displays a drawing on the S-scan of the weld geometry, and the weld geometry reflection for each leg.

Legs

Displays dotted lines representing the limit of each leg. A leg is the straight-line path traveled by the shear wave before it is reflected by the opposite surface of the inspected part. The dotted lines for the back wall or bottom position are labeled B_n . The dotted lines for the entry surface or top positions are labeled T_n .

8.5.3 RayTracing Submenu

Legs

Used to set the number of visible legs in the R-scan display.

8.6 Sizing Menu

The **Sizing** menu provides the functions needed to choose, configure, measure, and build curves for various types of sizing functions.

As the sound beam travels through a part, the signal amplitude from reflectors of the same size (but at different distances from the transducer) varies owing to material attenuation and beam spread. Sizing functions allow you to evaluate the size of a reflector anywhere in the part by measuring or compensating for the signal attenuation. The available sizing functions are the **DAC** (distance-amplitude correction), **Linear DAC**, **TCG** (time-corrected gain), **DGS** (distance gain size), and **AWS** (American Welding Society).

The **Linear DAC** curve is useful and quick to set up. However, you need to know the attenuation of the material and understand that it does not perfectly represent reality. The point-by-point **DAC** curve is more precise because it is built using signals from known reflectors in a calibration block.

TIP

To select and configure a sizing function, select **Wizard > Calibration**. In the **Select Calibration** step, select **Type = Sizing**, and then select **Mode = DAC, TCG**, followed by **DGS**, or **AWS**. Choose **Start** and follow the wizard steps. See “Calibration Submenu” on page 242 for details.

Select **Display > Overlay > A-Scan = Sizing Curves** to show or hide the sizing curve lines on the display.

8.6.1 Type Submenu

The **Type** submenu allows you to select the type of sizing function for configuration. The available submenus depend on your selection from the **Type** submenu list. After you have completed configuration of the sizing function, the **Type** submenu list will also display the current active sizing function, and enable you to quickly switch between the various sizing functions.

The following types of sizing functions are available:

None

Deactivates the sizing function. Any existing sizing-function definition remains available so you can quickly reactivate it. No submenus are available with this selection.

DAC

A distance-amplitude correction (DAC) curve is used to plot amplitude variations of signals from reflectors of the same size, but at increasing distances from the transducer. These reflectors produce echoes of decreasing amplitude resulting from material attenuation and beam spread as the sound beam travels through the part. The purpose of the DAC curve is to graphically show the effect of material attenuation, near-field effects, beam spread, and surface roughness on the signal amplitude.

After a DAC curve is plotted, reflectors of the same size as those used to create the curve produce echoes that peak along the curve, despite being at different locations within the part. Similarly, reflector echoes smaller than those used to create the curve fall below the level, while larger reflector echoes exceed the curve level.

The DAC is built by using up to 32 signals from identically sized calibration reflectors (in UT or PA mode). Predefined DAC curve parameters compliant with the JIS (Japan Industrial Standards) and ASME (American Society of Mechanical Engineers) are available, as shown in Table 21 on page 290.

IMPORTANT

To display a DAC curve, you must have a reference gain. If you do not have one and try to activate a DAC curve, a reference gain will automatically be created. You will not be able to cancel the reference gain without first canceling the DAC curve.

Linear DAC

Displays a linear distance-amplitude correction (DAC) curve automatically calculated using the value for the material attenuation of the inspected part. Reflectors of the same size produce echoes that peak along the curve despite being at different locations within the part. Reference reflectors are not needed to define a linear DAC. Specify the material attenuation in the **Sizing > Curves Setup > Mat. Attenuation** parameter.

TCG

The time-corrected gain (TCG) function increases the gain applied to the signal as a function of the time the echoes take to return. The result is that the echo peaks (from same-sized reference flaws) appear at the same screen height, independently of their position in the part. The TCG uses the same factors as the DAC.

DGS

The distance gain size (DGS) function is used to size reflectors based on a calculated DGS curve for a given transducer, material, and reflector size. The main DGS curve represents the signal amplitude of an equivalent flat-bottom hole (FBH) reflector of a specific size. The DGS function only needs one reference reflector to create a DGS curve, as opposed to the several reference reflectors needed for the DAC and TCG functions. The DGS function displays a main curve and a warning curve.

AWS

The American Welding Society (AWS) sizing calibration function is compliant with AWS-D1.1/1.5 codes, and provides calibration for three angle beams on a reference reflector.

8.6.2 Operator Mode Submenu

The **Operator Mode** submenu includes parameters needed to select the active curve and specify gain values.

Active Curve (not applicable to **DGS** and **AWS**)

Used to select one of the defined sizing curves as the active curve. The active curve turns green, while the other curves turn white. You must make a curve active in order to change its configuration. This parameter is only accessible when more than one sizing curve is defined.

Next Curve (not applicable to **DGS** and **AWS**)

Used to select the next defined sizing curve as the active curve. You must make a curve active in order to configure it. This parameter is only accessible when more than one sizing curve is defined.

DAC Gain (**DAC** and **Linear DAC** only) and **TCG Gain** (**TCG** only)

Used to simultaneously modify the gain of the signal and the gain of the **DAC** or **TCG** curves. Use this function if you need to modify the gains in order to improve the sizing process.

Reference Gain (**DAC** and **Linear DAC** only)

Used to modify the reference gain exclusively affecting the signal, as opposed to the **DAC** gain.

Reg. Level (**DGS** only)

Reg. Level represents the height of the main **DGS** curve. The **DGS** curve represents the expected amplitude response (across a defined thickness range) from a flat-bottom hole whose diameter equals the registration level. The **Reg. Level** value usually equals the critical flaw size for the application, and it is limited to the effective diameter of the transducer being used.

Warn Curve (**DGS** only)

Used to select one of the warning curves.

Warning Level (**DGS** only)

The warning level is the secondary **DGS** curve positioned below the main **DGS** curve. The main curve appears in red, while the warning curve appears in white. If the warning level is set to **0**, the associated warning curve is turned off.

Delta Vt (**DGS** only)

Used to compensate for the attenuation of the coupling variation resulting from the surface condition of the calibration block and the inspected part. The ISO 16827 standard provides methods for calculating transfer corrections.

Specimen At. (**DGS** only)

This is the attenuation value in dB/mm for the test piece (specimen). In some cases, it is necessary to calculate the relative attenuation within the test piece and enter the value here.

8.6.3 Type Setup Submenu

The **Type Setup** submenu includes the parameters that normally only need to be configured once for a given inspection. The parameters available under the **Type Setup** submenu depend on the selection made for the **Sizing > Type** submenu list value.

8.6.3.1 Parameters for DAC, Linear DAC, and TCG Curves

The following parameters are available when you select **Sizing > Type = DAC, Linear DAC, or TCG**:

Compliance (not applicable to **Linear DAC**)

Used to select a customized sizing function type, or a preset sizing function type, for JIS or ASME code (see Table 21 on page 290 for details). When a preset is selected, specific sizing function parameters are automatically set, making them read-only.

Table 21 DAC curve properties relative to standards

Parameter	JIS	ASME
Curve 1 (main)	0 dB	0 dB
Curve 2	+24 dB	N/A
Curve 3	+18 dB	N/A
Curve 4	+6 dB	N/A
Curve 5	-6 dB	N/A
Curve 6	+12 dB	N/A
Curve 7	-12 dB	N/A
Curve type	Straight	Polynomial
X-axis grid divisions	10	10
Y-axis amplitude	110 % (Calibration only)	100 %
Grid overflow (see Extra Range)	0 % to 10 %	0 % to 10 %

Table 21 DAC curve properties relative to standards (continued)

Parameter	JIS	ASME
Delay before point of curve	Straight	N/A

The following types of sizing functions are available:

Custom

Allows point-by-point creation and modification of one or more sizing functions.

JIS

Used to define seven sizing curves that are compliant with the JIS (Japan Industrial Standards) code.

ASME

Used to define a single sizing curve that is compliant with the ASME (American Society of Mechanical Engineers) code.

Ref. Amplitude

Used to specify the A-scan full-screen height for the reference amplitude. The value is expressed as a percentage of the A-scan full-screen height. The default value is 80.0 %. The value modifies the **UT Settings > Advanced > Set to XX.X%** parameter. For example, when you set **Ref. Amplitude = 75%**, the other parameter changes to **Set to 75.0%**.

Curve Type (DAC only)

Used to specify the type of interpolation between the points of the curve. This parameter is read-only for sizing curves compliant with a standard. The following choices are available (see Figure 8-31 on page 292):

Straight

Linear interpolation (straight lines) between the DAC points.

Logarithmic

Logarithmic interpolation between the DAC points. For the DAC, the interpolation is calculated based on two reference points. For the linear DAC, the interpolation is calculated based on the attenuation in the material.

Polynomial

Third-order polynomial interpolation between the DAC points.

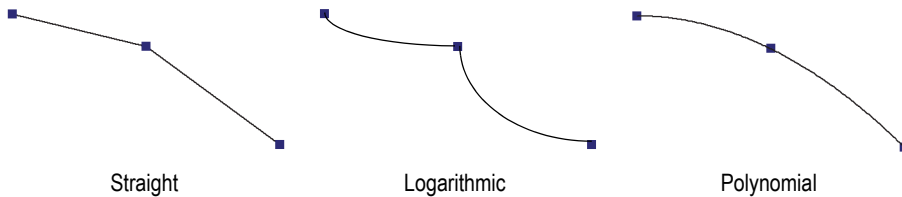


Figure 8-31 Illustration of Curve Type interpolation choices

Curve Qty.

Used to define the number of sizing curves. This parameter is read-only for sizing curves compliant with a standard.

Curves Setup

Used to select the curve whose **Offset** value you want to change.

Offset

Used to specify the amplitude offset (dB) of the curve selected with the **Curves Setup** parameter relative to the main curve [**1 (0.0 dB)**]. The offset range is from -24 dB to $+24$ dB. Two secondary curves cannot have the same offset. **Offset** is only enabled when you select **Sizing > Type Setup > Compliance = Custom** and when the **Curves Setup** parameter is set to **2** or higher (which implies that the **Curve Qty.** is set to **2** or higher).

8.6.3.2 Parameters for DGS

The instrument calculates the distance gain size (DGS) curve based on the echo of one reference reflector and the value specified for the following parameters:

Angle

Used to select the current DGS angle.

Specimen At.

Used to specify the attenuation (dB/mm) of the material for the inspected part. In some cases, you need to calculate the relative attenuation within the inspected part, and then enter the value in this parameter.

Apply

Used to apply the **Specimen At.** parameter to the DGS calibration.

Reset

Used to reset the **Specimen At.** parameter to its previous value. Note that the **Reset** parameter will have no effect after the **Apply** button has been tapped.

Correction

Used to turn **ON** or **OFF** the S-scan amplitude correction. When turned **ON**, the **Correction** parameter normalizes the colors of the signal amplitude on the S-scan in the same way as a TCG. For example, this means that the S-scan will display an orange color at 80 % for the corresponding calibration-curve signal on the A-scan, even if the A-scan represents a signal that is weaker than 80 %.

8.6.3.3 Parameters for AWS Curves

The following parameters are available when you select **Sizing > Type = AWS**:

Standard

Used to specify which version of the AWS code you want to comply with: AWS-D1.1 or AWS-D1.5.

Weld Type

Used to specify the type of weld to be inspected. The available choices are **Statically** and **Cyclically** when you select **Standard = 1.1**, or **Compressive** and **Tensile** when you select **Standard = 1.5**.

Angle

Used to adjust the real angle entering the material for the AWS 45°, AWS 60°, and AWS 70° indication readings.

AWS 45°

Used to adjust the real angle entering the material for the AWS 45 indication reading.

AWS 60°

Used to adjust the real angle entering the material for the AWS 60 indication reading.

AWS 70°

Used to adjust the real angle entering the material for the AWS 70 indication reading.

Thickness

Used to set the thickness of the part.

8.6.4 Curves Setup Submenu

The parameters available for the **Curves Setup** submenu depend on the selection you made for the **Sizing > Type** submenu list value.

8.6.4.1 Parameters for DAC and TCG Curves

The following parameters are available in the **Sizing > Curves Setup** submenu when you select **Sizing > Type = DAC** or **TCG**. These parameters allow you to define a curve point by point. See “Manually Configuring the DAC Curve” on page 162 for the procedure.

Apply To (PA group only)

Used to specify the scope for the DAC sizing curves.

All Laws: The sizing curves apply to all focal laws.

Current Laws: The sizing curves only apply to the current focal laws.

Point

Used to select the point to be edited. The selected point appears as a white dot on the view.

Position

Used to set the position of the selected point on the ultrasound axis.

Amplitude (DAC only)

Used to specify the amplitude (%) for the selected DAC point.

Gain (TCG only)

Used to simultaneously modify the gain of the signal and the gain of the TCG curve. Use this function if you need to modify the gain in order to improve the sizing process.

Add

Used to add a point to the TCG or DAC curve. The DAC/TCG curve can have up to 32 points. The **Add** parameter is disabled in Linear DAC.

Delete

Used to delete the selected point from the TCG or DAC curve.

8.6.4.2 Parameters for Linear DAC Curve

The following parameters are available in the **Sizing > Curves Setup** submenu when you select **Sizing > Type = Linear DAC**.

Mat. Attenuation

Used to set the attenuation of the material for the inspected part.

Delay

Used to specify the offset on the ultrasound axis for the first point of the curve. The offset corresponds to the location at which the ultrasound beam enters the inspected part.

8.7 Gates/Alarms Menu

The **Gates/Alarms** menu contains the parameters needed to configure the gates, alarms, and outputs.

A gate is represented by a red, green, or yellow colored line in the A-scan and S-scan views.

An alarm is a reaction of the instrument to the occurrence of a specific condition related to a gate or to the thickness (**T**) reading. The OmniScan instrument enables you to configure three independent software alarms. You can assign these alarms to one of the three hardware outputs. Each hardware output corresponds to a front-panel indicator, and to a signal on the alarm and I/O connector of the OmniScan instrument. As an option, the output can be configured to emit a sound.

8.7.1 Gates Submenu

The **Gates** submenu contains the parameters used to configure the gates. “Gate Position” on page 300 describes how to modify the gate position and size using the parameter buttons, the Scroll knob, or a mouse.

Gate

Used to specify which gate you want to configure. There are three color-coded gates.

I

Gate I (yellow) is used to synchronize the position of another gate.

A

Gate A (red) is used for general purposes.

B

Gate B (green) is used for general purposes.

Parameters

Used to select a set of gate parameters that appear to the right of the **Parameters** button (see Figure 8-32 on page 296).

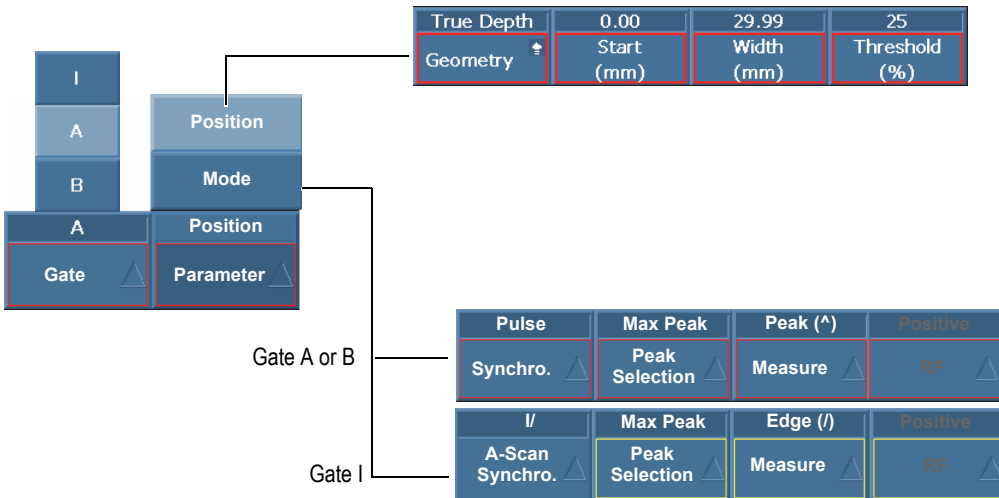


Figure 8-32 Available selection of gate parameters

Select **Gates/Alarms > Gates > Parameters = Position** to configure the following parameters for the gate selected with **Gates/Alarms > Gates > Gate**:

Geometry

Used to set one of the following gate-type options: **True Depth**, **Sound Path**, or **Off**.

True Depth sets the gate according to the depth in the material. **Sound Path** sets the gate according to the distance traveled in the material. **Off** turns off the selected gate.

Start

Used to set the starting position of the selected gate. This position is related to the gate synchronization. The real position of the gate is the synchronization position plus the gate start position.

Width

Used to set the width of the selected gate.

Threshold

Used to set the height value of the selected gate. This parameter determines the amplitude of a signal in the gate for detection.

Select **Gates/Alarms > Gates > Parameters = Mode** to configure the following parameters for the gate selected with **Gates/Alarms > Gates > Gate**:

Synchro. (for **Gates/Alarms > Gates > Gate = A or B**)

Used to specify the synchronization type of the selected gate:

Pulse: Synchronizes at the beginning of the pulse.

I/: Synchronizes where the signal crosses gate I. If the signal does not cross gate I, then it is synchronized at the end of gate I.

A^: Synchronizes at the position of the amplitude peak of gate A. If the signal does not cross gate A, it is synchronized at the end of gate A. If, for gate A, you have selected **Gates/Alarms > Gates > Parameters = Mode** and **Measure = Peak**, **A^** appears in the gate B synchronization.

A/: Synchronizes where the signal first crosses gate A. If the signal does not cross gate A, then it is synchronized at the end of gate A. If, for gate A, you have selected **Gates/Alarms > Gates > Parameters = Mode** and **Measure = Edge**, **A/** appears in the gate B synchronization.

A-Scan Synchro. (for **Gates/Alarms > Gates > Gate = I**)

Used to specify the gate I synchronization type (see Figure 8-33 on page 298):

Pulse: Synchronizes at the beginning of the pulse. Gate I is fixed while gate A and gate B move with the A-scan.

I/: Synchronizes the A-scan starting point where the signal crosses gate I. If the signal does not cross gate I, it is synchronized at the end of gate I. The display starts at the starting position of gate I.

Use this gate synchronization type for immersed inspections to stabilize the display of the signal from the part, and to hide the signal through the water between the probe and the part.

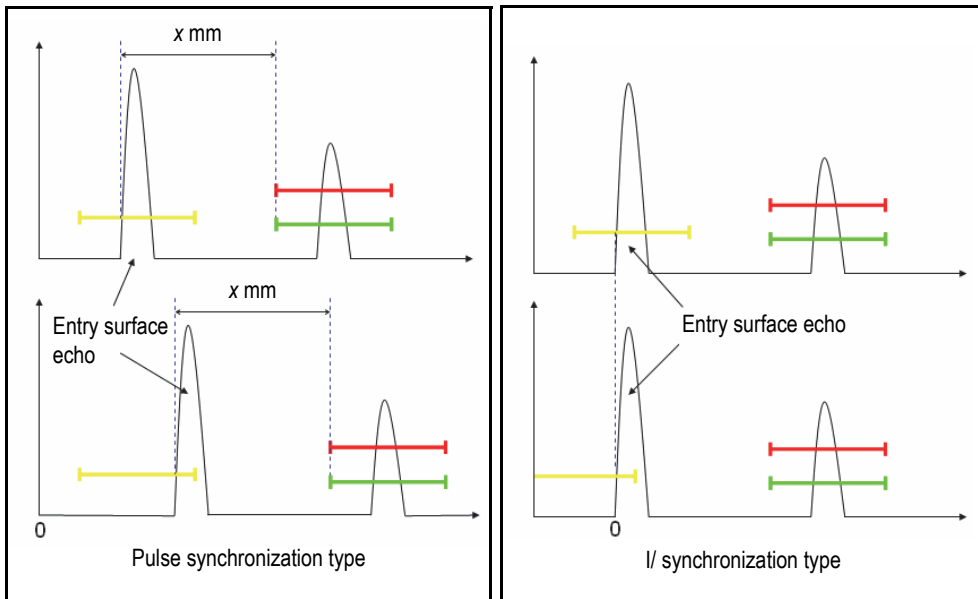


Figure 8-33 A-scan synchronization types for gate I

Peak Selection

The **Peak Selection** parameter can be set to **Max Peak** or **First Peak**.

- When **Max Peak** is selected for a specific gate (A, B, or I), the data, readings, and parameters displayed only correspond to the highest (or maximum) peak crossing this particular gate.
- When **First Peak** is selected for a specific gate (A, B, or I), the data, readings, and parameters displayed only correspond to the first peak crossing this particular gate.

NOTE

The **Peak Selection** parameter should be set independently for each gate (A, B, and I).

When the **Peak Selection** parameter is set to **First Peak (Peak Selection = First Peak)** for a selected gate, the “%” readings and the “^” readings (**Gates/Alarms > Thickness > Source**) automatically correspond to the first peak crossing the selected gate. However, selecting **First Peak** will not affect

the edge readings (/), but rather only those readings corresponding to the selected gate (A, B, or I).




When **First Peak** is selected, the number “1” appears at the end of the reading name. For example, if you select gate A, the **DA^** and **A%** readings change to **DA^1** and **A%1**.

Measure

Used to select whether the measurement is taken at the **Peak** or at the **Edge** of the signal. The selection in this parameter influences all related measurements (see the example in Table 22 on page 299):

- Position C-scans
- Volumetric readings (ex.: SA, PA, DA,...)
- Thickness (Reading “T”, C-scan, Strip Chart)
- Calibration (on Peak or on Edge)
- Gate synchronization

Table 22 Example of the influence of the Measure parameter

Configuration	Result example
Gates/Alarms > Gates > Gate = A Gates/Alarms > Gates > Parameters = Mode Gates/Alarms > Gates > Measure = Edge (/)	Reading SA = SA/ 
Gates/Alarms > Gates > Gate = B Gates/Alarms > Gates > Measure = Peak (^)	Reading SB = SB^ 
Gates/Alarms > Thickness > Source = B^ - A/	Reading thickness = T (B^ - A/) 

RF

Used to specify the signal part used to trigger the gate:

Absolute: The gate is triggered if either the HW+ or HW– signal crosses the gate.

Positive: Only the positive section of the signal triggers the gate.

Negative: Only the negative section of the signal triggers the gate.

8.7.1.1 Gate Position

There are many ways to set the start, width, and threshold of a gate. You can enter the values in the parameter buttons using the function keys (OmniScan MX2 only), Scroll knob, touch screen, or a mouse.

Touch screen

Tapping the left end of the gate opens the **Gate Start** parameter pop-up button in edit mode. Tapping the middle of the gate opens the **Gate Threshold** parameter pop-up button in edit mode. Tapping the right end of the gate opens the **Gate Width** parameter pop-up button in edit mode.

NOTE

When a gate is short, it might be impossible to tap a specific zone. Visual references indicate which sections of the gate are available to tap.

Keys (shortcuts) [OmniScan MX2 only]

Press the Gate key, and then use the Scroll knob to select the parameter to be edited. Press the Accept key to enter the edit mode. Use the Scroll knob to change the value or press the Accept key again to display the virtual keyboard. Press the Accept key to set the value or press the Cancel key to cancel the value. After pressing Accept to accept a value, press Cancel to close the **Gate** pop-up button.

8.7.1.2 Saturated Gate

A saturated gate means that the signal amplitude of at least one focal law element, or the sum of all elements in a beam (the focal law itself), exceeds 200 % within gate A range. If gate A on the A-scan blinks, it means that the saturation is on the current law.

8.7.2 Alarm Submenu

The **Alarm** submenu contains the parameters used to define the alarm conditions. Alarms are useful for highlighting the occurrence of specific conditions.

There are three logical alarms available. A logical alarm is built from the gate conditions. The condition can be assigned to one or several groups. For a description on how to configure an alarm output, see “Output Submenu” on page 302.

Alarm

Used to select the alarm to be edited.

Apply To (OmniScan MX2 only)

Used to select one or several groups (channels) to be used to trigger an alarm.

Condition (for the group selected in the **Apply To** list)

Used to assign a condition for triggering an alarm.

Gate A

The alarm is triggered if the signal crosses gate A.

Gate B

The alarm is triggered if the signal crosses gate B.

Gate I

The alarm is triggered if the signal crosses gate I.

Not Gate A

The alarm is triggered if the signal does not cross gate A.

Not Gate B

The alarm is triggered if the signal does not cross gate B.

Not Gate I

The alarm is triggered if the signal does not cross gate I.

< Min. Thickness

The alarm is triggered if the thickness value becomes smaller than the minimum thickness criteria set in **Gates/Alarms > Thickness > Min.**

> Max. Thickness

The alarm is triggered if the thickness value becomes greater than the maximum thickness criteria set in **Gates/Alarms > Thickness > Max.**

Operator

Used when you have two conditions related to an alarm.

AND

Used when both conditions assigned to the alarm must occur simultaneously in order to trigger the alarm.

OR

Used when either one of the conditions must occur to trigger the alarm.

Condition (for group B)

Assigns a second condition in order to trigger an alarm.

State

Enables or disables the alarm.

NOTE

For more details on alarms, see “Setting an Alarm” on page 161.

8.7.3 Output Submenu

The **Output** submenu contains the parameters needed to configure the output signals available for the alarm triggers. The OmniScan has three alarm outputs and two analog outputs. The following alarm output types are available:

- Visual alarms using the indicators located on the front panel of the OmniScan MX2
- Audio alarm using the buzzer
- TTL alarms using the alarm and I/O connector signals

For procedures on setting alarm outputs, see “Setting an Alarm Output” on page 161.

Alarm

Read-only parameter. Used to indicate which output will be configured. Each output is associated with only one alarm.

Count

Used to configure the number of times an alarm must be triggered in order to activate the associated output.

Sound

Used to enable generation of a sound from the internal speaker.

Delay

Used to set the time delay between the alarm signal and the activation of the alarms (TTL port, visual, and audio alarms).

For online measurement systems, this function is used to trigger a downstream paint gun with the delayed TTL alarm signal. Set the delay to the time it takes for the part to travel from the ultrasonic probe location to the paint gun location.

IMPORTANT

While the delay is elapsing, the OmniScan cannot detect other alarms.

Hold Time

Used to set the time delay between the end of the alarm condition and the deactivation of the alarms (TTL ports, visual, and audio alarms).

8.7.4 Analog Submenu (OmniScan MX2 Only)

The **Analog** submenu contains parameters that enable the A-scan signal to be available for other systems connected to the OmniScan instrument.

Analog Output

Used to select the analog outputs that will be configured. **Analog 1** to **2** are analog signal outputs on which the A-scan signal can be transferred.

Group

Used to select the group from which the A-scan signal is sent to the analog output.

Data

Used to select the type of A-scan data sent to the analog output. The available choices are **A%**, **B%**, and **Thickness**.

A%

Sends the signal crossing gate A to the analog output.

B%

Sends the signal crossing gate B to the analog output.

Thickness

Sends the signal corresponding to the thickness value, as calculated for the T reading, to the analog output.

State

Used to turn the analog output on and off.

8.7.5 Thickness Submenu

The **Thickness** submenu contains parameters for the minimum and maximum values for part thickness measured with the following readings:

Source

Used to select the type of gate combination that provides the thickness value in the T reading (see Figure 8-34 on page 305).

The parameters in this list are a combination of the following variables:

A^

Maximum point (peak) on gate A; available when you select **Gates/Alarms > Gates > Parameter = Mode** and **Gates/Alarms > Gates > Measure = Peak(^)** for gate A.

B^

Maximum point (peak) on gate B; available when you select **Gates/Alarms > Gates > Parameter = Mode** and **Gates/Alarms > Gates > Measure = Peak(^)** for gate B.

A/1

Crossing point on gate A; available when you select **Gates/Alarms > Gates > Parameter = Mode** and **Gates/Alarms > Gates > Measure = Edge(/)** for gate A.

B/1

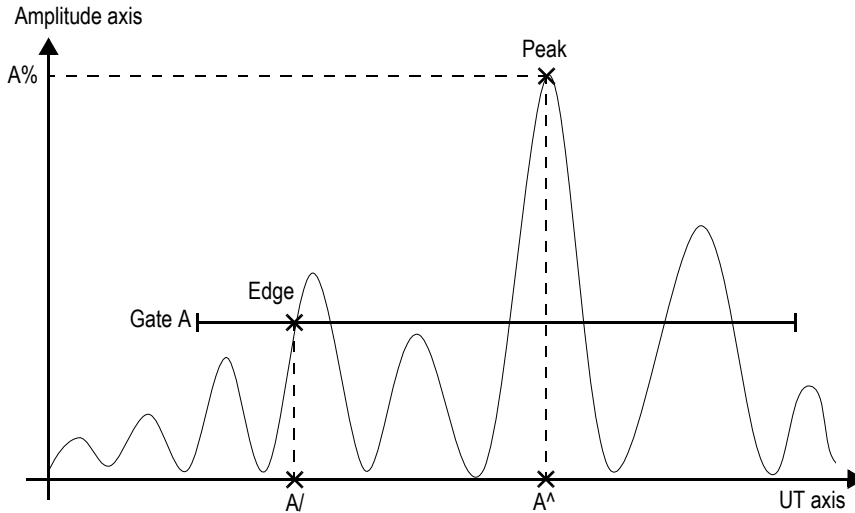
Crossing point on gate B; available when you select **Gates/Alarms > Gates > Parameter = Mode** and **Gates/Alarms > Gates > Measure = Edge(/)** for gate B.

I/1

Crossing point on gate I (always available)

I^

Maximum point on gate I (always available)

**Figure 8-34 Description of the source codes for gate A****Min.**Used to set the minimum allowed thickness for the alarm condition under **Gates/Alarms > Alarm > Condition = < Min. Thickness.****Max.**Used to set the maximum allowed thickness for the alarm condition under **Gates/Alarms > Alarm > Condition = > Max. Thickness.****Echo Qty.**

Used to set the number of back wall echoes used to measure the thickness of a part. This feature is particularly useful when measuring small thicknesses. Taking measurements using many round trips provides a more accurate thickness value.

8.8 Group/Probe & Part Menu

The **Group/Probe & Part** menu contains parameters for defining the inspected part, in addition to the grouping and configuration of pulsers, receivers, probes, and wedges.

8.8.1 Probe & Wedge Submenu

The **Probe & Wedge** submenu provides parameters for defining the probe and wedge to be used with the current group.

Select/Edit

Offers two choices: Select, or define probes and wedges. The other parameter buttons available change according to the selection.

Select (available in pulse echo)

Provides parameters for selecting a probe and wedge from predefined lists.

Select Transmitter (available in pitch-and-catch)

Provides parameters for selecting the transmitter probe and wedge from predefined lists.

Select Receiver (available in pitch-and-catch)

Provides parameters for selecting the receiver probe and wedge from predefined lists.

Probe: Opens the probe selection dialog box to make a selection from a list of predefined probes (see Figure 8-35 on page 307). In the dialog box, select a probe category from the list on the left side, select a probe in the list on the right side, and then choose **Select**.

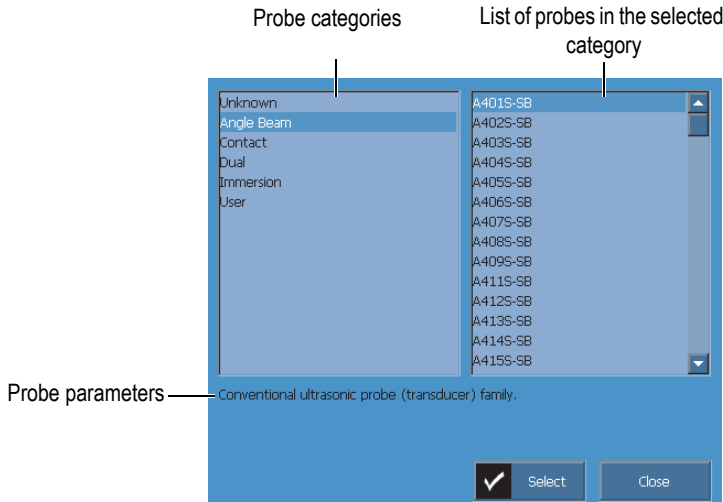


Figure 8-35 The probe selection dialog box

Wedge: Opens the wedge selection dialog box to enable a selection from a list of predefined wedges.

Edit

Used to edit a new probe or new wedge.

Edit Probe: Opens the Probe Definition Manager, enabling you to define a new probe (see “Probe Definition Manager” on page 307 for details).

Edit Wedge: Opens the Wedge Definition Manager, enabling you to define a new wedge (see “Wedge Definition Manager” on page 310 for details).

Auto Detect

When **On** is selected, the autodetection chip in the probe automatically loads the definition and parameters.

Wedge Length (available only if the specimen is specified as **Tube > OD > Axial**)

Used to specify the length of the wedge to generate accurate focal laws according to this specific part geometry. This value will be automatically filled for generic wedges, but must be manually entered for custom wedges.

8.8.1.1 Probe Definition Manager

The Probe Definition Manager is used to define the parameters of a custom probe.

Close

Closes the Probe Definition Manager and switches back to the data view screen with no probe selected.

Browse

Used to consult the parameters of the available probe definitions.

Select and Close: Used to select the probe and close the Probe Definition Manager.

New

Creates a new probe template.

Serial Number: Used to enter the serial number of the new probe.

Model: Used to select an existing probe model on which the probe you are defining will be based.

Edit

Used to edit the parameters of the selected probe.

Type (UT probe)

Used to select the number of elements for the probe (**Single** or **Dual**).

Freq.

Used to set the probe frequency.

Geometry.

Used to set the probe-element geometry (**Circular** or **Rectangular**).

Element Diameter (UT group only)

For circular probe elements, used to set the probe-element diameter.

Side 1 Length and **Side 2 Length** (UT group only)

For rectangular probe elements, used to set the probe-element size.

Ref. Point (PA group only)

Sets the probe reference point, which represents the distance between the front of the probe (wedge) and the first element of the probe (see Figure 8-36 on page 309).

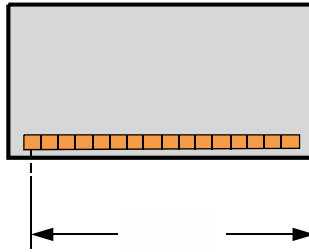


Figure 8-36 The probe reference point measurement

Type (PA probe)

Used to select **Custom**, **Angle Beam**, **Contact**, **Immersion** (PA group only), or **Single/Dual** (UT group only).

Element Qty. (PA group only)

Used to set the number of probe elements.

Pitch (PA group only)

Used to set the probe pitch center-to-center distance.

Manage

Provides parameters to manage the probe definition files on the storage card.

Change Serial #

Used to edit the serial number of a probe.

Delete

Used to delete the selected probe.

Copy

Used to copy the selected probe definition.

Save

Saves the information selected to define your probe.

To start the Probe Definition Manager

1. Select **Group/Probe & Part > Probe & Wedge** or go to the Setup wizard **Select Probe and Wedge** step.
2. Choose **Select/Edit = Edit**.

3. Choose **Edit Probe**.

8.8.1.2 Wedge Definition Manager

The Wedge Definition Manager is used to define the parameters of a custom wedge.

Close

Closes the Wedge Definition Manager and switches back to the data view screen with no wedge selected.

Browse

Used to consult the parameters of the available wedge definitions.

Select and Close

Used to select the wedge and close the Wedge Definition Manager.

New

Creates a new wedge template.

Serial Number

Used to enter the serial number of the new wedge.

Model

Used to select the model number of the new wedge.

Edit

Used to edit the parameters of the selected wedge.

Wedge Angle (PA group only)

Used to specify the angle of the ultrasonic beam in the wedge.

Refracted Angle (UT group only)

Used to specify the angle of the ultrasonic beam in the wedge.

Probe Delay (UT group only)

Used to specify the delay in the wedge.

Wave Type (UT group only)

Used to select the wave type (**LW** or **SW**) in the wedge.

Ref. Point (UT group only)

Used to set the probe and wedge assembly reference point, which is the distance between the front of the wedge and the beam exit point (see Figure 8-37 on page 311). The beam exit point is usually marked by a line on the wedge.

This value is negative because the OmniScan MXU software establishes the 0-reference point by default at the beam exit point. To place the 0-reference point at the front edge of the wedge, you must measure the distance between the front edge of the wedge and the beam exit point, and then subtract it from the default 0-reference point (UT group only).

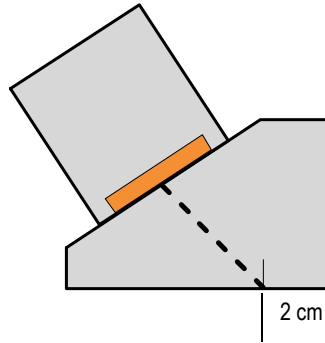


Figure 8-37 The wedge reference point measurement

Orientation (PA group only)

Used to set the orientation of the wedge (**Normal** or **Reverse**).

Velocity

Used to set the velocity of the wedge material.

Pri. offset (PA group only)

Specifies the primary offset, which is the distance between the front of the wedge and the first element of the probe (see Figure 8-38 on page 311).

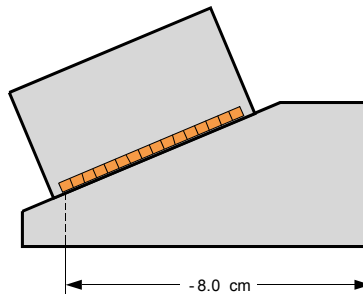


Figure 8-38 The primary offset measurement

Sec. offset (PA group only)

Used to set the offset of the wedge relative to the secondary axis. The value is usually 0 because it represents the center of the probe on the passive axis.

Height (PA group only)

Used to set the height of the wedge.

Manage

Manages the wedge definition files on your memory card.

Change Serial #

Changes the serial number of a wedge.

Delete

Deletes a selected wedge profile.

Copy

Copies a selected wedge profile.

Save

Saves the information selected to define your wedge.

To start the Wedge Definition Manager

1. Select **Group/Probe & Part > Probe & Wedge**, or go to the **Setup** wizard **Select Probe and Wedge** step.
2. Choose **Select/Edit = Edit**.
3. Choose **Edit Wedge**.

8.8.2 Position Submenu

The **Position** submenu contains the parameters that define the relationship between the probe and the 0, 0 position for the inspection, thus ensuring data is displayed correctly.

Scan Offset (OmniScan MX2 only)

Used to define the difference on the scan axis between the 0, 0 position of the inspection and the front face of the probe.

Index Offset

Used to define the difference on the index axis between the 0, 0 position of the inspection and the front face of the probe. Negative for skew 90, and positive for skew 270. The weld centerline is typically 0 on the index axis (see Figure 8-39 on page 313).

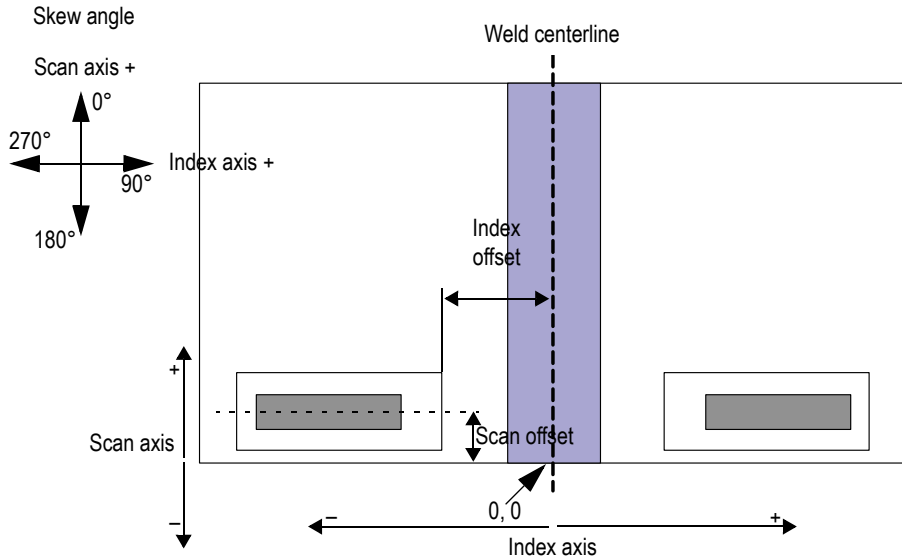


Figure 8-39 Scan and index offsets

Scan Type (TOFD)

Used to select the orientation (**Parallel** or **Nonparallel**) of the ultrasonic beam with regard to the scan axis (see Figure 8-40 on page 314). Only available for the TOFD group.

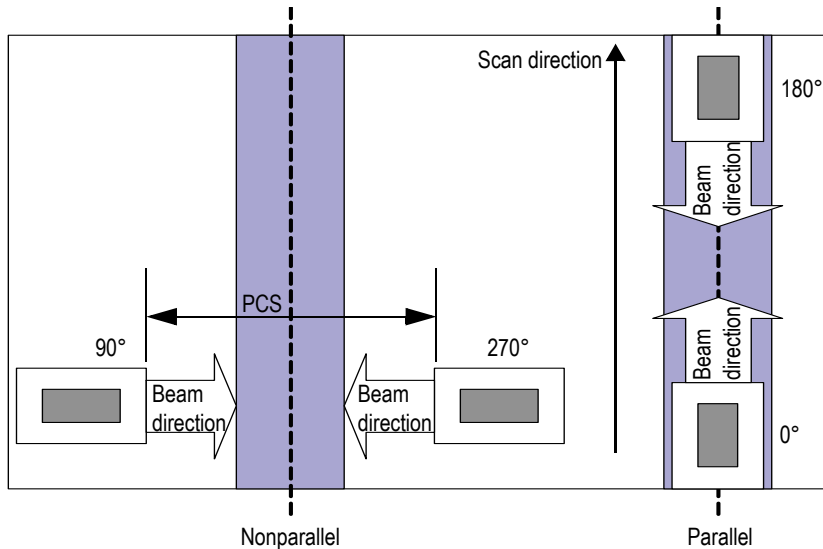


Figure 8-40 The parallel and nonparallel scan types

PCS (TOFD)

Used to set the probe center separation (PCS). This is the exit point-to-exit point distance between the two probes. Only available for the TOFD group (see Figure 8-40 on page 314).

Skew

Orientation of the ultrasonic beam relative to the scan axis.

8.8.3 Parts Submenu

The **Parts** submenu contains the information pertaining to the part to be inspected.

Material

Used to select the component material. This will set the velocities for both the shear and longitudinal waves for this material.

Thickness

Used to set the thickness of the part.

NOTE

When **Display > Overlay > Indicators = Legs** is checked, dotted lines spaced at intervals corresponding to the specified thickness value appear on the views.

Specimen Type

Used to specify the geometry of the part to be inspected.

Plate

A flat surface

Tube

A curved surface

CSC Mode

Used to define the curved-surface correction for the geometry of the part to be inspected.

OD

The inspection is performed on the outer diameter of a tube.

ID

The inspection is performed on the inner diameter of a tube.

Weld Orientation

Used to define the weld orientation on the pipe. This parameter is only available if the **Specimen Type** is set to **Tube**.

Axial

The weld is oriented lengthwise on the tube.

Circ.

The weld is circumferentially oriented on the tube.

Diameter

Used to set the diameter of a tube. This parameter is only available if the **CSC Mode** parameter is set to either **ID** or **OD**.

8.9 Focal Law Menu (PA Group Only)

The **Focal Law** menu manages the parameters related to the probe programming.

8.9.1 Configuration Submenu

The **Configuration** submenu manages the parameters related to the scan types that will be performed by the probe.

Law Config.

Used to specify the scan type for the selected probe group.

Sectorial

Provides a multiple angle scan using the same elements for every angle of the scan (see Figure 8-41 on page 316).

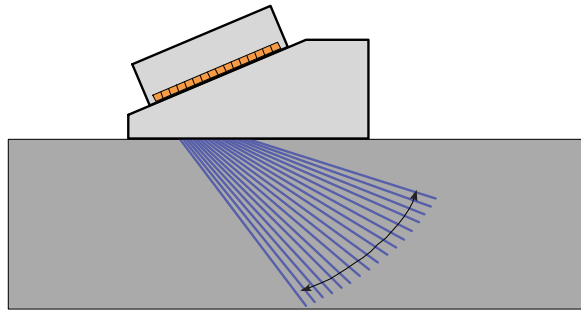


Figure 8-41 Sectorial scanning

Linear

Provides a linear scan at a configurable angle (see Figure 8-42 on page 317). You can use this mode at a zero-degree angle if you do not need overlapping scans.

Linear at 0°

Provides a volume corrected linear sweep at zero degrees (see Figure 8-42 on page 317). Use this mode when you perform overlapping scans. This feature is available with one group only.

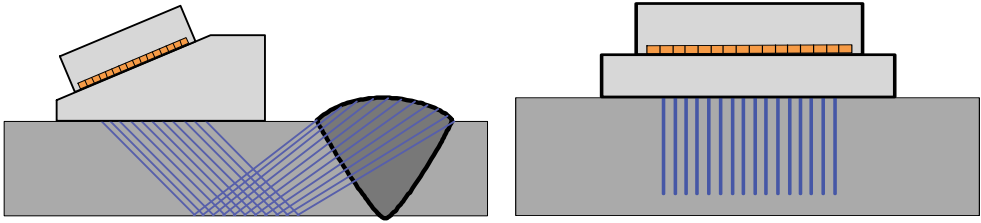


Figure 8-42 Linear scanning with an angle (left) and at zero degrees (right)

Compound

Provides a multiple angle scan using different elements (with the same aperture across the length of the probe) for every angle of the scan (see Figure 8-43 on page 317).

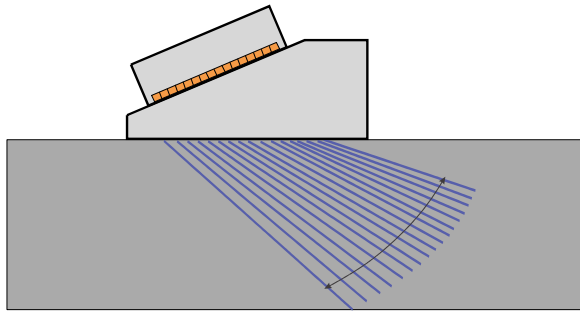


Figure 8-43 Compound scanning

Pulser (OmniScan MX2 only)

Used to enter the number of the first element of the pulser probe when the automatic probe selection is off (in **Group/Probe & Part > Probe & Wedge > Auto Detect = Off**). Used to select the pulser (UT group). This parameter is required when using a splitter for multiple-probe inspection. Typically 1 and 65 for a two-probe inspection using a XX:128 instrument.

Receiver (OmniScan MX2 only)

This read-only parameter displays the receiver of the selected channel. When in P/E mode, the **Receiver** parameter is set to the same value as the **Pulser** parameter.

8.9.2 Aperture Submenu

The **Aperture** submenu manages the selection of the elements used to create each beam (focal law).

Element Qty.

Used to set the number of elements in the aperture.

First Element

Used to set the first element in the aperture.

Last Element

In the **Sectorial** scan type, this parameter indicates the last element to be used. In the **Linear** scan type, this selection is used to set the last element to be used in order to determine the number of laws.

Element Step

Used to set the element step in the aperture when the **Linear** scan type is chosen.

8.9.3 Beam Submenu

The **Beam** submenu manages the configuration of the beams (angle and focus points).

Min. Angle

Used to set the minimum angle of the beam (**Sectorial** law configuration).

Max. Angle

Used to set the maximum angle of the beam (**Sectorial** law configuration).

Angle Step

Used to set the step value between each angle (**Sectorial** law configuration).

Focus Depth

Used to set the focusing depth in the part to be inspected.

Angle

Used to set the law angle (**Linear** law configuration). When the law configuration is set to **Linear at 0°**, the **Angle** is 0 and cannot be edited.

8.10 Scan Menu

The **Scan** menu contains parameters for configuring the scan inspection sequence (encoders, inspection type, data storage, and the area of the part for inspection).

8.10.1 Inspection Submenu

The **Inspection** submenu contains the configuration for the different scan types.

Type

Used to select the desired type of inspection. The following choices are available:

One-Line Scan

In one-line scanning, the acquisition is based either on an encoder or the internal clock (counts/s).

Raster Scan

When the phased array probe is moving on both the scan and index axes, the ultrasound data is acquired in a bidirectional or unidirectional scanning pattern.

Scan

Used to specify the source of the data positioning for the scan axis.

Time

Data acquisition at precise time intervals. Not available for **Type = Raster Scan**.

Encoder 1

Encoder 1 is the source used for data positioning for the scan axis.

Encoder 2

Encoder 2 is the source used for data positioning for the scan axis.

Index

Used to specify the source of the data positioning for the index axis. The possible choices are as follows:

Encoder 1

Encoder 1 is the source used for the data positioning for the index axis.

Encoder 2

Encoder 2 is the source used for the data positioning for the index axis.

Scan Speed (mm/s or in./s)

Used to set the scanning speed, expressed in mm/s or in./s. Modifying this parameter automatically adjusts the Acq Rate value. The unit type (length or angle) depends on the selection for the **Scan-Axis Unit** parameter. This parameter is available only when **Scan > Inspection > Scan = Time**.

Max. Scan Speed

Used to set the maximum scanning speed of your scanner, expressed in mm/s or in./s, to automatically adjust the PRF value, which prevents gaps in the data. This parameter is only available when **Scan > Inspection > Scan = Encoder 1 or Encoder 2**.

TIP

Scanning mechanisms can produce irregular scanning speeds, which may at times exceed the rated scan-speed value. In such cases, you can prevent gaps in the acquired data by using the **Max. Scan Speed** parameter to enter a value slightly higher than the rated speed of the scanning system.

8.10.2 Encoder Submenu

The **Encoder** submenu contains the encoder configuration.

NOTE

The **Encoder** submenu only appears when **Scan > Inspection > Scan = Encoder 1 or Encoder 2**.

Encoder

Used to select the encoder to be configured (1 or 2).

Polarity

Used to reverse the count of the encoder if the probe is only able to scan in the opposite direction.

Type

Used to select the type of the currently selected encoder. There are two encoder hardware inputs that can be configured independently. Each encoder input has two channels, A and B, enabling a dual-channel encoder for quadrature readings of the resolution.

Clock/Dir

Select this item when you use a stepper controller, and its documentation specifies that the position output signal is a clock/direction type (5 V pulse for the position/speed and 5 V signal for the direction).

Quad

Select this item when the attached encoder (5 V TTL output) is a dual-channel output encoder. The channels are generally designated A and B. When the encoder is rotating clockwise (from left to right in Figure 8-44 on page 321), channel B follows channel A with a 90-degree delay. When the encoder is rotating counterclockwise, channel A follows channel B with a 90-degree delay. In this way, you can determine if the rotation is clockwise or counterclockwise. The decoder counts one step each time it detects a rising or falling edge on channel A or channel B. This means that if the real encoder resolution is 1000 steps/revolution, the final resolution with the quadrature reading is 4000 steps/revolution.

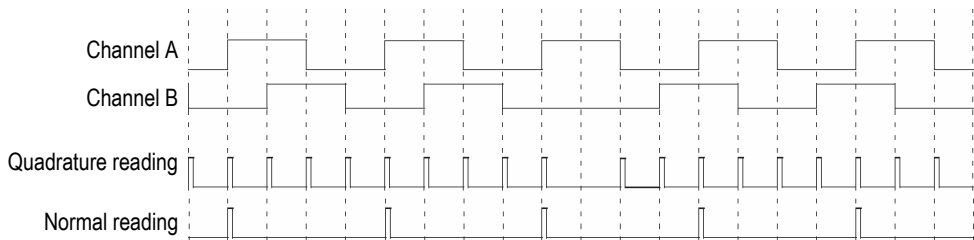


Figure 8-44 Quadrature and normal reading of the resolution

Up

The decoder only reads channel A, and increments the counter even if the encoder is turning counterclockwise. The final resolution is the real encoder resolution.

Down

The decoder only reads channel A and decreases the counter even if the encoder is turning clockwise. The final resolution is the real encoder resolution.

Clock/Dir Up

The decoder only reads channel A and increments the counter. When the signal of channel B (direction) is high, the acquisition stops to prevent the data from being overwritten while you are moving the probe back, and the counter is being decreased.

Clock/Dir Down

The decoder only reads channel A and decreases the counter. When the signal of channel B (direction) is high, the acquisition stops to prevent the data from being overwritten while you are moving the probe back and the counter is being increased.

Quad Up

The decoder reads channel A and channel B in quadrature mode (4 times the encoder resolution), and increments the counter when the encoder rotates clockwise. When the encoder rotates counterclockwise, the acquisition stops to prevent overwriting the data, and the counter decreases.

Quad Down

The decoder reads channel A and channel B in quadrature mode (4 times the encoder resolution), and decreases the counter when the encoder rotates counterclockwise. When the encoder rotates clockwise, the acquisition stops to prevent overwriting the data, and the counter increments.

Clicker

Used when you use an encoder on Encoder 1, and an Olympus indexer device (clicker) on Encoder 2. The encoder measures the position on the scan axis. Press the push-button of the indexer device to increment the position on the index axis after manually moving the probe (bidirectional raster scan). This procedure is frequently used to perform a manual raster scan.

Clicker + Preset

Used when you use an encoder on Encoder 1, and an Olympus indexer device (clicker) on Encoder 2. The encoder measures the position on the scan axis. Press the push-button of the indexer device to increment the position on the index axis and to reset the position on the scan axis after manually moving the probe (unidirectional raster scan). This procedure is frequently used to perform a manual raster scan.

Resolution

Used to set the number of counts per unit for the selected encoder. Expressed in Step/mm or Step/Rotation.

Origin

Used to set the value to which the selected encoder is preset when the **Preset** command button is used.

Preset

Resets the selected encoder to its **Origin** value.

8.10.3 Area Submenu

The **Area** submenu contains the parameters for the area to be scanned. The area is the same for all groups.

Scan Start

Used to set the starting location of the scan axis (expressed in mm or in.).

Scan End

Used to set the length of the scan axis (expressed in mm or in.).

Scan Res.

Used to set the step (resolution) at which the points will be acquired on the scan axis.

Index Start

Used to set the starting location of the index axis. Available for the **Raster Scan** type only.

Index End

Used to set the length of the index axis. Available for the **Raster Scan** type only.

Index Res.

Determines the index resolution. Available for the **Raster Scan** type only. Cannot be modified in the **Linear at 0°** scan.

Encoder Tolerance

Determines the index encoder overlapping zone on the index axis. The data acquired for this zone is transferred to the next index step to avoid losing it. Available for the **Linear** scan only.

8.10.4 Data Submenu

The **Data** submenu contains data storage parameters, which are only available for acquisitions synchronized on an encoder. Data is stored according to the parameter selected in the **Storage** list. The number of acquisition points to be saved is determined in the **Area** submenu according to the scan length and inspection resolution, and also according to the memory capacity of the OmniScan instrument.

When the scan goes over the same position (scan, index), the data saved is based on the selection made in the **Storage** list.

Storage

Last

Saves the last data.

A%

Saves data for the signals with the highest amplitude detected in gate A.

Max. Thickness

Saves data based on the maximum thickness calculation. The thickness is calculated according to the value specified in **Gates/Alarms > Thickness > Source**.

Min. Thickness

Saves data based on the minimum thickness calculation. The thickness is calculated according to the value specified in **Gates/Alarms > Thickness > Source**.

8.10.5 Start Submenu

The **Start** submenu contains parameters for starting and stopping inspections.

Start Mode

Used to specify the action to be applied when the acquisition is started (using the

Play key [] or the **Start** parameter button).

Reset All

Resets all parameters (resets both encoders to their **Origin** values and clears the data).

Reset Encoder

Resets both encoders to their **Origin** values.

Reset Data

Clears the data.

Start

Applies the action specified in the **Start Mode** parameter button.

TIP

Press the Play key () to directly access the **Scan > Start > Start** parameter.

Pause

Controls the acquisition by toggling between inspection mode and analysis mode.


On

Stops the acquisition and freezes the current data display. The instrument operates in analysis mode.

Off

Starts the acquisition and unfreezes the current data display. The instrument operates in inspection mode.

TIP

Press the Pause key () to directly toggle **Scan > Start > Pause = On/Off**, and toggle between analysis mode and inspection mode (see Table 2 on page 17 or Table 3 on page 20 for details).

8.11 Preferences Menu

The **Preferences** menu includes submenus for configuring various aspects and accessing tools.

8.11.1 Setup Submenu

The **Setup** submenu contains parameters related to the various hardware and software configurations.

Category

Used to select the category for the other parameters of the submenu. Selecting an item in this list changes the parameter buttons available to the right of the **Category** button. The following parameter categories are available:

- **DIN** (see “DIN Category” on page 326)
- **Gates** (see “Gates Category” on page 328)
- **250 % Mode** (see “250 % Mode Category” on page 328)
- **Interpolation** (see “Interpolation Category” on page 328)
- **Interleave** (see “Interleave Category” on page 328)

8.11.1.1 DIN Category

Select **Preferences > Setup > Category = DIN** to view the following parameters, which are required when configuring the digital inputs (DIN). Each of the four **DIN n** parameters has an exclusive function. The other listed functions can be assigned to any digital input.

Use the digital inputs and the digital outputs to control the OmniScan instrument remotely. Connect your remote controller to the appropriate OmniScan connector. Refer to the *OmniScan MX and MX2 User’s Manual* or *OmniScan SX User’s Manual* for details on the signals and connectors.

Select DIN

Used to select a digital input (**DIN1** through **DIN4**), for which you assign a function in the **Assign DIN** list.

Assign DIN

Used to assign a function to the digital input selected in the **Select DIN** list. The following functions are available:

Preset Encoder 1

DIN1 exclusive function. When this parameter is selected, if the remote signal Din1/Preset1 is set to the high level, the encoder 1 position will automatically be set to the origin. This is equivalent to manually selecting **Scan > Inspection > Scan > Encoder 1**, and then selecting **Scan > Encoder > Preset**. While the remote signal is high, the encoder is maintained at its origin position, even if the probe is moved.

Preset Encoder 2

DIN2 exclusive function. When this parameter is selected, if the remote signal Din2/Preset2 is set to the high level, the encoder 2 position will automatically be set to the origin. This is equivalent to manually selecting **Scan > Inspection > Scan > Encoder 2**, and then selecting **Scan > Encoder > Preset**. While the remote signal is high, the encoder is maintained at its origin position, even if the probe is moved.

Acquisition State

DIN3 exclusive function. When this parameter is selected, setting and maintaining the remote signal Din3/AcqEn at the high level activates the data acquisition at the hardware level. Setting and maintaining the remote signal at the low level inhibits data acquisition.

Pause/Resume

Used to switch back and forth between inspection mode and analysis mode. The mode changes when the remote signal rises from the low to the high level. This is equivalent to manually selecting **Scan > Start > Pause = On or Off**, or pressing the Pause key.

Save Data

Used to save the data when the remote signal rises from the low to the high level. This is equivalent to manually selecting **File > Data > Save**, or pressing the Save/Print key.

Clear All

Used to clear all data when the remote signal rises from the low to the high level. This is equivalent to manually selecting **Scan > Start > Start Mode = Reset All**.

State

Used to enable or disable the DIN.

NOTE

The selection for the **File > Data Settings > Save Key** parameter determines the type of data that is saved (**Report, Data, or Image**).

8.11.1.2 Gates Category

Select **Preferences > Setup > Category = Gates** to view the following parameter:

Gate Mode

The **Gate Mode** can be applied to all contexts (**By Group**), or to only one context at a time (**By Context**).

8.11.1.3 250 % Mode Category

Set the **250 % mode** to **ON** (default value) to have access to **250 %** A-scan readings and a 10 dB range of attenuation gain in analysis mode.

8.11.1.4 Interpolation Category

Interpolation is only available with the OmniPC software; set this parameter to **ON** (default value) to smooth the amplitude C-scan image.

8.11.1.5 Interleave Category

Set this parameter to **ON** (default value) to enable interleaving of the focal law firing sequence, which results in a delayed apparition of ghost echoes.

8.11.2 Instrument Submenu

The **Instrument** submenu contains parameters used to configure various aspects of the instrument. These parameters are permanently saved in the instrument, and not in the setup file.

Category

Used to select the category for the other parameters of the submenu. Selecting an item in this list changes the parameter buttons available to the right of the **Category** button. The following parameter categories are available:

- **Units** (see “Units Category” on page 329)
- **Screen** (see “Screen Category” on page 329)
- **System** (see “System Category” on page 330)
- **Network Settings** (see “Network Settings Category (OmniScan MX2 Only)” on page 330)
- **External Storage** (see “External Storage Category (OmniScan MX2 Only)” on page 331)
- **Help** (see “Help Category” on page 332) [available with the RayTracing option]

8.11.2.1 Units Category

Select **Preferences > Instrument > Category = Unit** to view the following parameter:

Length Units

Used to specify either metric (**Millimeters**) or US customary (**Inches**) length measurement units.

8.11.2.2 Screen Category

Select **Preferences > Instrument > Category = Screen** to view the following parameters:

Scheme

Used to select the appropriate color and contrast scheme for the OmniScan instrument screen for either indoor or outdoor operation.

Indoor

Color set for indoor use.

Outdoor

Color set for outdoor use.

Touch Screen

Used to enable (**On**) or disable (**Off**) the touch screen. When the touch screen is disabled, it will be temporarily enabled when a virtual keyboard appears.

Brightness

Used to set the brightness level of the screen from 1 % through 100 % in increments of 1 %. Set by default to 75 % to minimize battery consumption. The maximum value is 75 % when operating the OmniScan instrument on battery power (to minimize consumption).

Calib. T. Screen

Used to recalibrate the touch screen if necessary. Follow the on-screen instructions to perform the recalibration.

8.11.2.3 System Category

Select **Preferences > Instrument > Category = System** to view the following parameters:

Clock Set

Used to set the time (see “Setting the Date and Time” on page 79 for more details).

Date Set

Used to set the date. Select once to select the year, twice to select the month, and three times to select the day.

Admin. Password

Used to change the administrator password. This parameter is used when unlocking a setup to access the Microsoft Windows CE operating system.

Manual Boot

Used to reset the automatic boot function.

In the launcher screen, you can select the **Always boot the selected application** check box in order to skip the launcher screen and automatically start the selected application. To reset this function, select **Manual Boot**.

8.11.2.4 Network Settings Category (OmniScan MX2 Only)

The **Network Settings** category contains the parameters used to connect the OmniScan MX2 to a network. Contact your network administrator to obtain appropriate values for the following parameters.

DHCP

Set to **On** to enable the OmniScan MX2 start-up process to request an IP address from the DHCP server. Set to **Off** to make the OmniScan MX2 start-up process use the fixed IP address specified in the **IP address** parameter.

IP address

Used to manually set an IP address for the OmniScan MX2.

Subnet Mask

Used to manually set the subnet mask coordinates for the OmniScan MX2.

Apply

Applies the changes made to the parameters of the **Network** submenu.

8.11.2.5 External Storage Category (OmniScan MX2 Only)

Select **Preferences > Instrument > Category = External Storage** to view the following parameters:

Remote PC

Used to specify the name of the computer connected to the OmniScan instrument. The name is listed in the computer's Windows **System Properties** dialog box, as shown in the example in Figure 8-45 on page 332. See "Configuring OmniScan MX2 Network Connection — Windows XP" on page 209 for the detailed procedure.

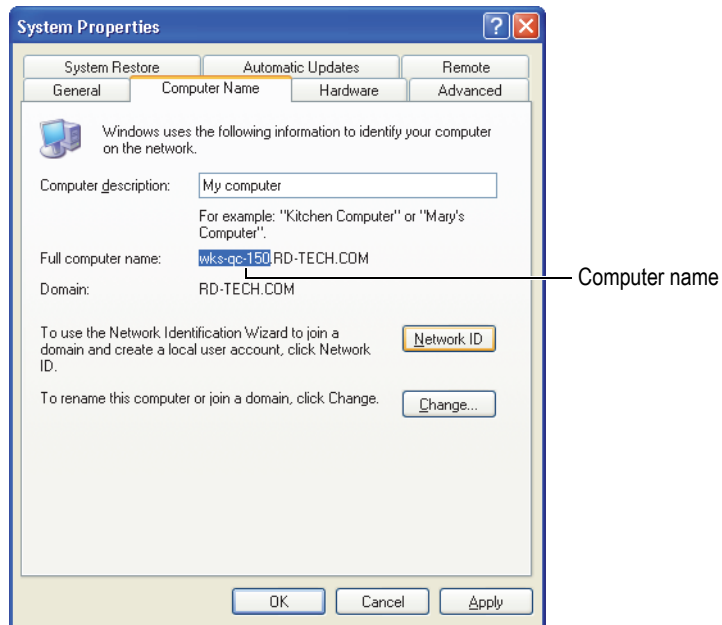


Figure 8-45 Computer name in System Properties dialog box (Windows XP)

Connect

Toggles the state of the connection between the OmniScan instrument and the PC.

8.11.2.6 Help Category

Select **Preferences > Instrument > Category = Help** to view the following parameter:

Wizard Help

Used to disable the help information display for the wizard steps, thus enabling the RayTracing illustration to use the full graphic area. The default value is **On**.

8.11.3 Tools Submenu

The **Tools** submenu contains the parameters used to access various tools.

File Manager

Opens the File Manager, which contains the following parameters:

Close

Closes the File Manager and returns to the data view.

File Type

Used to specify the type of file you are looking for. All other types are ignored.

Create Folder

Creates an empty folder in the selected folder.

Select

Used to select a file.

Select All

Used to select all files in the selected folder.

Copy

Copies the selected files into the folder displayed in the right pane.

Move

Moves the selected files into the folder displayed in the right pane.

Delete

Deletes a selected file.

Rename

Used to rename a selected file.

System Info.

Displays system information such as the hardware version number, the software version number, and the software options.

To close the **System Info.** window, select **Close**, or press the Cancel key ()

Win CE

Used to access the Microsoft Windows CE operating system environment. The administrator password is required to complete the operation. See "Setting the Administrator Password" on page 186 to set the administrator password.

To switch back to the OmniScan instrument interface, you can use either the ALT+TAB key combination (if you have a keyboard plugged in), or the OmniScan button on the Windows taskbar.

8.11.4 FFT Submenu

The FFT (characterize) submenu contains parameters that enable you to characterize a probe. Characterization is useful for estimating the degradation of a probe and verifying whether its nominal frequency is shifting or spreading.

Characterization uses a fast Fourier transform (FFT) to display the frequency content curve and data for a clean echo produced by the probe. See “Characterizing a Probe Using the FFT” on page 102 for a detailed procedure.

IMPORTANT

The FFT is only available with **UT Settings > Receiver > Rectifier = RF, File > Data Settings > Data Selection = All A & C-Scans** when the compression (scale factor) value is 1, and gate A is fully visible in the A-scan. You can only save the probe characterization data to the setup file if the probe is defined.

FFT

When the option is turned on, the FFT curve and data below the A-scan are displayed (see Figure 8-46 on page 335). When you turn this function off, you have the option to save the probe characterization data to the setup file. The saved data can also be generated in reports. In order to ensure the FFT appears in the report, you must include the probe in the report (**File > Report > Category = Format and Component = Probe**).

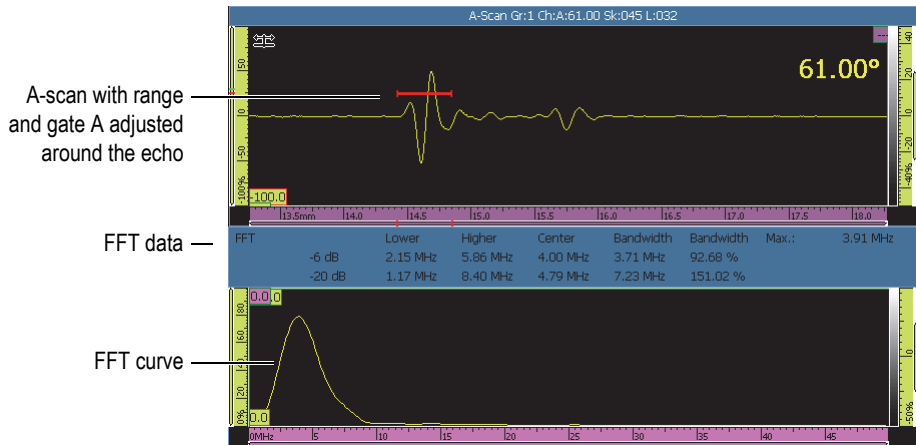


Figure 8-46 FFT curve and data example

The FFT parameter is used to display the FFT curve (**On** or **Off**) [see Figure 8-46 on page 335].

Gain

Used to modify the gain value to optimize the echo amplitude.

Start

Used to set the starting position of gate A.

Width

Used to set the width of gate A.

Procedure

Used to enter a procedure name that is saved with your probe characterization data.

Block Name

Used to enter the identification of the calibration block used for the characterization. The name is saved with your probe characterization data.

8.12 Setup Configuration Menu

OmniScan MXU includes a setup configuration menu that enables you to efficiently review and adjust multiple settings:

- You can quickly review all group settings without having to pass through menus, wizards, or reports.
- You can quickly adjust all visible parameters while remaining in acquisition mode:
 - Each parameter is controlled in the same way as it is in the wizards and other areas of the software interface.
 - Parameter changes are calculated when you tap **Accept**, which enables rapid changes to multiple values.


To access the setup configuration menu

1. Tap and hold the Angle/VPA section of the title bar.
The **Setup Configuration** button appears.
2. Tap **Setup Configuration**.
All main configuration parameters are displayed on the screen for immediate access in a single location (see Figure 8-47 on page 337).



Figure 8-47 The Setup Configuration screen

To modify parameters

1. If necessary, tap the **Group Name** button and name the current group.
2. Tap the parameter that you want to modify.
3. Use the Scroll knob or the software's virtual keyboard to adjust the parameter value.
(This follows the standard method for making adjustments on the OmniScan.)
4. Press the Accept key ().
5. Repeat the adjustment steps for any other parameters, as required.

6. When you have completed the necessary parameter adjustments, proceed as follows:

- ◆ Tap **Accept & Next** to accept and apply the modifications to the current group.

The configuration screen for the next group becomes available for adjustment.

OR

Tap **Accept & Close** to accept and apply the modifications and exit configuration.

The configuration screen closes and you return to the “live scan” screen.

OR

Tap **Cancel** to discard your adjustments to the current group and exit configuration.

The configuration screen closes and you return to the “live scan” screen.

NOTE

- You can adjust parameters in any order.
- You can access and modify only the necessary parameters (there is no need to adjust all parameters).
- The Probe and Wedge Manager screen appears when you access the **Probe** and **Wedge** parameters. This lets you select from a list of available models.
- If your adjustments cause an incorrect parameter combination for a focal law, the **Accept** buttons turn red and remain disabled until you enter acceptable values.
- You can exit the configuration screen at any time by tapping **Cancel**.

You can access the configuration screen in both acquisition and analysis modes, either through the menu screen or full-screen mode (for both MXU and OmniPC software). In analysis mode, some of the displayed parameters cannot be adjusted. However, you can adjust the **Scan Offset**, **Index Offset**, and **Skew** parameters.

Certain setup adjustments are not possible through the setup configuration screen. For example, if you are creating a completely new scan setup, you must still use the **Part & Weld** and **Setup** wizards.

8.13 Scan Configuration Menu

A scan configuration menu is available in both acquisition and analysis modes (for both MXU and OmniPC software). In acquisition mode, you can use it to make “live” parameter adjustments. In analysis mode, you can use it to review the parameters.

To use the scan configuration menu

1. Tap and hold either the Scan or Index section of the title bar.
The **Scan Configuration** button appears.
2. Tap **Scan Configuration**.
The scan configuration parameters are displayed on the screen (see Figure 8-48 on page 339).

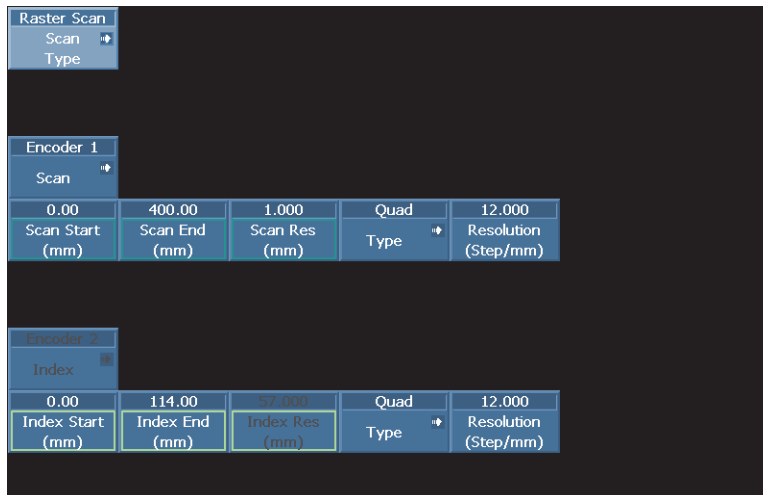
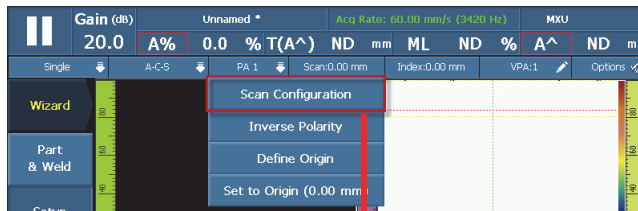


Figure 8-48 The Scan Configuration parameters

3. If necessary, tap **Scan Type** to change the type of scan (for example, to change from a one-line scan to a raster scan).

The corresponding parameters for the selected scan type are displayed.

4. Make any necessary parameter adjustments.
5. Tap **Accept** to apply the modifications and exit the scan configuration screen.

OR

Tap **Cancel** to discard your modifications and exit the scan configuration screen.

List of Figures

Figure 1-1	Front panel controls of the OmniScan MX2	8
Figure 1-2	Front panel controls of the OmniScan SX	9
Figure 1-3	The Shut Down button	10
Figure 1-4	Saving the setup	11
Figure 1-5	Requirements for UT probe connection	12
Figure 1-6	Requirements for PA probe connection	12
Figure 1-7	UT probe connection	13
Figure 1-8	PA probe connection	13
Figure 1-9	The main controls	14
Figure 1-10	Functions on MX2 keypad	17
Figure 1-11	Functions on SX keypad	19
Figure 1-12	OmniScan MX2: Example pop-up button for the Gain function key	21
Figure 1-13	Virtual keyboard with numeric keypad (<i>left</i>) or without (<i>right</i>)	23
Figure 1-14	Example of selection zone overlaps	24
Figure 1-15	The visual reference on the gate	26
Figure 1-16	The power indicator	27
Figure 1-17	OmniScan MX2 acquisition indicator light	28
Figure 1-18	OmniScan SX acquisition indicator light	28
Figure 2-1	Example screen image of the user interface	33
Figure 2-2	The title bar shortcut menus	34
Figure 2-3	The readings shortcut menu	41
Figure 2-4	The File Manager shortcut menu	42
Figure 2-5	The System shortcut menu	43
Figure 2-6	The acquisition parameters	43
Figure 2-7	The OmniScan MXU software window components	44
Figure 2-8	The menu hierarchy and identification syntax	45
Figure 2-9	Menu item groups	47
Figure 2-10	The reading fields and data screen	49
Figure 2-11	The parameters in the reading-field area	50
Figure 2-12	The information available in a reading field	50

Figure 2-13	The Gain value field	51
Figure 2-14	The Gain value field with an active reference	51
Figure 2-15	An example of the status indicators	52
Figure 2-16	OmniScan MX2 battery charge status example	54
Figure 2-17	OmniScan SX battery charge status example	54
Figure 2-18	The acquisition modes	56
Figure 2-19	Example of ultrasonic scan views	59
Figure 2-20	The RayTracing graphical elements	61
Figure 2-21	Example of RayTracing in the Setup wizard	62
Figure 2-22	Example of indications in the RayTracing view	64
Figure 2-23	The Multiple Group layouts	65
Figure 2-24	Example of multiple views with various rulers/scales	66
Figure 2-25	Three readings outlined in red, and one in purple	69
Figure 2-26	The purple color for ultrasound axis and mode button	71
Figure 2-27	Editing a parameter value with the increment/decrement step	74
Figure 2-28	The Compression feature symbol	75
Figure 3-1	File Name parameter button	82
Figure 3-2	Example of header note (<i>top</i>) and footer note (<i>bottom</i>) in a report	87
Figure 3-3	Example of Label and Content parameters in a report	87
Figure 4-1	Elements of a wizard step	92
Figure 4-2	The probe selection dialog box	94
Figure 4-3	The lists of wedges in the wedge selection dialog box	95
Figure 4-4	The probe reference point measurement	97
Figure 4-5	Probe-element pitch	97
Figure 4-6	Wedge angle	99
Figure 4-7	The primary offset measurement	99
Figure 4-8	The secondary offset	100
Figure 4-9	The height of the first element	100
Figure 4-10	Example of FFT curve and data for a 5 MHz probe	103
Figure 4-11	Example of a Probe Characterization section in a report	104
Figure 4-12	Example adaptor to connect UT probes to PA connector	106
Figure 5-1	Choice of parameters to calibrate in Select Calibration	110
Figure 5-2	Choice of parameters to calibrate — TOFD group	110
Figure 5-3	Example of reflector signals in a linear scan	115
Figure 5-4	Setting the gate on the second signal	116
Figure 5-5	Velocity indicator (V) in green after velocity calibration	116
Figure 5-6	Building the envelope for the wedge delay calibration	118
Figure 5-7	Wedge delay indicator (W) after calibration for UT group	119
Figure 5-8	The velocity (V) and wedge delay (W) indicators	123
Figure 5-9	Building the envelope for the wedge-delay calibration	125
Figure 5-10	Wedge delay indicator (W) after calibration for all focal laws	126
Figure 5-11	Building the envelope for the first angle/VPA	127

Figure 5-12	Wedge delay indicator (W_p) after calibration for 2 or 3 focal laws	128
Figure 5-13	Reference reflector signal before and after sensitivity calibration	129
Figure 5-14	Building the envelope for the sensitivity calibration	131
Figure 5-15	Sensitivity indicator (S) in green after calibration	132
Figure 5-16	Building the envelope for the sensitivity calibration of the first angle	134
Figure 5-17	Sensitivity indicator (S_i) after calibration for 2 or 3 focal laws	134
Figure 5-18	Calibration blocks for the DAC calibration	135
Figure 5-19	Set Gate A on Echo for a PA group	137
Figure 5-20	Calibration blocks for the TCG calibration	138
Figure 5-21	Set Gate A on Echo for a PA group	140
Figure 5-22	Set Gate A over the signal envelope	144
Figure 5-23	Example of DGS sizing curves	145
Figure 5-24	Completed DGS calibration status indicator	145
Figure 5-25	Completed AWS calibration status indicator	147
Figure 6-1	Choice of transmission/reception modes	152
Figure 6-2	An example of the PA-UT Weld (A-B-S) layout	155
Figure 6-3	The Export C-scan function	158
Figure 6-4	The Profile B-scan view	159
Figure 6-5	The Export function	160
Figure 6-6	The first point of the DAC curve	163
Figure 6-7	The DAC curve on the next signal	164
Figure 6-8	Verification of the DAC curve	165
Figure 6-9	An envelope with peaks at 80 %	167
Figure 6-10	Dialog box for choosing predefined readings	168
Figure 6-11	Example of an indication table	169
Figure 6-12	Selecting an entry in the indication table	170
Figure 6-13	Example of cursors	172
Figure 6-14	Example of a J-type weld geometry overlay	175
Figure 6-15	A green grid on the A-scan view	176
Figure 6-16	The palette on the right side of the A-scan view	177
Figure 6-17	Scan and Index reading fields	179
Figure 6-18	The Index Step function	181
Figure 7-1	The File Manager areas	184
Figure 7-2	The parent folder icon	185
Figure 7-3	The source pane highlighted	186
Figure 7-4	The Report folder path	188
Figure 7-5	The path for the custom report files	188
Figure 7-6	Gate A on the ball signal	195
Figure 7-7	The Local Area Connection Status dialog box (Windows XP)	198
Figure 7-8	The Properties dialog box for selected connection (Windows XP)	199
Figure 7-9	The Internet Protocol (TCP/IP) Properties dialog box (Windows XP)	200

Figure 7-10	The shortcut menu for the My Computer icon	201
Figure 7-11	The Computer Management dialog box (Windows XP)	201
Figure 7-12	The New User dialog box (Windows XP)	202
Figure 7-13	The Sharing tab of the Omniscan Properties dialog box (Windows XP) .	203
Figure 7-14	The Folder Options dialog box (Windows XP)	204
Figure 7-15	The Permissions for Omniscan dialog box (Windows XP)	205
Figure 7-16	The Select Users or Groups dialog box (Windows XP)	206
Figure 7-17	The Permissions for Omniscan dialog box (Windows XP)	207
Figure 7-18	The Security tab of the Omniscan Properties dialog box (Windows XP)	208
Figure 7-19	Security permissions for the Omniscan user	209
Figure 7-20	The computer name found in the System Properties dialog box	211
Figure 7-21	The Local Area Connection Status dialog box (Windows 7)	212
Figure 7-22	The Local Area Connection Properties dialog box (Windows 7)	213
Figure 7-23	The Internet Protocol Version 4 (TCP/IPv4) Properties dialog box (Windows 7)	214
Figure 7-24	The shortcut menu for the Computer icon (Windows 7)	215
Figure 7-25	The Computer Management dialog box (Windows 7)	215
Figure 7-26	The New User dialog box (Windows 7)	216
Figure 7-27	The Omniscan Properties dialog box (Windows 7)	217
Figure 7-28	The File Sharing dialog box (Windows 7)	218
Figure 7-29	Setting Permissions in the File Sharing dialog box (Windows 7)	219
Figure 7-30	The Advanced Sharing dialog box (Windows 7)	220
Figure 7-31	The Permissions for Omniscan dialog box (Windows 7)	220
Figure 7-32	The Select Users or Groups dialog box (Windows 7)	221
Figure 7-33	The Permissions for Omniscan dialog box (Windows 7)	222
Figure 7-34	Computer name in System Properties dialog box (Windows 7)	223
Figure 7-35	Interconnections between OmniScan MX2, MCDU, and computer	226
Figure 8-1	Wizard step components	239
Figure 8-2	Gain value display	245
Figure 8-3	The time intervals for the Acq Rate and PRF calculations (PA group)	248
Figure 8-4	The time intervals for Acq Rate in a multiple group configuration	248
Figure 8-5	The Acq Rate indicator in the reading field section	249
Figure 8-6	RF mode	252
Figure 8-7	HW+ rectification	252
Figure 8-8	HW- rectification	252
Figure 8-9	FW rectification	253
Figure 8-10	The scan offset	254
Figure 8-11	The beam index offset in a weld inspection example	254
Figure 8-12	The Gain value field with an active reference	256
Figure 8-13	The four s at the top of the screen	259
Figure 8-14	Dialog box for selecting measurement mode predefined readings	260
Figure 8-15	Reading field selection dialog box	261

Figure 8-16	Example for the AdBA reading	263
Figure 8-17	Example for the AdBr reading	264
Figure 8-18	Example for the A%, A [^] , and A/ readings	265
Figure 8-19	Diagram of the LA and SA readings	266
Figure 8-20	Diagram of the RA, PA, DA, ViA, and VsA readings	267
Figure 8-21	Example for the %(r), %(m), and %(m-r) readings	269
Figure 8-22	Example for the U(r), U(m), and U(m-r) readings	270
Figure 8-23	Example for the %U(r) reading	271
Figure 8-24	Illustration of the sizing category readings	273
Figure 8-25	Example for the E%, E [^] , and E-6dB readings	276
Figure 8-26	Example of corrosion inspection results, and T and ML readings	278
Figure 8-27	Example of an indication table	279
Figure 8-28	Example of an extra range set to "5% - 5%"	282
Figure 8-29	Cursor overlay example	284
Figure 8-30	A-scan envelope curve	285
Figure 8-31	Illustration of Curve Type interpolation choices	292
Figure 8-32	Available selection of gate parameters	296
Figure 8-33	A-scan synchronization types for gate I	298
Figure 8-34	Description of the source codes for gate A	305
Figure 8-35	The probe selection dialog box	307
Figure 8-36	The probe reference point measurement	309
Figure 8-37	The wedge reference point measurement	311
Figure 8-38	The primary offset measurement	311
Figure 8-39	Scan and index offsets	313
Figure 8-40	The parallel and nonparallel scan types	314
Figure 8-41	Sectorial scanning	316
Figure 8-42	Linear scanning with an angle (left) and at zero degrees (right)	317
Figure 8-43	Compound scanning	317
Figure 8-44	Quadrature and normal reading of the resolution	321
Figure 8-45	Computer name in System Properties dialog box (Windows XP)	332
Figure 8-46	FFT curve and data example	335
Figure 8-47	The Setup Configuration screen	337
Figure 8-48	The Scan Configuration parameters	339

List of Tables

Table 1	Main control functions for various contexts	15
Table 2	Key shortcuts for OmniScan MX2	17
Table 3	Key shortcuts for OmniScan SX	20
Table 4	OmniScan power indicator meaning	27
Table 5	Acquisition indicator meaning	29
Table 6	Choosing a menu from the menu list	29
Table 7	Choosing a submenu from a menu	30
Table 8	Choosing a parameter from a submenu	30
Table 9	Choosing a value from a parameter list	31
Table 10	Going back one level or canceling a selection	31
Table 11	Entering a value in an edit field	31
Table 12	Opening a shortcut menu	32
Table 13	The status indicators and their meanings	52
Table 14	Battery charge indicator variations	55
Table 15	Basic data views	58
Table 16	Ruler/scale colors	67
Table 17	Parameter button types	72
Table 18	Reflector, probe, and calibration block types	112
Table 19	RGB color codes	191
Table 20	Color of the Acq Rate indicator	249
Table 21	DAC curve properties relative to standards	290
Table 22	Example of the influence of the Measure parameter	299

Index

Numerics

0-reference point 97, 99
250 % mode 328

A

AC connection status 52
Accept key 14, 15
acquisition
 indicator 27
 meaning 29
 MX2 28
 SX 28
 modes 55
 analysis 56
 inspection 56
 rate indicator 50, 249
 rate, multiple group configuration 248
acquisition parameters adjustment 43
adding a group 105, 241
adjustment gain 51, 245, 256
administrator password 186, 330
Advanced Calculator application 226
Advanced submenu 255
alarm
 indicators 29
 output setting 161
 setting 161
Alarm submenu 301
amplitude, color palette 177
analog output, setting 162
Analog submenu 303
Aperture submenu 318
application setup 91

Area submenu 323

A-scan 58, 59
 overlay 174
A-Scan Settings submenu 280
ASME compliance 290
AWS 288
 calibration 146
 indicator 147
 parameters 293

B

batteries
 charge indicators 54
 charge status 52, 54, 55
Beam submenu 318
block types, calibration 111
boot, manual 330
B-scan 58, 60
buttons
 See also specific menu entries
 disabled control buttons 240
 menu 44
 parameter 44, 71, 72
 submenu 44

C

cable, phased array extension 12
calibration 109
 AWS 146
 block types 111
 DAC (distance-amplitude correction) 135
 DGS 141
 element selection 109
 encoder 148

- indicator
 - DGS 145
 - wedge delay 119
- TCG 138
- Calibration submenu 242
- calibration, ultrasonic 113
 - sensitivity
 - PA group 129
 - PA group, all focal laws 129
 - PA group, two or three focal laws 132
 - UT group 128
 - velocity 114
 - velocity and wedge delay 121
 - wedge delay 117, 123
 - all focal laws 124
 - two or three focal laws 126
 - wedge delay (WD) and probe center separation (PCS) 119
- Cancel key 14, 15
- caution notes
 - Complete folder 234
 - original report template files 189
- CAUTION signal word 2
- characterization, probe 194
- Characterize submenu 334
- charge, battery indicators 54
- clock setting 330
- colors
 - codes 190, 191
 - outlines 68
 - palette
 - changing amplitude 177
 - creating 190
 - file format 191
 - loading 228
 - rules 193
 - parameter buttons 70
 - gates 70
 - ultrasound units 70
 - reading fields 68
 - cursors 69
 - gates 69
 - ultrasound units 69
 - rulers/scales 67
 - scheme 81, 329
 - indoor 81
 - outdoor 81
- compliance
 - ASME 290
 - JIS 290
- components, window main 44
- Compression feature 75
 - parameter 75
 - symbol 75
- computer
 - configuration
 - Windows 7 211, 222
 - Windows XP 197, 209
 - data saving on 223
- configuration
 - computer
 - Windows 7 211, 222
 - Windows XP 197, 209
 - C-scan view parameters 176
 - DAC curve, manual 162
 - digital input (DIN) 80
 - focal laws 241
 - grid 175
 - indication table 169
 - linear DAC automatically 165
 - reading fields 168
 - reference gain 152
 - report 86
 - Save/Print key 64, 83
 - screen 171
 - TCG curve, manually 166
- configuration menus 48
 - File 49
 - Preferences 49
 - Scan 339
 - Setup 336
- Configuration submenu 316
- connection
 - OmniScan to a computer 196
 - OmniScan to a network 224
 - UT and PA probes
 - MX2 11
 - SX 12
- connectivity file opening 84
- contextual help 76
- controls, front panel
 - MX2 8

-
- SX 9
 - controls, main 14
 - Accept key 14, 15
 - Cancel key 14, 15
 - Scroll knob 14, 15
 - C-scan 58, 60
 - parameter configuration 176
 - ratio 1:1 availability 177
 - thickness C-scan configuration 155
 - cursors 25
 - active cursor blinking 173
 - color, reading fields 69
 - cursor and gate colors 69
 - data boxes 173
 - displaying 173
 - Cursors submenu 257
 - Curves Setup submenu 294
 - custom report template 187
 - D**
 - DAC (distance-amplitude correction) 287
 - calibration 135
 - configuring curve manually 162
 - converting DAC curve into TCG curve 167
 - curve(s) 135
 - DAC/TCG parameters 294
 - deleting curve point 164
 - example 164
 - verifying the curve 164
 - DANGER signal word 2
 - data
 - importing into TomoView 225
 - maximizing transfer rate
 - MCDU-02 226
 - TomoView 226
 - saving data
 - on computer 223
 - remotely 238
 - screen 44, 57
 - views 57
 - A-scan 58, 59
 - B-scan 58, 60
 - C-scan 58, 60
 - D-scan 60
 - end 60
 - End view 58
 - Ray Tracing 60
 - S-scan 58, 60
 - data file
 - opening 84
 - saving 82
 - Data Settings submenu 236
 - Data submenu 235, 324
 - date 50
 - setting 79, 330
 - decrement steps 74
 - defining
 - part to be inspected 101
 - probe 96
 - wedge 98
 - deleting
 - DAC curve point 164
 - group 241
 - indication 170
 - TCG curve point 166
 - depth color palette, changing 178
 - DGS 288
 - calibration 141
 - calibration indicator 145
 - parameters 292
 - digital input (DIN)
 - configuration 80
 - preferences 326
 - display
 - area shortcut menu 39
 - color scheme 81
 - indoor color scheme 81
 - multiple groups 35
 - outdoor color scheme 81
 - single group 35
 - Display menu 48
 - displaying
 - A-scan overlay 174
 - contextual help 76
 - cursors 173
 - gates 173
 - leg overlay 175
 - weld geometry overlay 175
 - wizard help 76
 - distance-amplitude correction (DAC) 287
 - dots
 - RayTracing, blue 63
-

RayTracing, red 63
D-scan 60
dual-element probe 12, 13

E

echo types 112
editing values 15
encoder
 calibration 148
 scan setup 179
Encoder submenu 320
End view 58, 60
envelope, reset 165, 167
Ethernet hub 226
extension cable, phased array 12
external storage preferences 331

F

feature, Compression 75
 symbol 75
FFT
 precision 104
 probe characterization 102, 334
 restrictions 334
fields, reading 44, 50
File Manager 183
 areas 183
 interface 183
 navigation 185
 parameters 333
File menu 49, 229
 Data 235
 Data Settings 236
 Image 235
 Report 231
 Setup
file types
 .html 88
 .jpg 238
 .law 226
 .opd 235
 .pal 190
 .rdt 225
 .xml 193
files
 logo.jpg 189, 190

management 81
opening 83
 any type of files 83
 connectivity files 84
 data files 84
 image files 84
 report files 85
 setup files 83
saving 82
 setup name 50
Focal Law menu 48, 315
 Aperture 318
 Beam 318
 Configuration 316
Focal Law submenu 241
folders, sharing creation
 Windows 7 217
 Windows XP 203
front panel controls
 MX2 8
 SX 9
full-screen mode 35
 activation 35
function keys 9, 19
 MX2 16
 editing parameters 21
 shortcuts 17
 shortcuts 20
 SX 19
 shortcuts 20

G

gain 50
 adjustment 51, 245, 256
 reference 51, 152, 245, 256
 value 50
gates 25
 colors
 parameter buttons 70
 reading fields 69
 displaying 173
 mode 50
 position 300
 preferences 328
Gates submenu 295
Gates/Alarms menu 48, 295

- Alarm 301
- Analog 303
- Gates 295
- Output 302
- Thickness 304
- general inspection parameters 151
- General submenu 244
- grid configuration 175
- group mode selection
 - MX2 106
 - SX 107
- Group/Probe & Part menu 47, 306
 - Characterize 334
 - Parts 314
 - Position 312
- groups
 - adding 105, 241
 - deleting 241
 - modifying 241
 - multiple group
 - display 36, 37, 106
 - layouts 37
 - removing 105
 - selecting 105
 - single group display 35
- H**
- hardware, connecting
 - Windows 7 197
 - Windows XP 197
- help
 - contextual 76
 - contextual, unavailability 76
 - Help key 76
 - preferences 332
 - wizards 76
 - deactivation 81
 - no help for Calibration wizard 77
- .html report file 88
- hub, Ethernet 226
- I**
- I/ parameter availability 156
- identifier, parameter 50
- image files opening 84
- Image submenu 235
- important information 1
- IMPORTANT signal word 3
- importing
 - law file 226
 - data into TomoView 225
 - OmniScan data into TomoView 225
- increment steps 74
 - MX2 16
 - SX 16
- indication table 279
 - adding an indication 170
 - configuring 169
 - deleting an indication 170
 - including in a report 171
 - modifying an indication 170
- Indication Table submenu 279
- indicators
 - AC connection 52
 - acquisition 27
 - meaning 29
 - MX2 28
 - SX 28
 - alarm 29
 - battery charge 52, 54, 55
 - Compression 75
 - power indicator 26, 27
 - PRF and Acq Rate 249
 - sensitivity calibration 132, 134
 - setup status 52
 - status 44, 51, 52
- indoor color scheme 81, 329
- input configuration, digital 80
- inspection 151
 - defining the part 101
 - menus 48
 - Display 48
 - Gates/Alarms 48
 - Measurements 48
 - Scan 48
 - Sizing 48
 - UT Settings 48
 - parameters, scan 50
 - technique 241
- Inspection submenu 319
- instruction manual 1
- Instrument submenu 328

- instrument, startup 9
 - Interleave 328
 - interpolation (OmniPC software) 328
 - introduction 5
 - J**
 - JIS compliance 290
 - K**
 - keyboard, virtual 22
 - keys
 - Accept 14, 15
 - Cancel 14, 15
 - function 9, 19
 - MX2 16
 - MX2, editing parameters 21
 - MX2, shortcuts 17
 - shortcuts 20
 - SX 19
 - SX, shortcuts 20
 - Help 76
 - Lower increment 16
 - Raise increment 16
 - Save/Print setting 85
- knob, Scroll 14, 15
- L**
- .law file 226
- .law file, importing 226
- layout 57
 - change 171
 - selection 36
- LEDs *See* indicators
- leg overlay, displaying 175
- length measurement units, selecting 80
- linear DAC 288
 - automatic configuration 165
 - parameters 294
- loading a custom palette 228
- logo
 - changing logo 189
 - logo.jpg file 189, 190
- Lower increment key 16
- M**
- main controls 14
 - Accept key 14, 15
 - Cancel key 14, 15
 - Scroll knob 14, 15
- management, file 81
- managers
 - File Manager 333
 - Probe Definition Manager 307
 - Wedge Definition Manager 310
- manual boot 330
- manual, instruction 1
- maximum scan speed 320
- MCDU-02, maximizing data transfer 226
- measurement units, length 80
- Measurements menu 48, 257
 - Cursors 257
 - Indication Table 279
 - Reading 259
- memory of peak values, reset 274
- menu, button 44
- menus
 - configuration 48
 - File 49, 229
 - Preferences 49, 326
 - Scan 339
 - Setup 336
 - inspection 48
 - Display 48
 - Gates/Alarms 48, 295
 - Measurements 48, 257
 - Scan 48, 319
 - Sizing 48, 286
 - UT Settings 48, 244
 - organization 46
 - setup definition 47
 - Focal Law 48, 315
 - Group/Probe & Part 47, 306
 - Wizard 47, 238
- menu-screen mode 35
- modes
 - acquisition 55
 - analysis 56
 - inspection 56
 - Pause key 325
 - Gate 50
 - P/C (pitch-catch) 152
 - P/E (pulse-echo) 152
 - TT (through-transmission) 152

modification
 group 241
 indication 170
 setup 93
 multiple group display 35, 106

N

navigation
 File Manager 185
 in the software 45, 46
 using hardware controls 29
 NDT SetupBuilder application 226
 network
 connection 196, 224
 preferences 330
 NOTE signal word 3
 notes, information signal words 3

O

Olympus technical support 4
 online help
See also help
 contextual 76
 wizards 76
 .opd data file 235
 opening 83
 any type of files 83
 connectivity files 84
 data files 84
 image files 84
 report files 85
 setup files 83
 Operator Mode submenu 288
 outdoor color scheme 81, 329
 outline colors 68
 output
 alarm output setting 162
 analog output setting 162
 Output submenu 302
 Overlay submenu 283
 overlays
 A-scan 174
 cursors 173
 gates 173
 grid 175
 leg 175

weld geometry 175

P

P/C (pitch-catch) mode 12, 13, 152
 P/E (pulse-echo) mode 12, 13, 152
 .pal files 190
 palette amplitude, color 177
 palettes
 changing depth color 178
 changing RF mode 178
 changing TOFD 178
 creating color palettes 190
 file format 191
 rules 193
 pan 25
 parameters
 buttons 44, 71, 72
 colors 70
 colors, gates 70
 colors, ultrasound units 70
 editing MX2 parameters 21
 general inspection parameters 151
 identifier 50
 ultrasonic parameter value 50
 units 50
 Part & Weld submenu 240
 part definition 101
 Parts submenu 314
 password, setting the administrator 186, 330
 PA-TOFD inspection setup 153
 peak values, reset memory of 274
 phased array extension cable 12
 pitch-catch (P/C) mode 152
 pop-up button 20
 Position submenu 312
 power
 battery charge indicator(s) 54
 indicator 26, 27
 preference settings 79
 Preferences menu 49, 326
 Instrument 328
 Setup 326
 Tools 332
 PRF
 definition 247
 for a multiple group configuration 248

- indicator 50, 249
 - maximizing the value 250
 - printing report from a computer 88
 - probe
 - adaptor 96
 - characterization 102, 194
 - connection
 - MX2 11
 - SX 12
 - definition 96
 - Definition Manager 307
 - selection 93
 - procedures *See specific procedure entries*
 - pulse
 - repetition frequency (PRF) indicator 50
 - width 247
 - pulse-echo (P/E) mode 152
 - pulser parameter setting 151
 - Pulser submenu 246
 - PW (pulse width) 247
- R**
- Raise increment key 16
 - RayTracing 60
 - dots
 - blue 63
 - red 63
 - hiding/showing in wizard 332
 - indication analysis 62
 - part inspection 62
 - setup definition 61
 - submenu 286
 - wizard 239
 - X with number in superscript 63
 - .rdt files 225
 - reading categories
 - code 274
 - corrosion 276
 - cursor 268
 - envelope 275
 - gate 262
 - generic codes 262
 - immersion 278
 - positioning 265
 - sizing 273
 - UT settings 279
 - reading fields 44, 50
 - area 49
 - colors 68
 - cursors 69
 - gates 69
 - ultrasound units 69
 - configuration 168
 - content 50
 - lists 259
 - Reading submenu 259
 - readings
 - Angle(Tmin) 277
 - Angle(TminZ) 278
 - colored contour 261
 - I(Tmin) 277
 - I(TminZ) 278
 - S(Tmin) 277
 - S(TminZ) 278
 - shortcut menu 40
 - thickness 277
 - Tmin 277
 - TminZ 277
 - receiver parameter setting 151
 - Receiver submenu 251
 - rectification 251
 - reference gain 51, 245, 256
 - configuration 152
 - reflector types 111
 - depth 112
 - radius 112
 - thickness 112
 - removal, group 105
 - report
 - changing logo 189
 - configuration 86
 - custom template 187
 - include user field 234
 - including an indication table 171
 - opening report files 85
 - printing from a computer 88
 - Report submenu 231
 - reset
 - envelope 165, 167
 - memory of peak values 274
 - RF mode palette 178
 - RGB color codes 191

- rulers 65
 - colors 67
- S**
- safety
 - signal words 2
 - symbols 1
- Save/Print key
 - configuration 64
 - setting 85
 - shortcut to save 232, 235
- scales 65, 67
- scan
 - inspection parameters 50
 - maximum speed 320
 - setup using encoders 179
 - types 113
- Scan menu 48, 319
 - Area 323
 - Data 324
 - Encoder 320
 - Inspection 319
 - Start 324
- scheme, color 81
 - indoor 81
 - outdoor 81
- screen
 - color and contrast 329
 - configuration 171
 - data 44
 - indoor use 329
 - outdoor use 329
 - preferences 329
 - touch screen usage 22
- screen modes
 - full-screen mode 35
 - menu-screen mode 35
 - unzoom 66
- screen shots, capturing 236
- Scroll knob 14, 15
- selection
 - element to calibrate 109
 - group 105
 - group mode
 - MX2 106
 - SX 107
 - layout 36
 - length measurement unit 80
 - probe and wedge 93
- sensitivity calibration
 - indicator 132, 134
 - PA group 129
 - all focal laws 129
 - two or three focal laws 132
 - UT group 128
- setting
 - administrator password 186
 - alarm output 161, 162
 - alarms 161
 - analog output 162
 - date and time 79
 - general inspection parameters 151
 - PA-TOFD inspection 153
 - preferences 79
 - Save/Print key 85
 - sizing functions 162
 - Tx/Rx mode 151
- setup 91
 - definition, menus 47
 - Focal Law 48
 - Group/Probe & Part 47
 - Scan Configuration 339
 - Setup Configuration 336
 - Wizard 47
 - modifying 93
 - RayTracing 61
 - status 52
- setup files
 - name 50
 - opening 83
 - saving 82
- Setup submenu 240, 326
- shared folder, creating
 - Windows 7 217
 - Windows XP 203
- shortcut keys 19
- shortcut menus 34
 - display area 39
 - readings 40
 - title bar 34
- shortcuts 20
 - MX2 17

- SX 20
- shutdown, software 10
- signal words
 - information notes 3
 - IMPORTANT 3
 - NOTE 3
 - TIP 3
 - safety 2
 - CAUTION 2
 - DANGER 2
 - WARNING 2
- single group display 35
- single-element probe 12, 13
- sizing functions 135
 - selection 286, 287
 - setting 162
- Sizing menu 48, 286
 - Curves Setup 294
 - Operator Mode 288
 - Type 287
 - Type Setup 290
- software
 - navigation 45, 46
 - shutdown 10
 - user interface 33
 - version ii, 50
- speed, maximum scan 320
- S-scan 58, 60
- Start submenu 324
- startup, instrument 9
- status indicators 44, 51, 52
- storage preferences 331
- submenu buttons 44
- submenus *See specific menu entries*
- support information, technical 4
- symbols, safety 1
- system preferences 330

T

- table, indication *See* indication table
- TCG 288
 - calibration 138
 - configuring curve manually 166
 - deleting a curve point 166
 - verifying the curve 166
- technical support 4

- technique, inspection 241
- thickness
 - C-scan, configuring 155
 - readings 277
 - T reading, dynamic 157
- Thickness submenu 304
- through-transmission (TT) mode 152
- time 50
 - setting 79, 330
- TIP signal word 3
- title bar shortcut menu 34
- TOFD (time-of-flight diffraction) palette 178
- TomoView
 - importing OmniScan data 225
 - maximizing data transfer 226
- Tools submenu 332
- touch screen
 - activation/deactivation 22
 - usage 22
- TT (through-transmission) mode 152
- Tx/Rx mode setup 151
- Type Setup submenu 290
- Type submenu 287
- types, reflector 111
 - depth 112
 - radius 112
 - thickness 112

U

- ultrasonic calibration 113
 - sensitivity
 - PA group 129
 - PA group, all focal laws 129
 - PA group, two or three focal laws 132
 - UT group 128
 - velocity 114
 - velocity and wedge delay 121
 - wedge delay 117, 123
 - all focal laws 124
 - two or three focal laws 126
 - wedge delay (WD) and probe center separation (PCS) 119
- ultrasonic parameter value 50
- ultrasound unit colors
 - parameter buttons 70
 - reading fields 69

- units
 - parameter units 50
 - preferences 329
 - setting length measurement 80
- unzoom 66
- use, intended 1
- user account creation
 - Windows 7 214
 - Windows XP 201
- user fields in report 234
- user interface, software 33
- UT Settings menu 48, 244
 - Advanced 255
 - General 244
 - Pulser 246
 - Receiver 251
- V**
- velocity
 - and wedge delay calibration 121
 - calibration 114
 - indicator 116
- verifying
 - DAC curve 164
 - TCG curve 166
- views
 - A-scan 58, 59
 - B-scan 58, 60
 - changing 171
 - C-scan 58, 60
 - data 57
 - D-scan 60
 - end 60
 - End view 58
 - Ray Tracing 60
 - S-scan 58, 60
- virtual keyboard 22
- W**
- WARNING signal word 2
- warning symbols
 - general 2
 - shock hazard 2
- warnings, general 3
- warranty information 4
- wedge
 - definition 98
 - Definition Manager 310
 - selection 93
- wedge delay (WD)
 - calibration 117, 123
 - all focal laws 124
 - two or three focal laws 126
 - indicator 119, 126
 - indicator for 2 or 3 laws 128
- wedge delay (WD) and probe center separation (PCS) calibration 119
- weld geometry overlay, displaying 175
- window main components 44
- Windows CE access 333
- wizard 76
 - deactivation 81
 - displaying help 76
 - help information 239
 - quitting 93
 - RayTracing in 239
 - step title 239
 - toggling display of help 81
- Wizard menu 47, 238
 - Calibration 242
 - Focal Law 241
 - Part & Weld 240
 - Setup 240
- X**
- .xml file 193
- Z**
- zero-reference point (0-reference point) 97, 99
- zoom 25

