



# OmniScan MXU Software

User's Manual

Software Version 4.4

DMTA-20072-01EN [Q1000010] — Rev. D  
September 2022

This instruction manual contains essential information on how to use this Evident 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.

EVIDENT CANADA, 3415, Rue Pierre-Ardouin, Québec (QC) G1P 0B3 Canada

Copyright © 2022 by Evident. All rights reserved. No part of this publication may be reproduced, translated, or distributed without the express written permission of Evident.

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. D

September 2022

Printed in Canada

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

---

---

# Table of Contents

---

<b>List of Abbreviations .....</b>	<b>11</b>
<b>Important Information — Please Read Before Use .....</b>	<b>13</b>
Intended Use .....	13
Instruction Manual .....	13
Safety Symbols .....	13
Safety Signal Words .....	14
Note Signal Words .....	15
Warnings .....	15
Warranty Information .....	16
Technical Support .....	16
<b>Introduction .....</b>	<b>17</b>
Other Relevant Evident Documents .....	17
<b>1. Instrument Overview .....</b>	<b>19</b>
1.1 Starting Up the OmniScan Instrument .....	21
1.2 Shutting Down the OmniScan MXU Software .....	22
1.3 Connecting UT and PA Probes on the OmniScan MX2 .....	23
1.4 Connecting UT and PA Probes on the OmniScan SX .....	24
1.5 Main Controls .....	26
1.6 OmniScan MX2 Increment Steps .....	28
1.7 OmniScan SX Increment Steps .....	28
1.8 OmniScan MX2 Function Keys .....	28
1.9 OmniScan SX Function Keys .....	31
1.10 Using OmniScan Function Keys as Software Function Shortcuts .....	32
1.11 Editing Parameters Using OmniScan Function Keys .....	33
1.12 Touch Screen .....	34
1.12.1 Entering Values on the Software Keyboard .....	34

1.12.2	Using the Touch Screen .....	36
1.13	Indicators .....	38
1.13.1	Power Indicator Light .....	38
1.13.2	Acquisition Indicator Light .....	39
1.13.3	Alarm Indicators .....	41
1.14	Navigating in the Menu Using Hardware Controls .....	41
<b>2.</b>	<b>Software User Interface .....</b>	<b>45</b>
2.1	Shortcut Menus .....	46
2.1.1	Title Bar Shortcut Menus .....	46
2.1.1.1	Menu-Screen and Full-Screen Modes .....	47
2.1.1.2	Single or Multiple Group Display (OmniScan MX2 Only) .....	47
2.1.1.3	Layout Selection .....	48
2.1.1.4	Group Selection Section (OmniScan MX2 Only) .....	49
2.1.1.5	Scan Section .....	49
2.1.1.6	Index Section .....	50
2.1.1.7	Angle/VPA Section .....	50
2.1.1.8	Display/Hide Options .....	51
2.1.2	Display Area Shortcut Menu .....	51
2.1.3	Shortcut Menus in the Reading-Field Area .....	52
2.1.4	Adjusting the Acquisition Parameters .....	55
2.2	Main Features of the Interface .....	56
2.3	OmniScan MXU Software Navigation .....	57
2.3.1	Navigating in the OmniScan MXU Software .....	58
2.3.2	Menu Organization .....	58
2.3.2.1	Menus Used to Define the Setup .....	59
2.3.2.2	Menus Used for Inspection .....	60
2.3.2.3	Menus Used for Configuration .....	60
2.4	Reading-Field Area .....	61
2.5	Gain .....	62
2.6	Status Indicators .....	63
2.7	Battery Charge Indicator(s) .....	66
2.8	Acquisition Modes .....	67
2.8.1	Inspection Mode .....	68
2.8.2	Analysis Mode .....	68
2.9	Data Screen .....	69
2.9.1	Scans, Views, and Layouts .....	69
2.9.2	RayTracing .....	72
2.9.2.1	Using RayTracing to Define the Setup .....	73
2.9.2.2	Using RayTracing to Inspect a Part .....	74
2.9.2.3	Using RayTracing to Analyze the Indications .....	74
2.9.2.4	Multiple Group Layouts in OmniPC .....	76

2.9.3	Rulers/Scales .....	77
2.10	Outline Colors .....	80
2.10.1	Reading-Field Colors .....	80
2.10.2	Parameter Button Colors .....	82
2.11	Parameter Buttons .....	83
2.12	Compression Feature .....	87
2.13	Online Help .....	87
2.13.1	Displaying Contextual Help .....	88
2.13.2	Displaying Online Help for Wizard Steps .....	88
<b>3.</b>	<b>General Procedures .....</b>	<b>91</b>
3.1	Setting Preferences .....	91
3.1.1	Setting the Date and Time .....	91
3.1.2	Selecting the Length Measurement Units .....	92
3.1.3	Configuring the Digital Inputs .....	92
3.1.4	Changing the Screen Color Scheme .....	93
3.1.5	Toggling the Wizard Help Screen .....	93
3.2	Managing Files .....	93
3.2.1	Saving Files .....	94
3.2.1.1	Saving a Setup .....	94
3.2.1.2	Saving a Data File .....	94
3.2.2	Opening Files .....	95
3.2.2.1	Opening Any Type of File .....	95
3.2.2.2	Opening Setup Files .....	95
3.2.2.3	Opening Connectivity Files .....	96
3.2.2.4	Opening Data Files .....	96
3.2.2.5	Opening Image Files .....	96
3.2.2.6	Opening Report Files .....	97
3.2.3	Setting the Save/Print Key .....	97
3.3	Producing Reports .....	98
3.3.1	Configuring a Report .....	98
3.3.2	Printing a Report from a Computer .....	100
<b>4.</b>	<b>Setup Procedures .....</b>	<b>103</b>
4.1	Creating the Application Setup with the Wizards .....	103
4.1.1	Configuring the Part and Weld .....	104
4.1.2	Modifying the Setup .....	105
4.2	Selecting the Probe and the Wedge .....	105
4.3	Defining a Probe .....	108
4.4	Defining a Wedge .....	110
4.5	Defining the Part to Be Inspected .....	113

4.6	Characterizing a Probe Using the FFT .....	114
4.7	Working with Multiple Groups (OmniScan MX2 Only) .....	117
4.8	Selecting the Group Mode on the OmniScan MX2 .....	118
4.9	Selecting the Group Mode on the OmniScan SX .....	118
<b>5.</b>	<b>Calibration Procedures .....</b>	<b>121</b>
5.1	Selecting the Element to Calibrate .....	121
5.2	Reflector Types .....	123
5.3	Scan Types .....	125
5.4	Ultrasonic Calibration .....	125
5.4.1	Ultrasonic Velocity Calibration .....	126
5.4.2	Wedge Delay Calibration for a UT Group .....	129
5.4.3	Wedge Delay and Probe Center Separation Calibration (TOFD Only) .....	131
5.4.4	Quick TOFD Calibration .....	132
5.4.5	Velocity and Wedge Delay Calibration for a UT Group .....	133
5.4.6	Wedge Delay Calibration for a PA Group .....	135
5.4.6.1	Calibrating the Wedge Delay for All Focal Laws .....	136
5.4.6.2	Calibrating the Wedge Delay for Two or Three Focal Laws in a PA Group .....	138
5.4.7	Sensitivity Calibration for a UT Group .....	140
5.4.8	Sensitivity Calibration for a PA Group .....	141
5.4.8.1	Calibrating the Sensitivity for All Focal Laws .....	141
5.4.8.2	Calibrating the Sensitivity for Two or Three Focal Laws .....	144
5.5	Sizing Calibration .....	147
5.5.1	DAC Calibration .....	147
5.5.2	TCG Calibration .....	150
5.5.3	DGS Calibration .....	153
5.5.4	AWS Calibration .....	158
5.6	Encoder Calibration .....	160
<b>6.</b>	<b>Inspection Procedures .....</b>	<b>163</b>
6.1	Setting the General Inspection Parameters .....	163
6.2	Setting the Tx/Rx Mode (UT Group Only) .....	163
6.3	Configuring the Reference Gain .....	164
6.4	Setting Up a PA-TOFD Inspection (OmniScan MX2 Only) .....	165
6.5	Configuring a Thickness C-Scan .....	167
6.6	Using the Export C-Scan Function .....	169
6.7	Using the Profile B-Scan View .....	170
6.8	Using the Export B-Scan Function (OmniPC Only) .....	171
6.9	Setting Alarms and Outputs .....	172

6.9.1	Setting an Alarm .....	173
6.9.2	Setting an Alarm Output .....	173
6.9.3	Setting an Analog Output (OmniScan MX2 Only) .....	174
6.10	Setting Sizing Functions .....	174
6.10.1	Manually Configuring the DAC Curve .....	174
6.10.2	Automatically Configuring the Linear DAC Curve .....	177
6.10.3	Manually Configuring the TCG Curve .....	178
6.11	Setting Measurements .....	179
6.11.1	Configuring Reading Fields .....	180
6.11.2	Configuring and Building an Indication Table .....	181
6.12	Configuring the Screen .....	183
6.12.1	Changing the Current Layout and the Current View .....	183
6.12.2	Displaying the Cursors .....	184
6.12.3	Displaying the Gates .....	185
6.12.4	Displaying Various A-Scan Related Overlays .....	186
6.12.5	Displaying the Weld Geometry Overlay .....	186
6.12.6	Displaying the Leg Overlay .....	187
6.12.7	Configuring the Grid .....	187
6.12.8	Configuring the View Parameters .....	188
6.12.9	Changing the Color Palettes .....	189
6.13	Performing an Inspection with an Encoder .....	191
6.14	Indexing a Scan .....	193
<b>7.</b>	<b>Complementary Procedures .....</b>	<b>195</b>
7.1	Using the File Manager .....	195
7.1.1	File Manager Interface .....	195
7.1.2	File Manager Navigation .....	197
7.2	Setting the Administrator Password .....	198
7.3	Creating a Custom Report Template .....	199
7.4	Changing the Company Logo .....	201
7.5	Creating a Color Palette .....	202
7.5.1	Color Palette File Format .....	203
7.5.2	Color Palette Rules .....	205
7.6	Characterizing a Probe .....	206
7.7	Connecting the OmniScan MX2 Directly to a Computer .....	208
7.7.1	Connecting the Hardware .....	208
7.7.2	Configuring the Computer Network Connection — Windows XP .....	209
7.7.3	Creating a User Account on a Computer — Windows XP .....	212
7.7.4	Creating a Shared Folder on a Computer — Windows XP .....	215
7.7.5	Configuring OmniScan MX2 Network Connection — Windows XP .....	221
7.7.6	Configuring the Computer Network Connection — Windows 7 .....	223

7.7.7	Creating a User Account on a Computer — Windows 7 .....	226
7.7.8	Creating a Shared Folder on a Computer — Windows 7 .....	229
7.7.9	Configuring the OmniScan MX2 — Windows 7 .....	234
7.7.10	Saving OmniScan Data on a Computer .....	235
7.8	Connecting the OmniScan MX2 to a Network .....	236
7.9	Importing OmniScan Data into TomoView .....	237
7.10	Transferring Data — OmniScan MX2 with MCDU-02 and TomoView .....	238
7.11	Importing a .law File into the OmniScan Instrument (PA Group Only) .....	238
7.12	Loading a Custom Color Palette .....	240

## **8. Menu Description ..... 241**

8.1	File Menu .....	241
8.1.1	Setup Submenu .....	241
8.1.2	Report Submenu .....	243
8.1.2.1	Open/Save Category .....	243
8.1.2.2	Format Category .....	244
8.1.2.3	User Fields Category .....	246
8.1.3	Data Submenu .....	247
8.1.4	Image Submenu .....	247
8.1.5	Data Settings Submenu .....	248
8.2	Wizard Menu .....	250
8.2.1	Part & Weld Submenu .....	252
8.2.2	Setup Submenu .....	252
8.2.3	Calibration Submenu .....	254
8.3	UT Settings Menu .....	256
8.3.1	General Submenu .....	256
8.3.2	Pulser Submenu .....	258
8.3.3	Receiver Submenu .....	263
8.3.4	Beam Submenu .....	265
8.3.5	Advanced Submenu .....	267
8.4	Measurements Menu .....	269
8.4.1	Cursors Submenu .....	269
8.4.2	Reading Submenu .....	271
8.4.2.1	Generic Reading Codes .....	274
8.4.2.2	Gate Category Reading .....	274
8.4.2.3	Positioning Category Reading .....	277
8.4.2.4	Cursor Category Readings .....	280
8.4.2.5	Sizing Category Readings .....	285
8.4.2.6	Code Category Reading .....	286
8.4.2.7	Envelope Category Readings .....	287
8.4.2.8	Corrosion and Composite Category Readings .....	288
8.4.2.9	Immersion Category Readings .....	290

8.4.2.10	UT Settings Category Readings .....	291
8.4.3	Indication Table Submenu .....	291
8.5	Display Menu .....	292
8.5.1	A-Scan Settings Submenu .....	292
8.5.2	Overlay Submenu .....	295
8.5.3	RayTracing Submenu .....	298
8.6	Sizing Menu .....	298
8.6.1	Type Submenu .....	299
8.6.2	Operator Mode Submenu .....	300
8.6.3	Type Setup Submenu .....	302
8.6.3.1	Parameters for DAC, Linear DAC, and TCG Curves .....	302
8.6.3.2	Parameters for DGS .....	304
8.6.3.3	Parameters for AWS Curves .....	305
8.6.4	Curves Setup Submenu .....	306
8.6.4.1	Parameters for DAC and TCG Curves .....	306
8.6.4.2	Parameters for Linear DAC Curve .....	306
8.7	Gates/Alarms Menu .....	307
8.7.1	Gates Submenu .....	307
8.7.1.1	Gate Position .....	312
8.7.1.2	Saturated Gate .....	312
8.7.2	Alarm Submenu .....	313
8.7.3	Output Submenu .....	314
8.7.4	Analog Submenu (OmniScan MX2 Only) .....	315
8.7.5	Thickness Submenu .....	316
8.8	Group/Probe & Part Menu .....	318
8.8.1	Probe & Wedge Submenu .....	318
8.8.1.1	Probe Definition Manager .....	319
8.8.1.2	Wedge Definition Manager .....	322
8.8.2	Position Submenu .....	324
8.8.3	Parts Submenu .....	326
8.9	Focal Law Menu (PA Group Only) .....	327
8.9.1	Configuration Submenu .....	328
8.9.2	Aperture Submenu .....	330
8.9.3	Beam Submenu .....	330
8.10	Scan Menu .....	331
8.10.1	Inspection Submenu .....	331
8.10.2	Encoder Submenu .....	332
8.10.3	Area Submenu .....	335
8.10.4	Data Submenu .....	336
8.10.5	Start Submenu .....	336
8.11	Preferences Menu .....	338
8.11.1	Setup Submenu .....	338

- 8.11.1.1 DIN Category ..... 338
- 8.11.1.2 Gates Category ..... 340
- 8.11.1.3 250 % Mode Category ..... 340
- 8.11.1.4 Interpolation Category ..... 340
- 8.11.1.5 Interleave Category ..... 340
- 8.11.2 Instrument Submenu ..... 340
  - 8.11.2.1 Units Category ..... 341
  - 8.11.2.2 Screen Category ..... 341
  - 8.11.2.3 System Category ..... 342
  - 8.11.2.4 Network Settings Category (OmniScan MX2 Only) ..... 342
  - 8.11.2.5 External Storage Category (OmniScan MX2 Only) ..... 343
  - 8.11.2.6 Help Category ..... 344
- 8.11.3 Tools Submenu ..... 344
- 8.11.4 FFT Submenu ..... 346
- 8.12 Setup Configuration Menu ..... 348
- 8.13 Scan Configuration Menu ..... 351

**List of Figures ..... 353**

**List of Tables ..... 359**

---

## 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 Evident 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.

---

<b>IMPORTANT</b>
------------------

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

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

The Evident 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 Evident for assistance with the damage claim and equipment replacement, if necessary.

This instruction manual explains the proper operation of your Evident 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, Evident 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.

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

## Technical Support

Evident 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 on the Evident Scientific Web site.

---

# 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.

---

<b>NOTE</b>
-------------

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 Evident Documents

Several other Evident 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

---

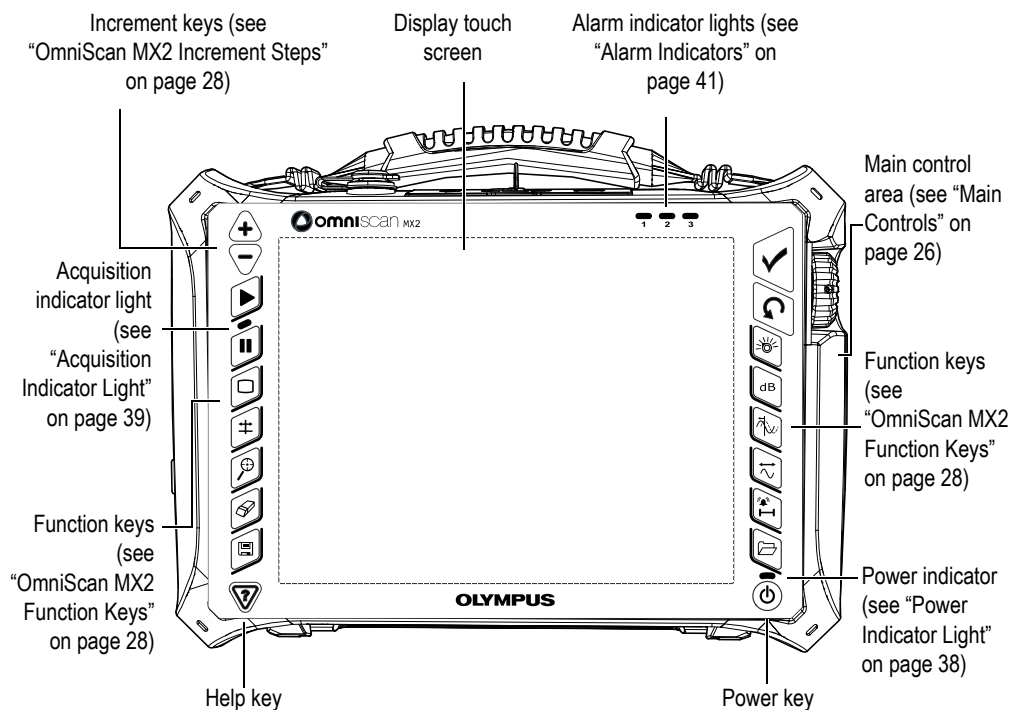
The OmniScan instrument features front panel controls for easy and efficient operation of the OmniScan MXU software. Figure 1-1 on page 20 and Figure 1-2 on page 21 show the OmniScan MX2 and OmniScan SX front panels and the available controls and indicators. The OmniScan instrument allows you to connect a USB keyboard and USB mouse to extend its user interface.

---

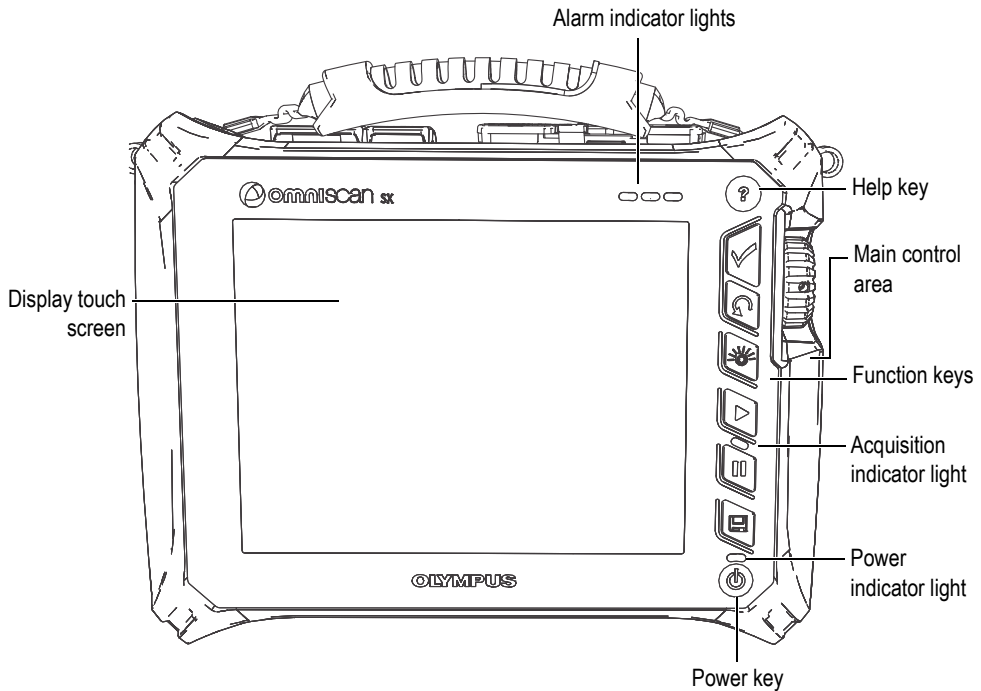
<b>NOTE</b>
-------------

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 Evident 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**.

---

<b>NOTE</b>
-------------

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.

---

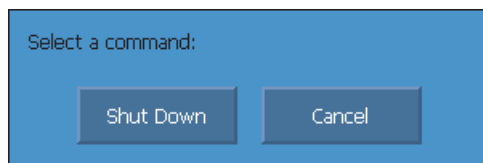
## 1.2 Shutting Down the OmniScan MXU Software

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.

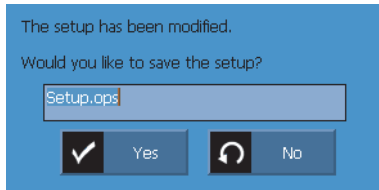
The “Select a command” message appears (see Figure 1-3 on page 22).



**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 23).



**Figure 1-4 Saving the setup**

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

---

<b>NOTE</b>
-------------

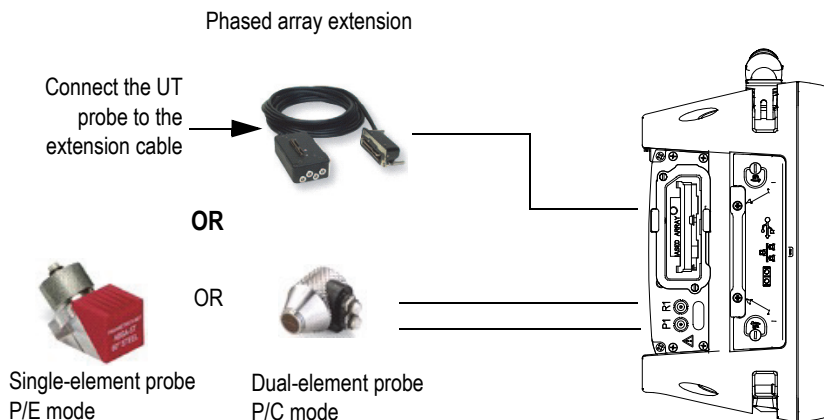
You can also turn off the OmniScan by pressing and holding the Power key for ten seconds. However, your setup will NOT be saved.

---

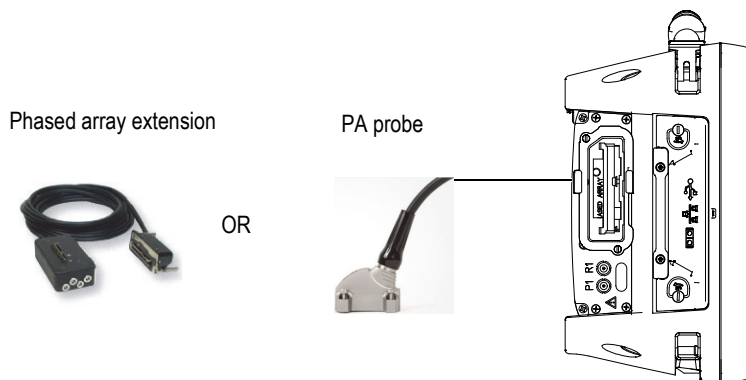
## 1.3 Connecting UT and PA Probes on the OmniScan MX2

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 24. 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 24.



**Figure 1-5 Requirements for UT probe connection**



**Figure 1-6 Requirements for PA probe connection**

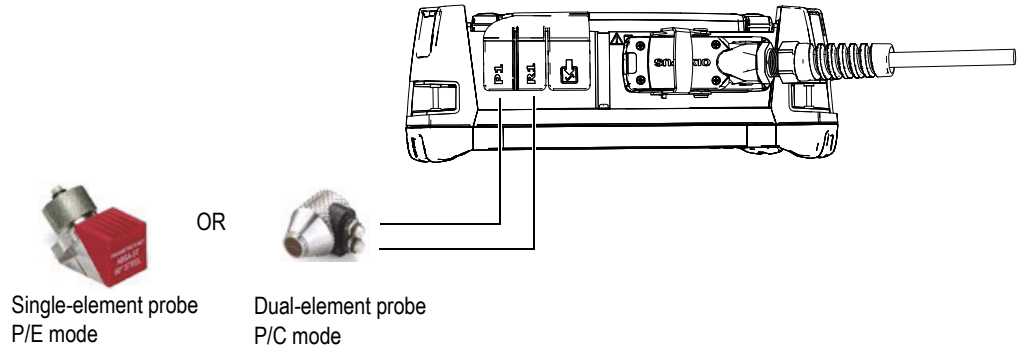
## 1.4 Connecting UT and PA Probes on the OmniScan SX

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 25. Use a dual element probe for the pitch-catch (P/C) mode.

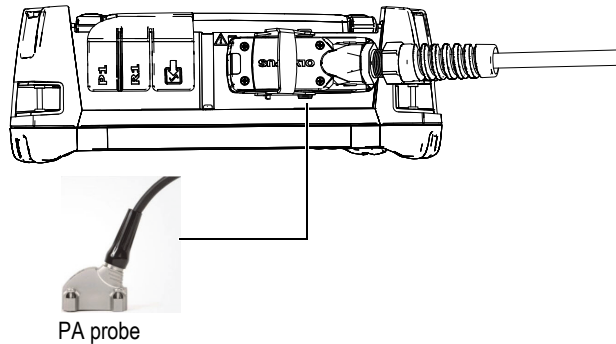
Connect a phased array probe to the OmniScan SX PA connector, as shown in Figure 1-8 on page 25.

**NOTE**

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



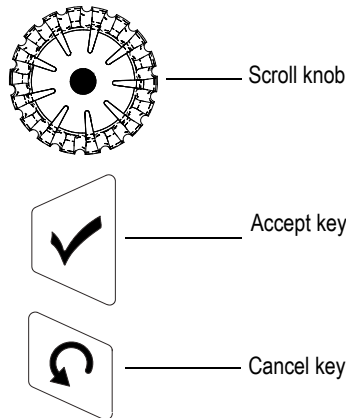
**Figure 1-7 UT probe connection**



**Figure 1-8 PA probe connection**

## 1.5 Main Controls

The three main controls shown in Figure 1-9 on page 26 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 27.

**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

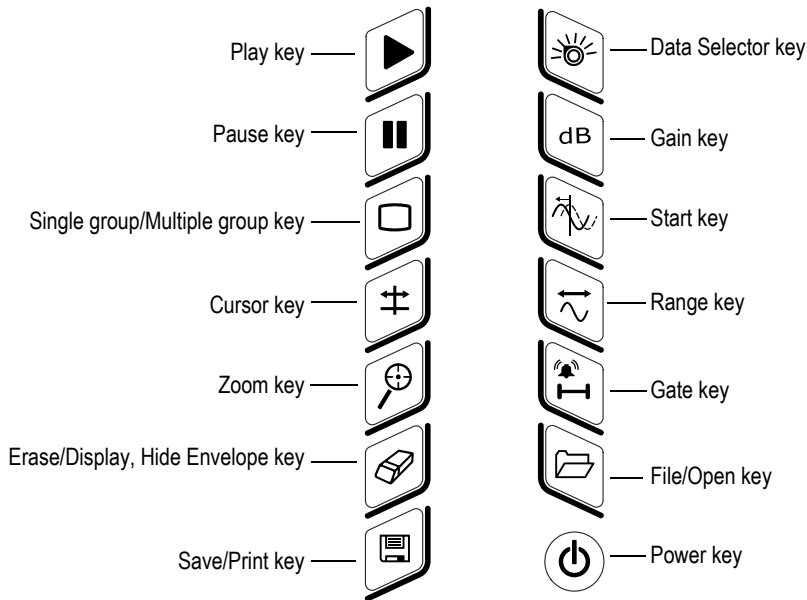
When using the OmniScan SX, increment steps can only be changed when the selected numeric parameter is in edit mode.

### 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 + $\Delta$  or - $\Delta$  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 29).












**Figure 1-10 Functions on MX2 keypad**

Table 2 on page 29 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 <b>Scan &gt; Start</b> 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 <b>Measurements &gt; Cursors &gt; Category</b> .
	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 <b>File &gt; Data Settings</b> menu.	N/A
	<p>In inspection mode:</p> <ul style="list-style-type: none"> <li>When a PA group is selected, used to select the current focal law.</li> <li>N/A when a UT group is selected.</li> </ul> <p>In analysis mode:</p> <ul style="list-style-type: none"> <li>When a PA group is selected, used to select the angle, scan, and index positions (if applicable).</li> </ul> <p>When a UT group is selected, used to select the scan and index positions (if applicable).</p>	Used to change the current group.
	Used to set the signal gain.	Brings up the <b>UT Settings</b> menu.
	Used to set the A-scan screen starting position.	Brings up the <b>UT Settings</b> menu.
	Used to set the A-scan screen range.	Brings up the <b>UT Settings</b> menu.
	Used to set the parameters for the displayed gates.	Brings up the <b>Gates/Alarm</b> 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 31).

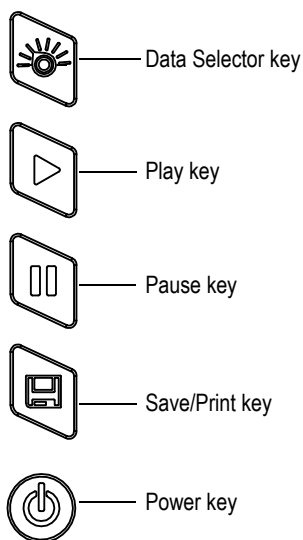




**Figure 1-11 Functions on SX keypad**

Table 3 on page 32 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 <b>Scan &gt; Start</b> 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 <b>File &gt; Data Settings</b> menu.	N/A
	<p>In inspection mode:</p> <ul style="list-style-type: none"> <li>When a PA group is selected, used to select the current focal law.</li> <li>N/A when a UT group is selected.</li> </ul> <p>In analysis mode:</p> <ul style="list-style-type: none"> <li>When a PA group is selected, used to select the angle, scan, and index positions (if applicable).</li> <li>When a UT group is selected, used to select the scan and index positions (if applicable).</li> </ul>	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 33). 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.

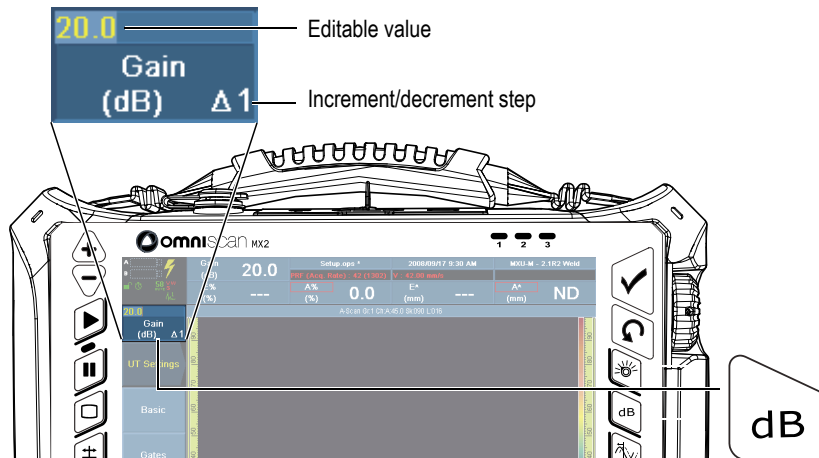


Figure 1-12 OmniScan MX2: Example pop-up button for the Gain function key

## 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 29 and Table 3 on page 32 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.

---

<b>NOTE</b>
-------------

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 35).
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"><li>◆ To change the incrementation, press the Raise increment or Lower increment keys.</li></ul>	<ul style="list-style-type: none"><li>◆ To change the incrementation:<ul style="list-style-type: none"><li>a) Tap the parameter twice, or press the Accept key twice to open the software keyboard.</li><li>b) Use the +Δ or -Δ buttons to increase or decrease the increment (select from the predefined list of increments).</li></ul></li></ul>

- 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)



## 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

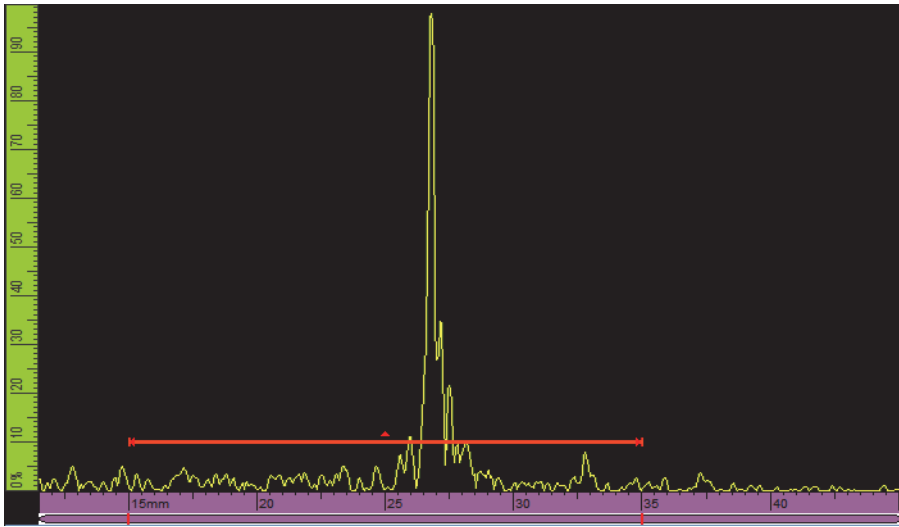
1. To open the gate **Start** parameter pop-up button in edit mode, tap the left end of the gate.
2. To open the gate **Threshold** parameter pop-up button in edit mode, tap the middle of the gate.
3. To open the gate **Width** parameter pop-up button in edit mode, tap the right end of the gate.

---

<b>NOTE</b>
-------------

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 38).

---



**Figure 1-15 The visual reference on the gate**

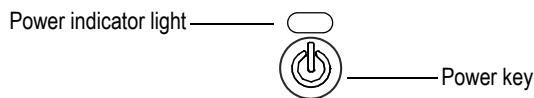
## 1.13 Indicators

There are three types of indicators on the OmniScan instrument front panel (see Figure 1-1 on page 20 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 39). The power indicator color identifies the power status of the OmniScan instrument (see Table 4 on page 39).



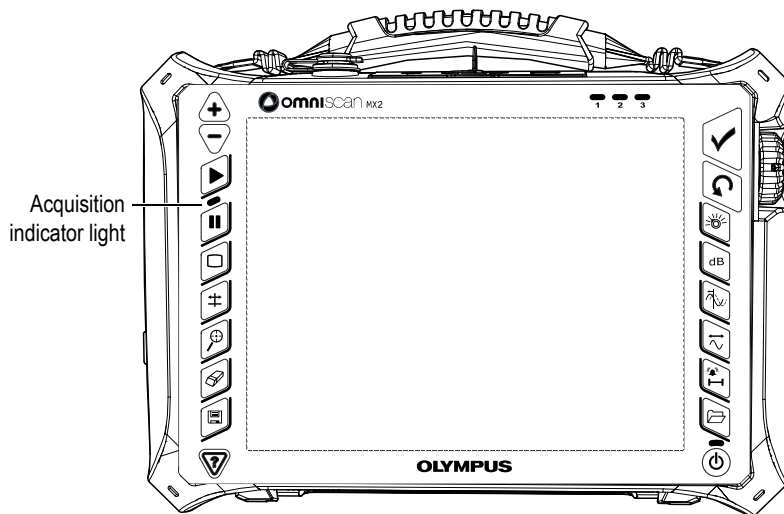
**Figure 1-16 The power indicator**

**Table 4 OmniScan power indicator meaning**

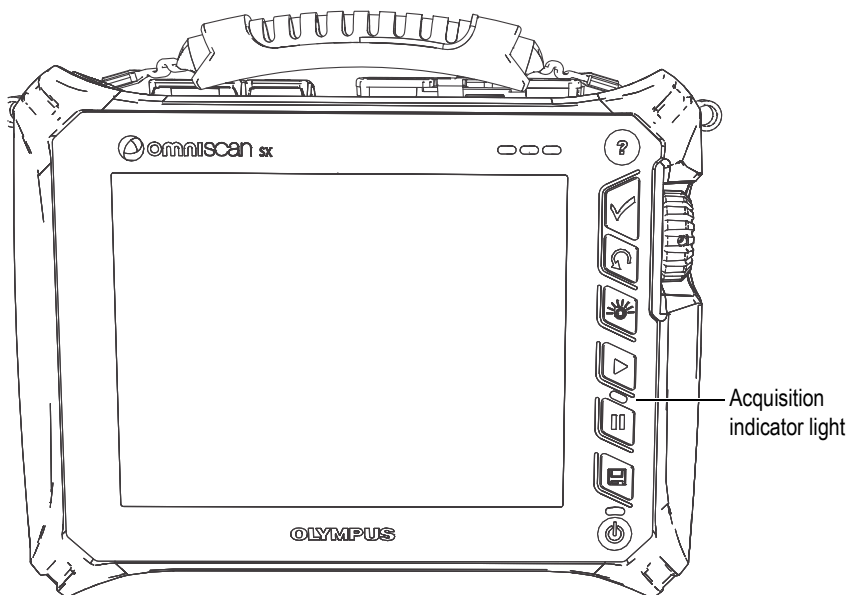
Power indicator color	Power status	Description
Off	Off	Normal
Green	On	<ul style="list-style-type: none"> <li>Normal</li> <li>Battery is charging.</li> </ul>
Flashing orange	Off	Battery is charging.
Orange	Off	Battery charge complete
Flashing red	On/Off	Critical factor (excessive temperature, very weak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

### 1.13.2 Acquisition Indicator Light

The acquisition indicator light is located below the Play key (see Figure 1-17 on page 40 or Figure 1-18 on page 40). The acquisition indicator color identifies the acquisition mode of the OmniScan instrument (see Table 5 on page 41).



**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"> <li>Paused acquisition</li> <li>Previously acquired data is available for analysis.</li> </ul>	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 20 or Figure 1-2 on page 21). An alarm indicator turns red when the associated alarm condition is encountered. See “Gates/Alarms Menu” on page 307 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 41 to Table 12 on page 44).

**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 27.
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 62)
Mouse	Right-click the appropriate area.

---

<b>IMPORTANT</b>
------------------

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 56. The shortcut menus that are available in full-screen mode are detailed in “Shortcut Menus” on page 46.

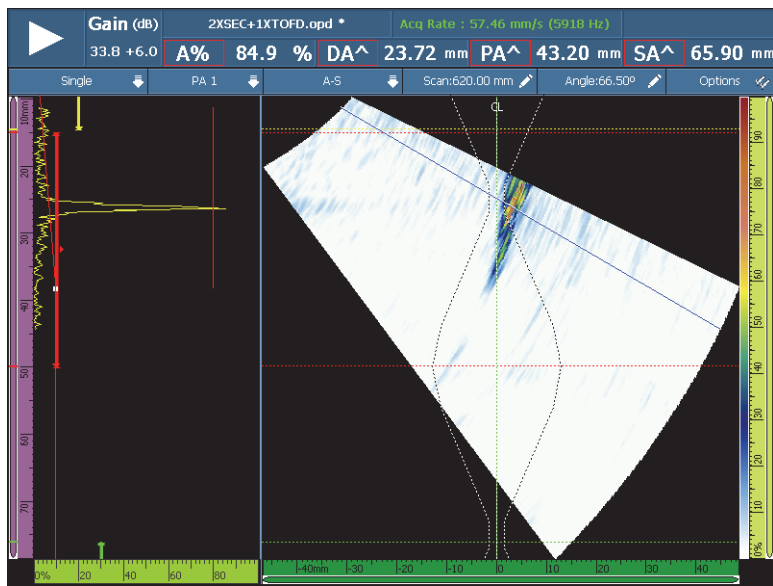


Figure 2-1 Example screen image of the user interface

## 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 46) 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

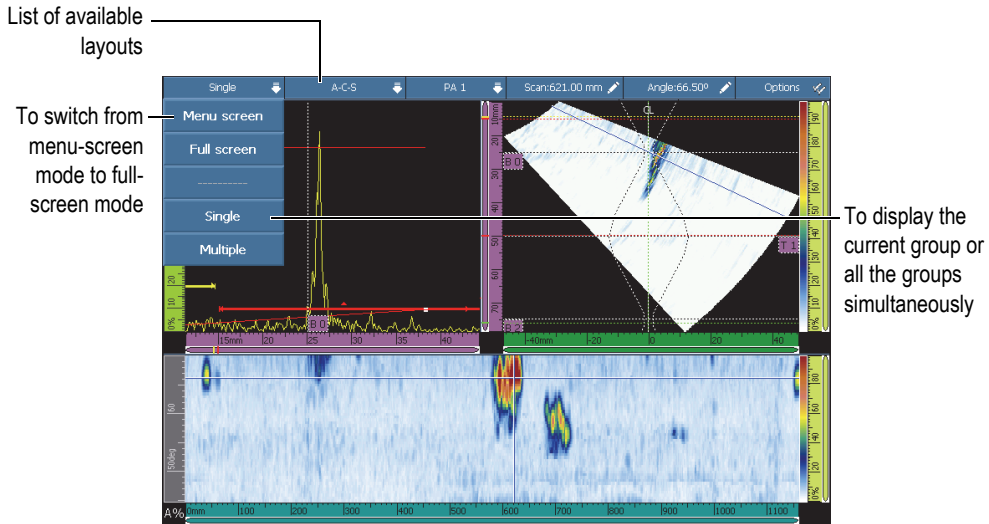


Figure 2-2 The title bar shortcut menus

### 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 46, 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**.

---

<b>NOTE</b>
-------------

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**.

---

<b>NOTE</b>
-------------

- 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.

---

<b>NOTE</b>
-------------

For more details about layouts and views, see “Scans, Views, and Layouts” on page 69.

---

#### 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.
- 

<b>NOTE</b>
-------------

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 46).  
A list of available groups appears.
  2. Select the desired group from the list.
- 

<b>TIP</b>
------------

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 46).
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 332 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 351.

#### 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 46).
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 332 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 351.

#### 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 46).
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 292 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 348.

#### 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 46).  
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)

---

<b>NOTE</b>
-------------

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 53).



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 54).

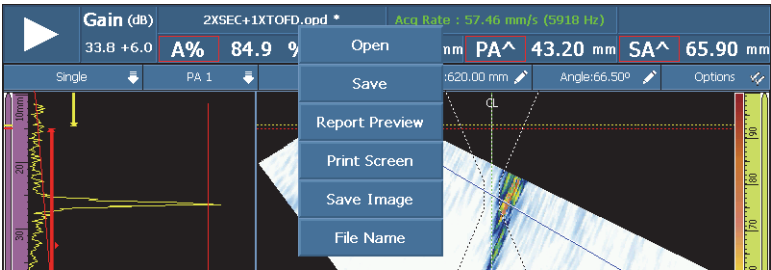


Figure 2-4 The File Manager shortcut menu

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 55).

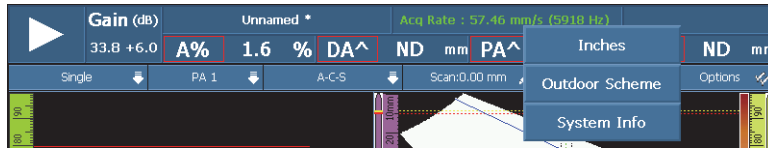


Figure 2-5 The System shortcut menu

- 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 55, 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 56 shows the OmniScan MXU software user interface, and identifies its main components.

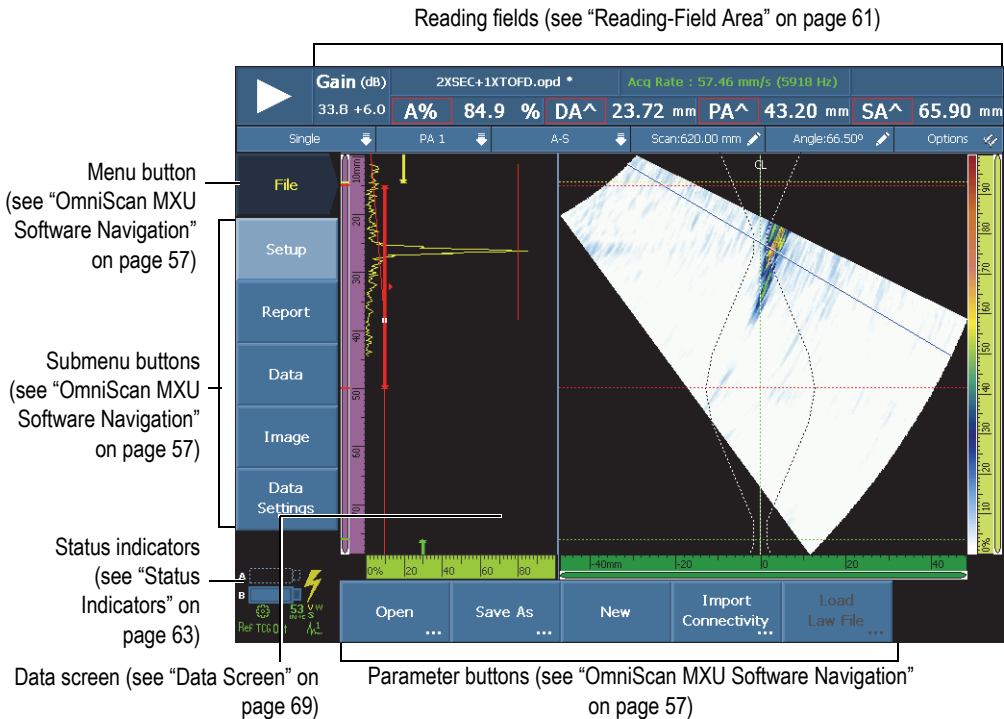


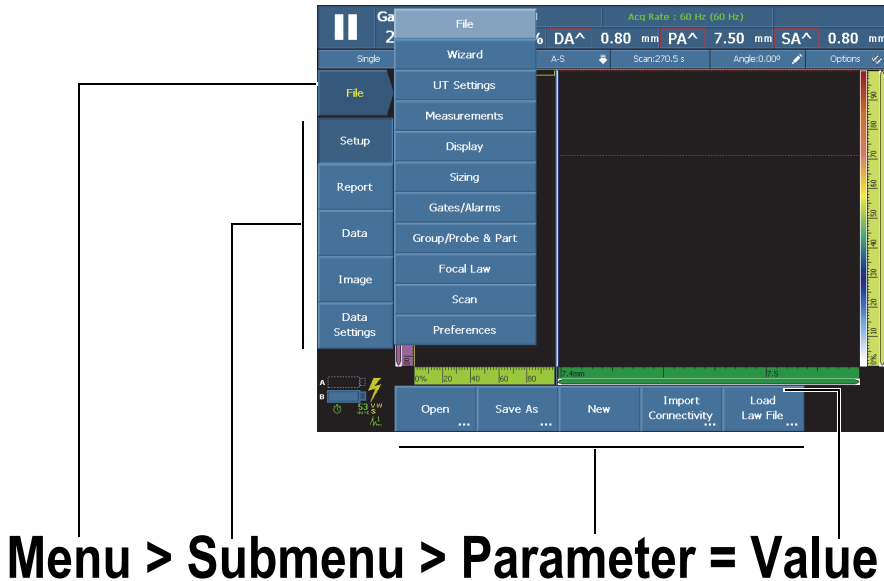
Figure 2-7 The OmniScan MXU software window components

## 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 57 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 58 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 57).

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 59).

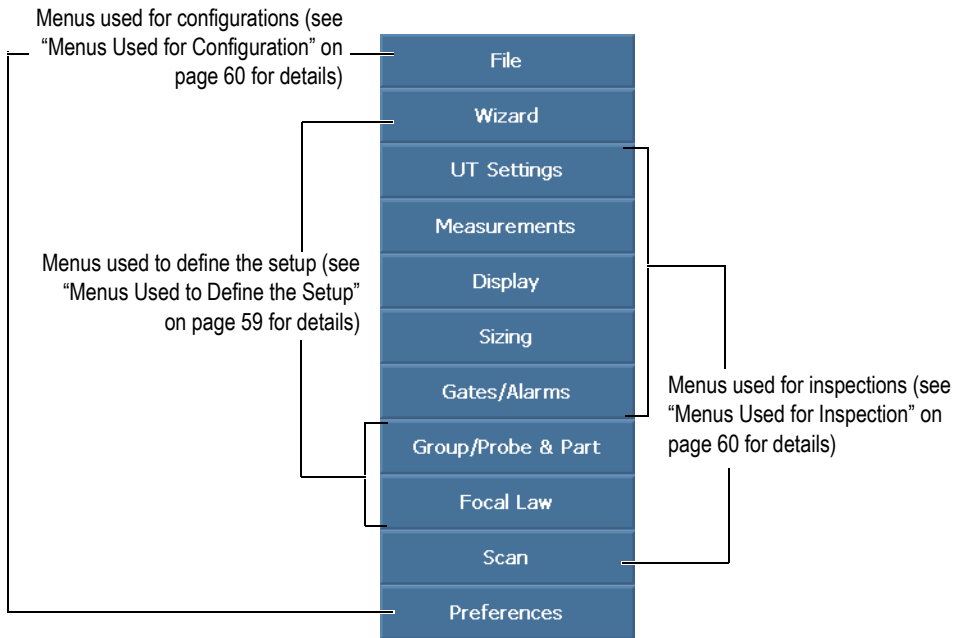


Figure 2-9 Menu item groups

### 2.3.2.1 Menus Used to Define the Setup

The following menus are used to define the setup (see "Setup Procedures" on page 103 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 318 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 327 for a complete description.

### **2.3.2.2 Menus Used for Inspection**

The following menus are used during inspections (see “Inspection Procedures” on page 163 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 256 for a complete description).

#### **Measurements**

This menu contains the parameters related to various measurement tools (see “Measurements Menu” on page 269 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 298 for a complete description.

#### **Gates/Alarms**

This menu contains parameters for configuring gates, alarms, and output signals (see “Gates/Alarms Menu” on page 307 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 331 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 91 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 241 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 338 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 61).

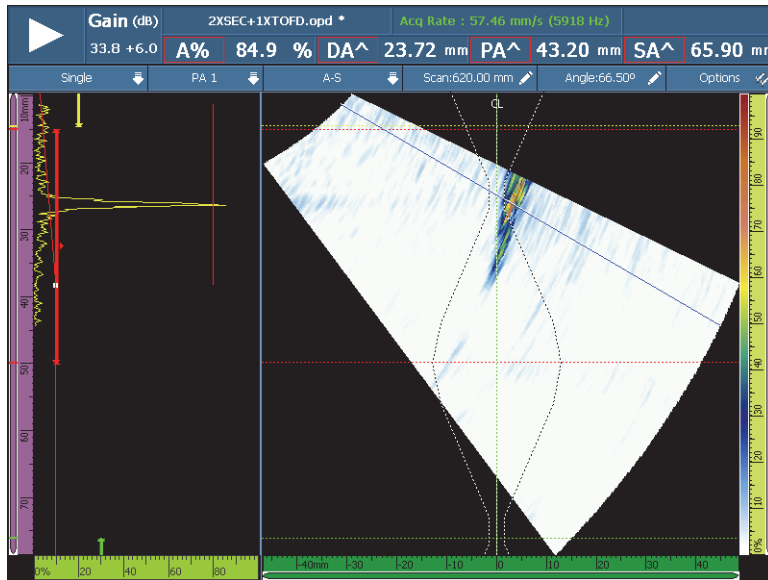


Figure 2-10 The reading fields and data screen

Figure 2-11 on page 62 identifies the various parameters and reading fields available, and lists sections in this manual that provide more information. Figure 2-12 on page 62 identifies the information available in a reading field.

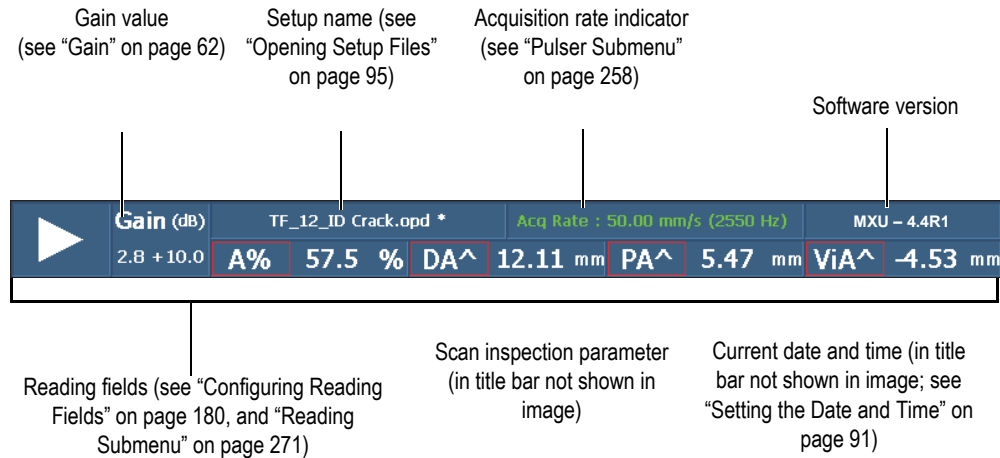


Figure 2-11 The parameters in the reading-field area

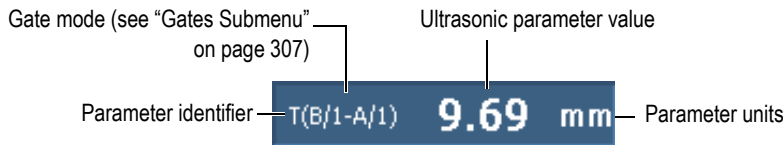
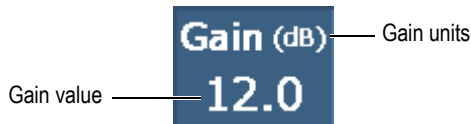


Figure 2-12 The information available in a reading field

## 2.5 Gain

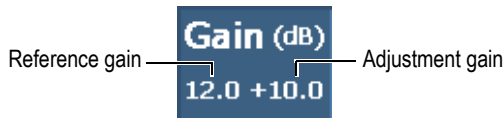
The gain value that is applied to the signal is an important parameter. The gain value applied to all focal laws appears in the upper-left corner of the reading-field area (see Figure 2-11 on page 62). Figure 2-13 on page 63 shows the information displayed in the **Gain** value field.



**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 63). 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 267 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 64 shows an example of the status indicators.

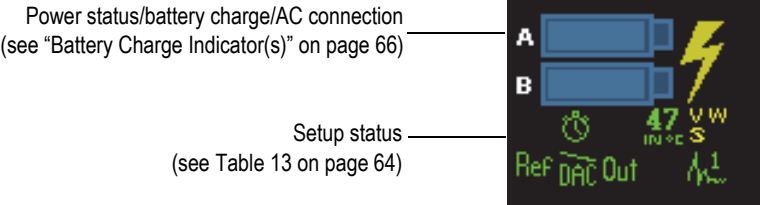



























Figure 2-15 An example of the status indicators

Table 13 on page 64 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: <b>Display &gt; A-Scan Settings &gt; Data Source = All Laws</b> (see "Data Source" on page 293).
	Tracking A-scan mode is enabled as follows: <b>Display &gt; A-Scan Settings &gt; Data Source = Highest or Thinnest</b> (see "Data Source" on page 293).
	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 299).
	The linear distance-amplitude correction (DAC) curve is enabled (see "Type Submenu" on page 299).
	The number of data points on the inspected area exceeds the number of available pixels (see "Compression Feature" on page 87).
	The time-corrected gain (TCG) is enabled (see "Type Submenu" on page 299).

**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 306).
	The distance-gain sizing (DGS) is enabled (see “Type Submenu” on page 299).
	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.
 (yellow)	The sensitivity is not calibrated.
 (green)	The sensitivity is calibrated.
 (green)	The sensitivity is calibrated for two or three selected angles/VPA, and interpolated in between the calibrated angles/VPA.
 (yellow)	The TOFD is not calibrated.
 (green)	The TOFD is calibrated.
 (yellow)	The ultrasonic velocity is not calibrated.
 (green)	The ultrasonic velocity is calibrated.
 (yellow)	The wedge delay is not calibrated.
 (green)	The wedge delay is calibrated.
 (green)	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 66 or Figure 2-17 on page 66).



**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).




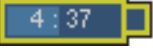



The charge indicator (bar) inside the battery charge indicator displays the approximate amount of power remaining in that battery. Table 14 on page 67 describes the various states of the battery charge indicator.

---

<b>NOTE</b>
-------------

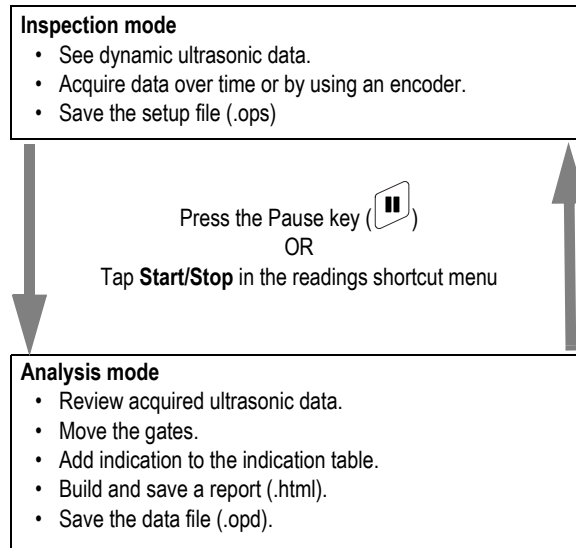
- If you attempt to turn the OmniScan instrument on with one or two batteries that are too low for operation, the power indicator light blinks red rapidly for about three seconds. Replace the battery or batteries, or plug in the DC power adaptor to operate the OmniScan instrument.
  - The OmniScan MX2 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 68 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 39 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 77). 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.

---

<b>TIP</b>
------------

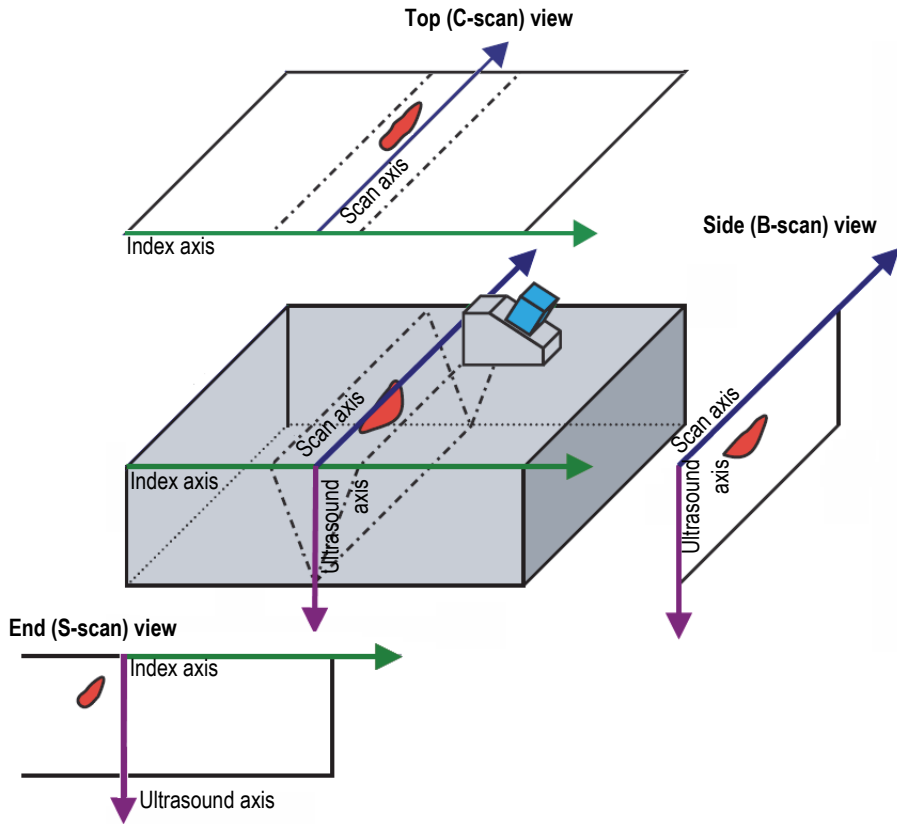
Tap the Layouts section of the title bar to select the layout.

---

Table 15 on page 70 lists the basic ultrasonic views (scans), which are illustrated in Figure 2-19 on page 71.

**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**

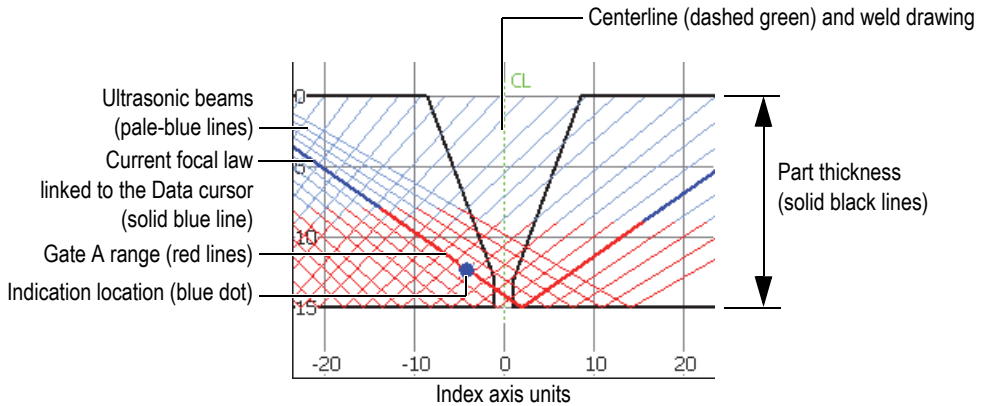
S-scan data view displayed with the ultrasound beam skips in the part. A single group or multiple groups can be represented simultaneously and will be merged. Improves the defect positioning in a weld inspection. The End view is intended only for flat parts, and therefore curved parts will be displayed 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 73 presents the elements of a RayTracing illustration.



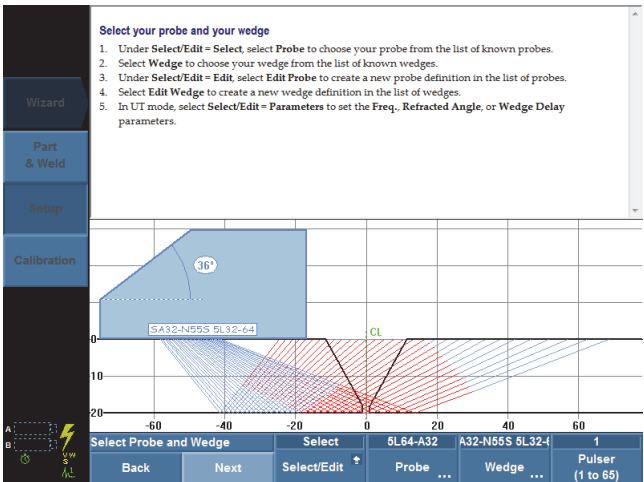
**Figure 2-20 The RayTracing graphical elements**

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 74).



Wizard help

RayTracing illustrates:

- Part and weld dimensions
- Ultrasound beam path
- Gate A limits

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 103).
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 76):
  - 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.

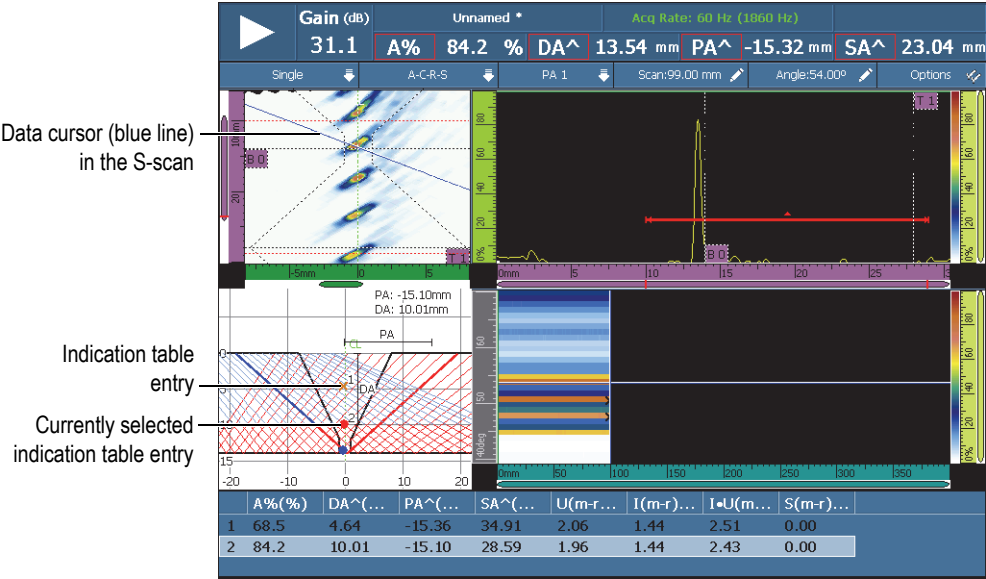


Figure 2-22 Example of indications in the RayTracing view

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

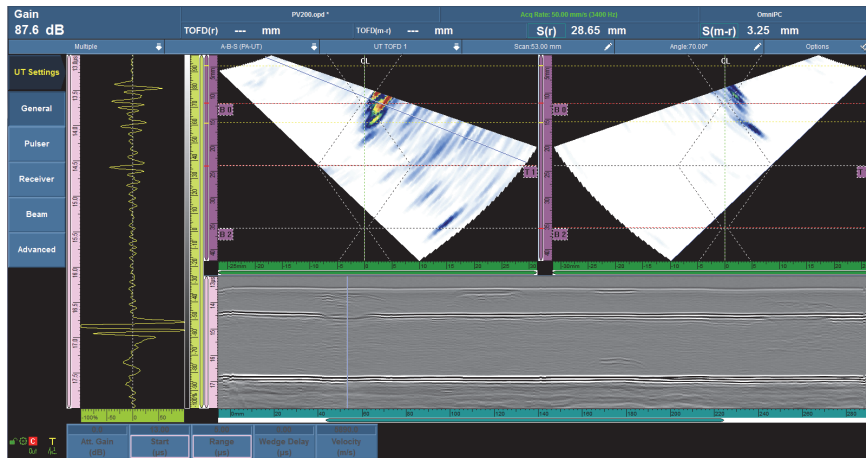
**TIP**

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**.

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

**2.9.2.4 Multiple Group Layouts in OmniPC**

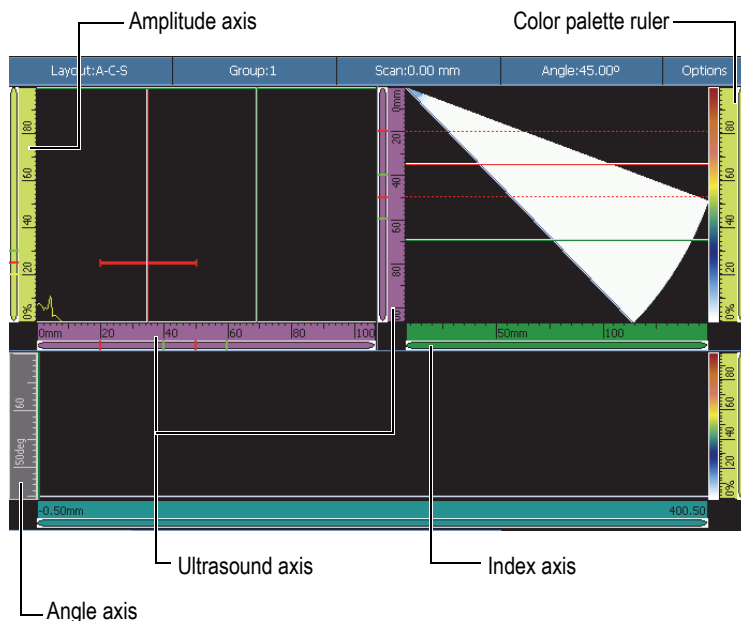
In the OmniPC software, Multiple Group layouts are available in the menu screen mode (see Figure 2-23 on page 77). This feature is not available on the OmniScan MX2 instrument.



**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 78 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 79 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<sup>a</sup>**

















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<sup>a</sup> (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.

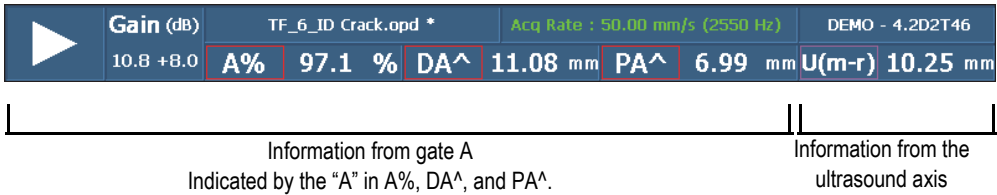
a. The colors in this document differ slightly from those on the OmniScan instrument screen.

## 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 81).



**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 81 the first three reading fields present information from gate A. For more details on reading-field codes, see "Reading Submenu" on page 271.

### UT unit colors

The following information applies to the ultrasound axis:

- Pale-pink: **Time** ( $\mu$ 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 81):

- 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 83). 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** ( $\mu\text{s}$ )
- Pink: **Sound Path** (mm) [also called *half path*]
- Purple: **True Depth** (mm)

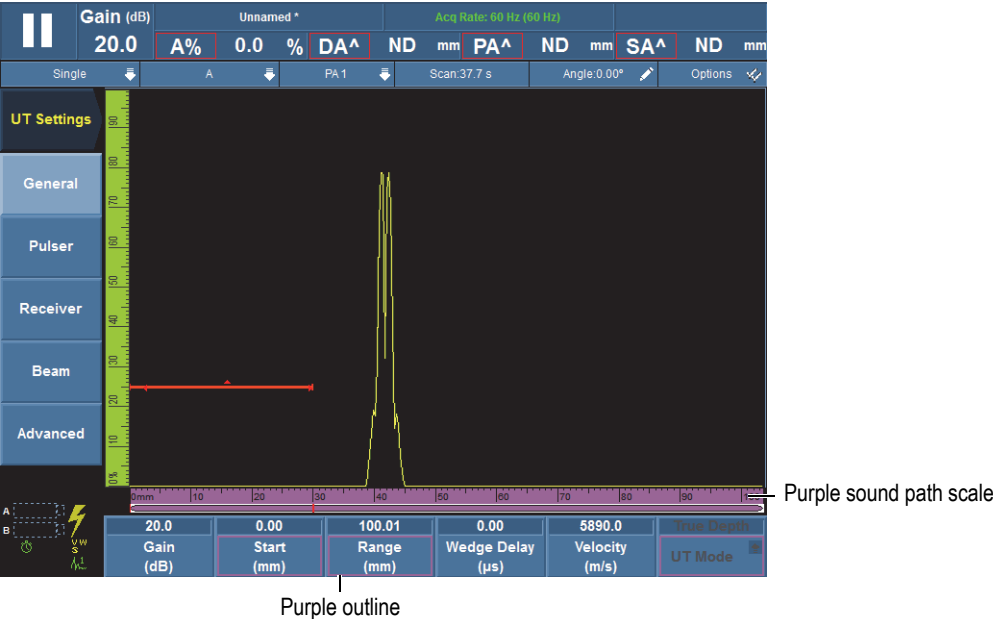


Figure 2-26 The purple color for ultrasound axis and mode button



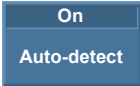
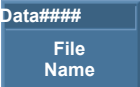
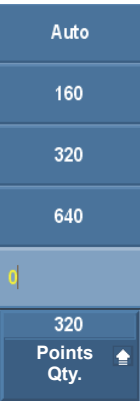
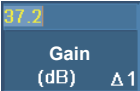
## 2.11 Parameter Buttons

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 84 lists the types of parameter buttons. See Table 11 on page 43 for the procedure on editing parameter values.

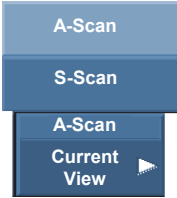
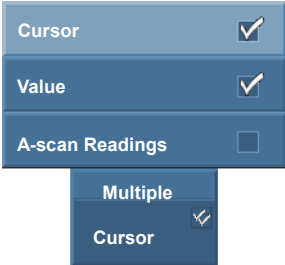
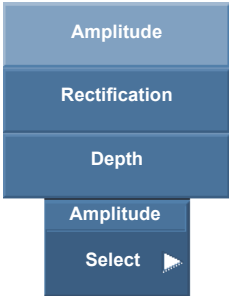
### NOTE

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: <i>a)</i> At the top, a numeric value similar to an edit button value <i>b)</i> At the bottom, a numeric increment/decrement step ( $\Delta$ ) value

**Table 17 Parameter button types (*continued*)**

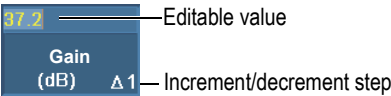
Type	Example	Description
List		Displays a list of predefined items for selection.
Check list		<p>Displays a list of one or more check box items that can be selected individually.</p> <p>Select an item in the list, and then press the Accept key to toggle the state of the check box. The effect is immediate.</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 86). 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 28 or “OmniScan SX Increment Steps” on page 28.



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

**To edit the numeric value of a parameter**

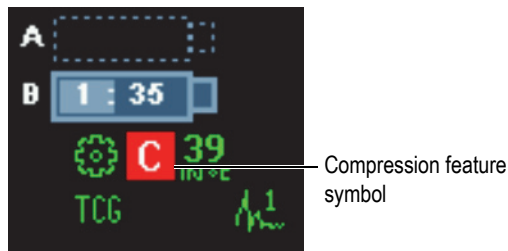
1. Choose the parameter to enable the edit mode.

MX2	SX
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.	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 87):



**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 241.

### 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.

---

<b>NOTE</b>
-------------

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.

---

<b>NOTE</b>
-------------

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**).

---

<b>NOTE</b>
-------------

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 94).  
For more information on file naming codes, see “Data Settings Submenu” on page 248.

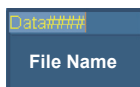


Figure 3-1 File Name parameter button

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**.

---

<b>TIP</b>
------------

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 248.

## 3.3 Producing Reports

This section outlines the procedures for configuring and printing reports.

---

<b>NOTE</b>
-------------

If you want to customize the reports with your company logo, see “Changing the Company Logo” on page 201 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 244) 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 99).

- 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 99).
  - 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 99).

<b>Contractor</b>	NDT Generics	<b>Technician</b>	John Smith
		<b>Project</b>	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 248.
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 98).
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 104). 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.

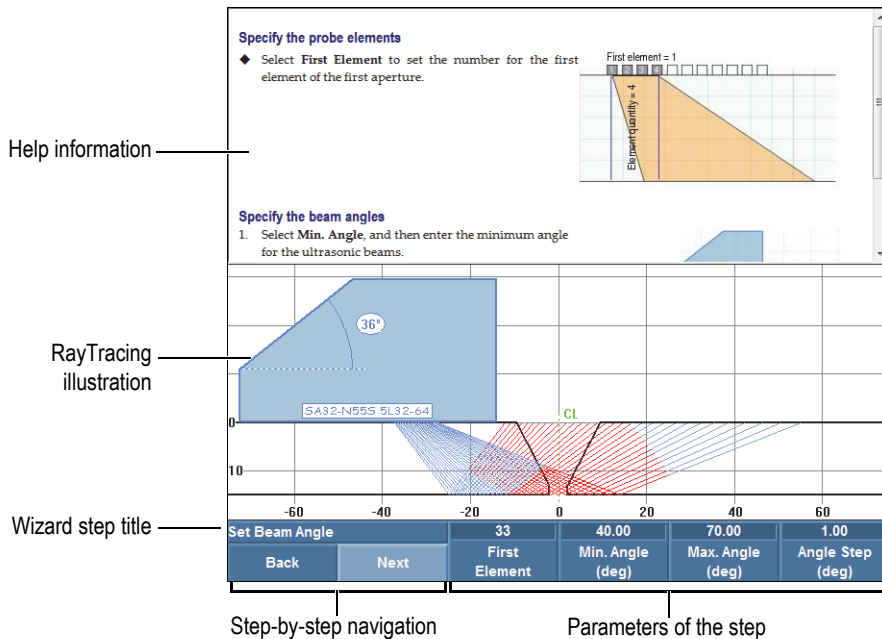


Figure 4-1 Elements of a wizard step

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 104)
- Setup (see “Modifying the Setup” on page 105)

## 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.

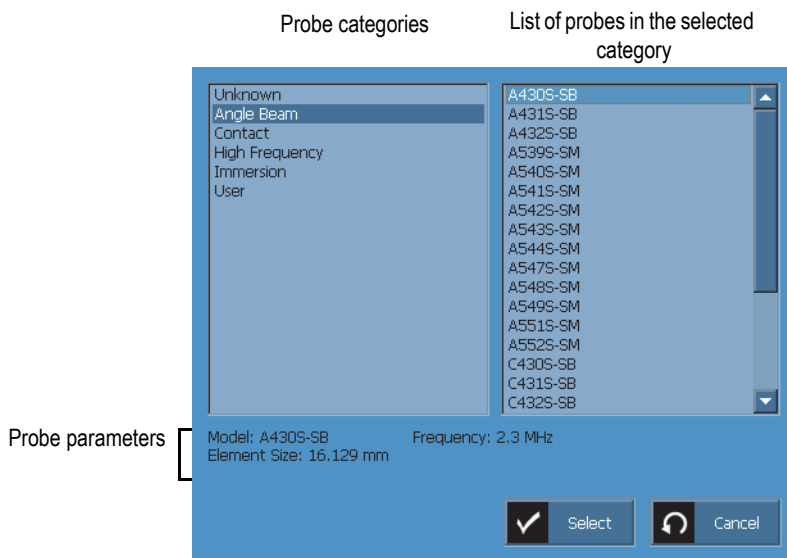
The OmniScan instrument automatically detects and loads the characteristics of an Evident PA probe when you connect one to the instrument with an OmniScan connector. When you use an OmniScan connector and a different brand of probe, you need to manually select the probe from the list of predefined probes.

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 106):
  - 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**

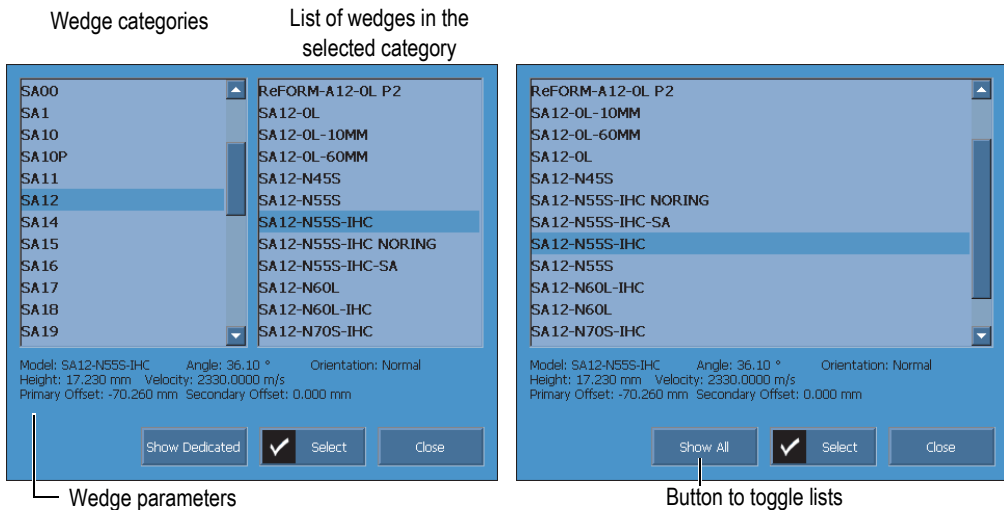
4. When your probe is not available in the predefined list, see “Defining a Probe” on page 108 for the procedure used to define a new probe.

## 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 107):
  - 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 110 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.

---

<b>NOTE</b>
-------------

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

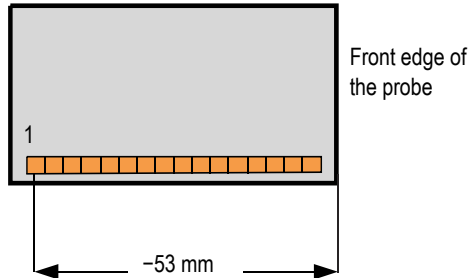
---

### 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 109).

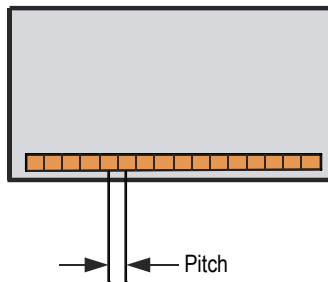
**IMPORTANT**

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 109.



**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.

---

<b>NOTE</b>
-------------

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 105).

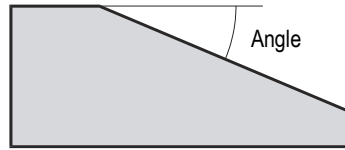
---

## 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 111).

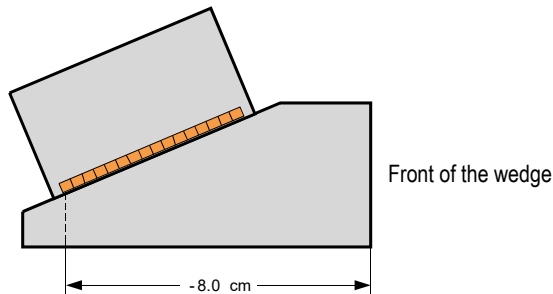


**Figure 4-6 Wedge angle**

- 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 Evident 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 111).

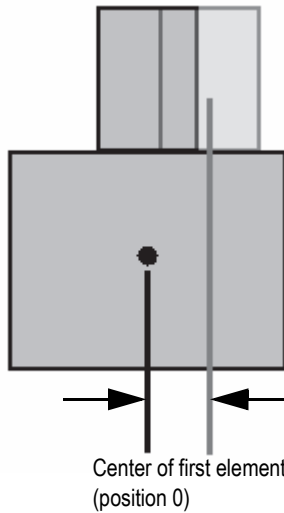
### IMPORTANT

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**

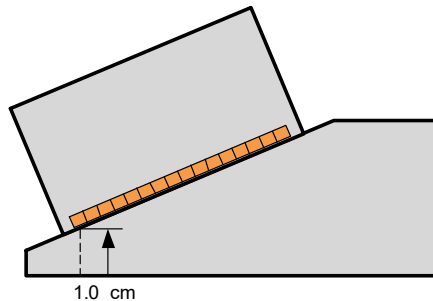
- e) Select **Sec. offset**, and then enter **0** to indicate that the probe is centered on the wedge on the secondary axis. If the probe is not centered on the wedge, enter the appropriate value (see Figure 4-8 on page 112).



**Figure 4-8 The secondary offset**

- f) Select **Height**, and then enter the height of the first element (see Figure 4-9 on page 112).

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  $\mu\text{s}$ .
    - 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**.
- 

<b>NOTE</b>
-------------

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 105).

---

## 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**).

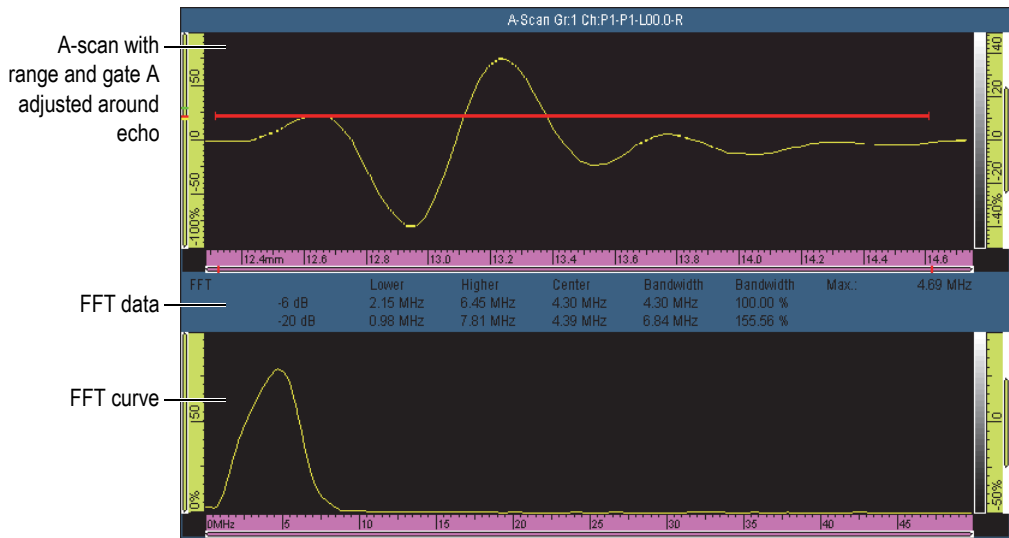
## 4.6 Characterizing a Probe Using the FFT

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 105) or define it (see “Defining a Probe” on page 108).
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 115.



**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 116).



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**.

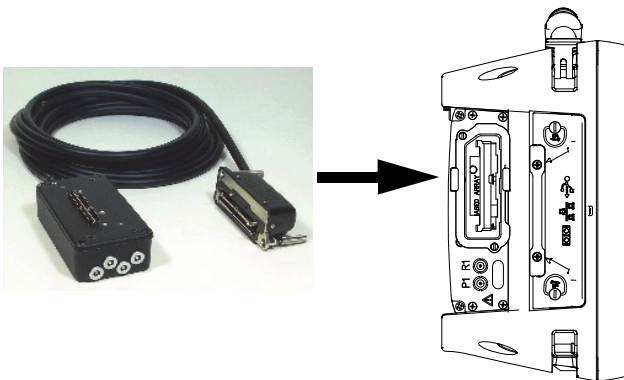
## 4.8 Selecting the Group Mode on the OmniScan MX2

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 118).

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 122 and Figure 5-2 on page 122).

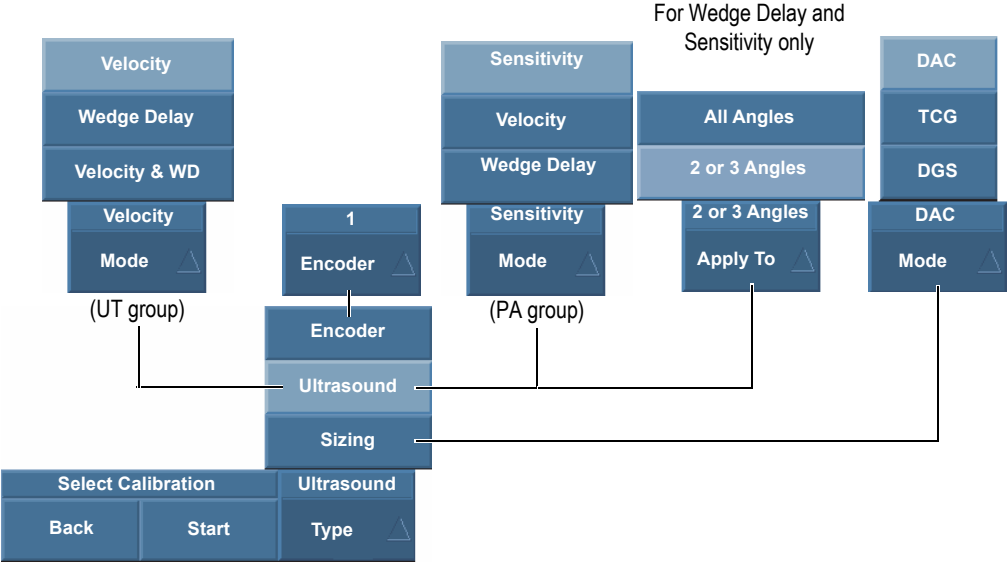


Figure 5-1 Choice of parameters to calibrate in Select Calibration

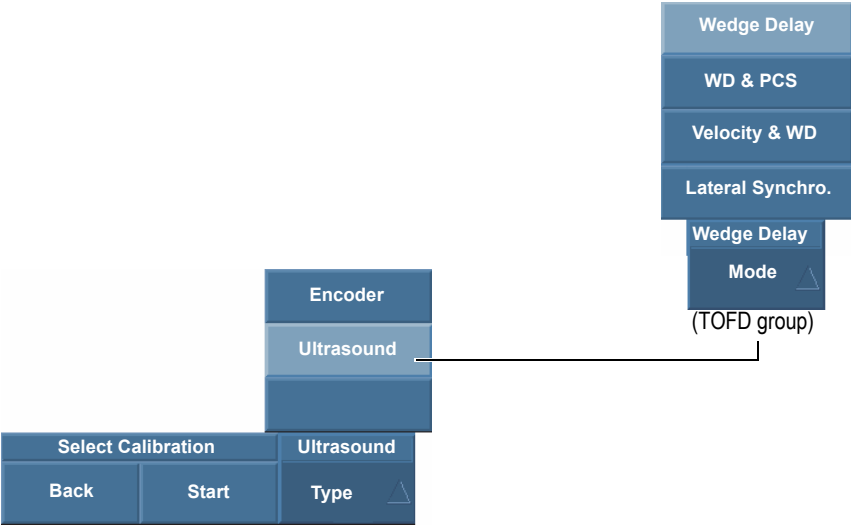


Figure 5-2 Choice of parameters to calibrate – TOFD group

3. Depending on the choice you made in the previous step (see Figure 5-1 on page 122), 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.

- 
4. Select **Start** to move to the first step of the selected wizard.

The wizard guides you through the rest of the procedure.

---

**TIP**

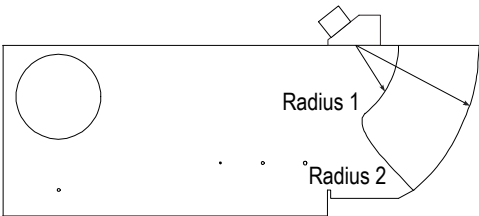
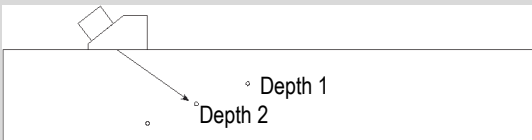
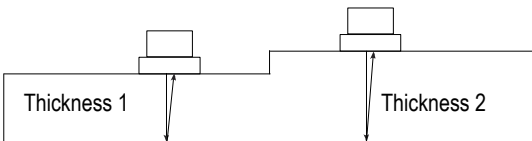
You can exit a Calibration wizard at any time by pressing the Cancel key. When exiting the wizard, the signal goes back to its original state (as before the calibration).

---

## 5.2 Reflector Types

Calibration procedures are performed using calibration blocks with different types of known reflectors. Table 18 on page 124 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 140 and “Sensitivity Calibration for a PA Group” on page 141 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 126 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 129 and “Wedge Delay Calibration for a PA Group” on page 135 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 133 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.

---

<b>IMPORTANT</b>
------------------

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 124.
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 127).
  - 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 127).

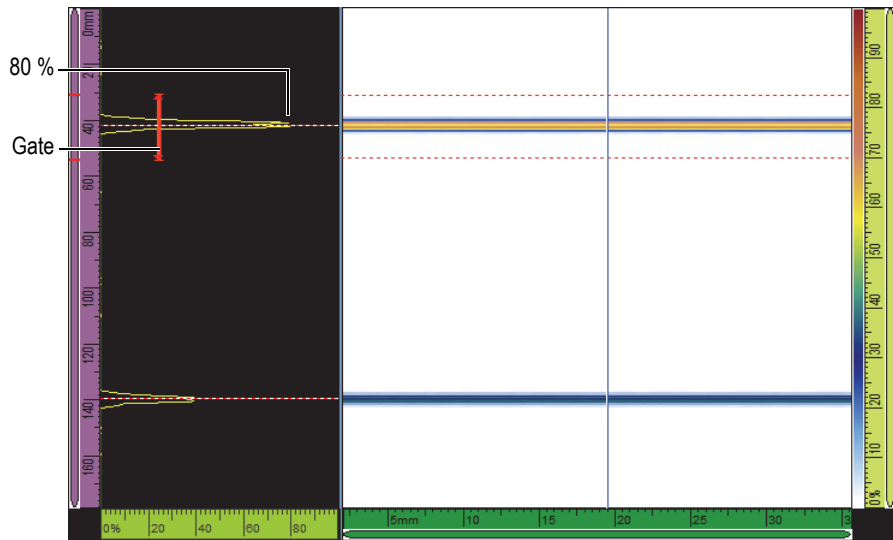


Figure 5-3 Example of reflector signals in a linear scan

- 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 127).
- 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 128).

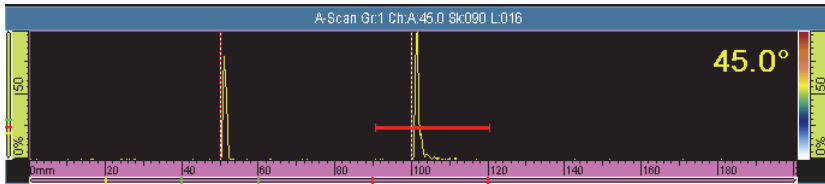


Figure 5-4 Setting the gate on the second signal

- 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 128).

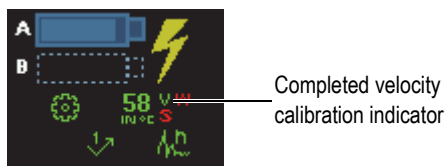


Figure 5-5 Velocity indicator (V) in green after velocity calibration

## 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.

---

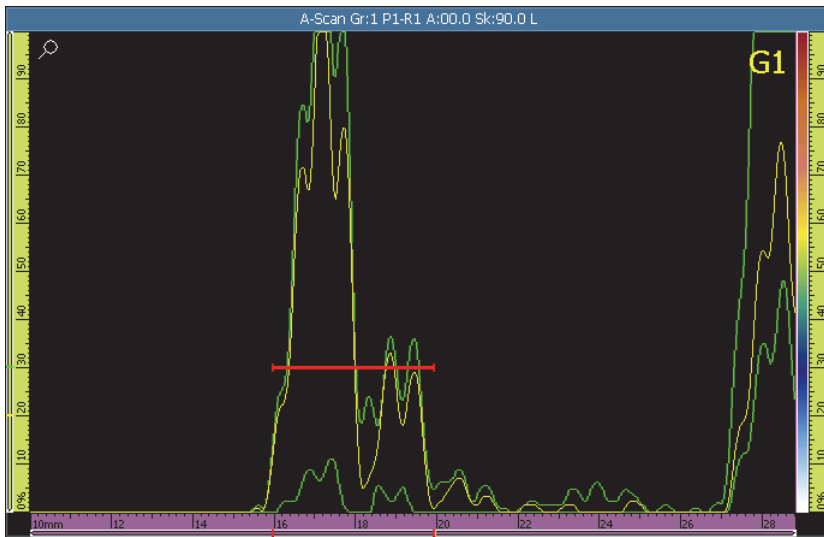
This procedure applies to the following groups and reflectors:

- 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 124.
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 130).
  - 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 131).

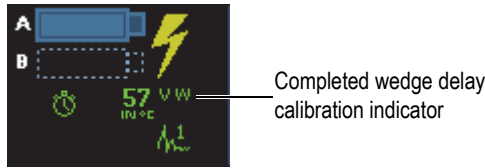


Figure 5-7 Wedge delay indicator (W) after calibration for UT group

### 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:

---

<b>NOTE</b>
-------------

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 [ $\mu$ 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.

---

<b>NOTE</b>
-------------

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**.

---

<b>NOTE</b>
-------------

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 124.
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 135).

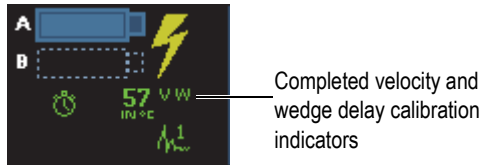


Figure 5-8 The velocity (V) and wedge delay (W) indicators

#### 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 124.
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 137).

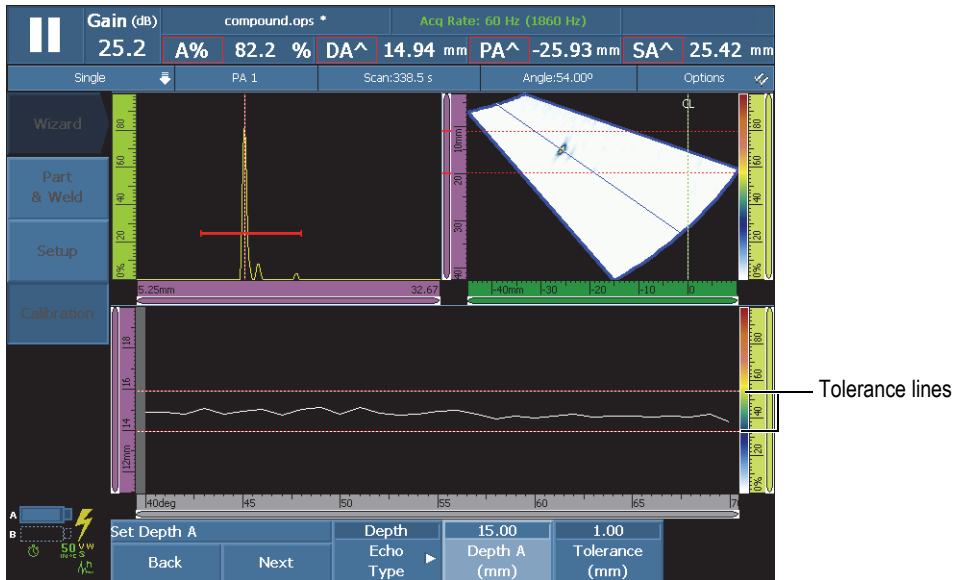


Figure 5-9 Building the envelope for the wedge-delay calibration

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

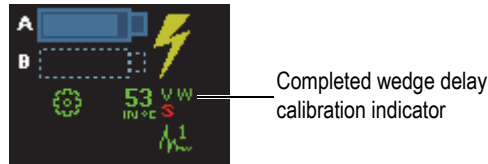
When the calibration is correct, the envelope is between the two red dotted lines.

- 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 138).



**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 124.
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 139).

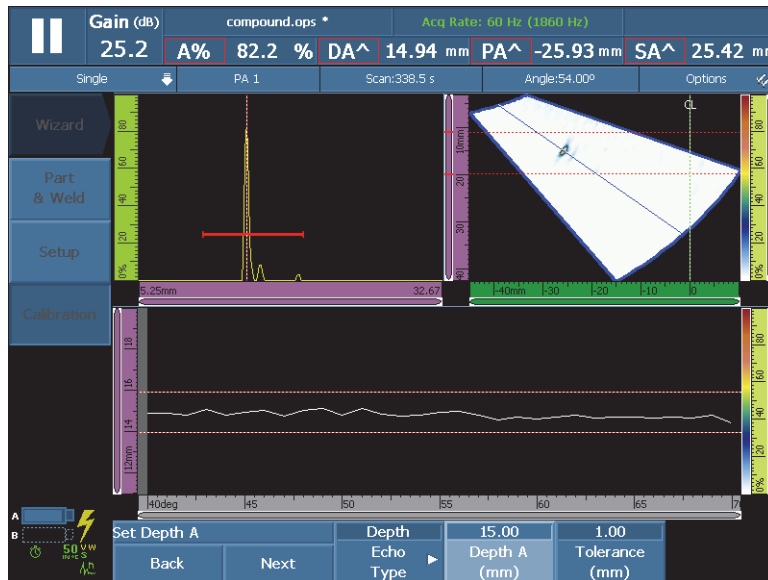


Figure 5-11 Building the envelope for the first angle/VPA

- 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 140).

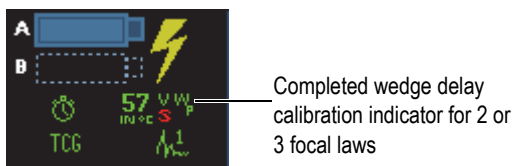


Figure 5-12 Wedge delay indicator ( $W_P$ ) after calibration for 2 or 3 focal laws

### 5.4.7 Sensitivity Calibration for a UT Group

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

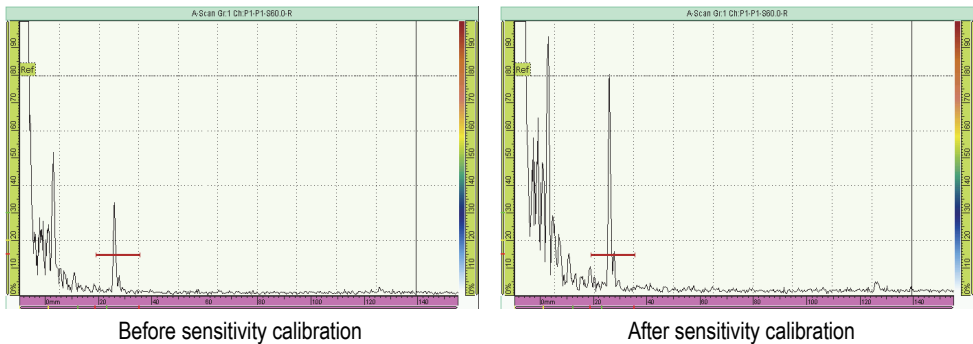
This procedure applies to the following groups and reflectors:

- 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 141).
4. Select **Set to XX.X%** to perform the sensitivity calibration.

Figure 5-13 on page 141 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

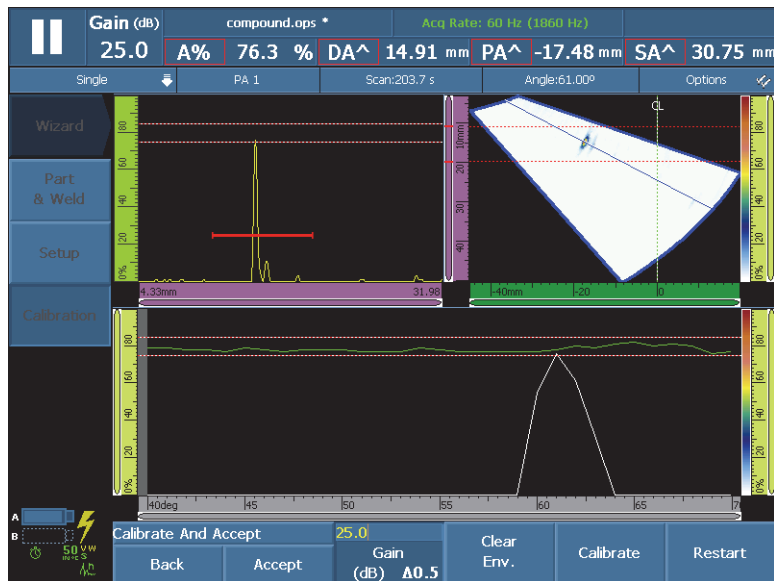
- Sectorial or linear scans (angle/VPA)

### To calibrate the sensitivity for all focal laws

1. Place the probe on a calibration block, as shown in Table 18 on page 124.
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 143).



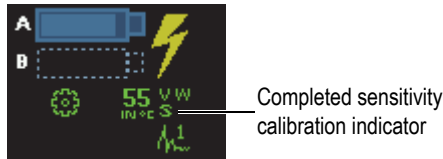
**Figure 5-14 Building the envelope for the sensitivity calibration**

- 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 144).



**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 124.
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 146).

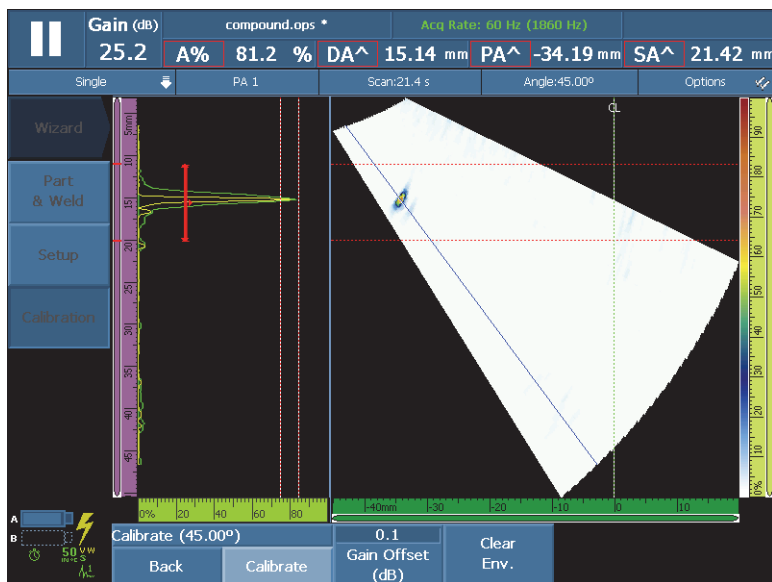


Figure 5-16 Building the envelope for the sensitivity calibration of the first angle

- 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 146).

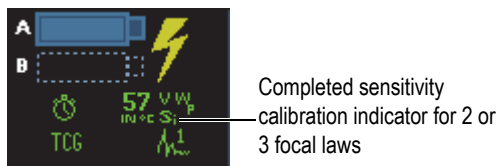


Figure 5-17 Sensitivity indicator ( $S_i$ ) after calibration for 2 or 3 focal laws

## 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 298 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

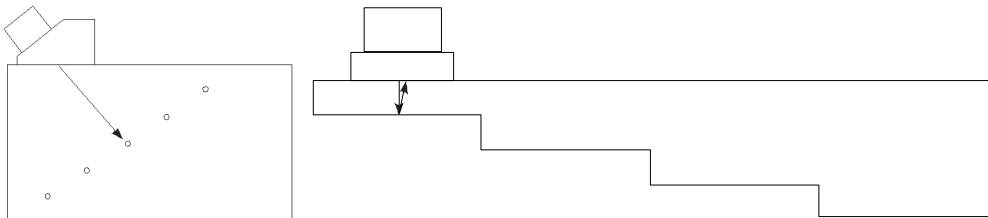
Because the DAC curve(s) will only be seen in the A-scan, they are more appropriate for conventional UT inspection. If using PA, a TCG is more appropriate, because the amplitude correction will be seen in the S-scan as well as the A-scan.

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

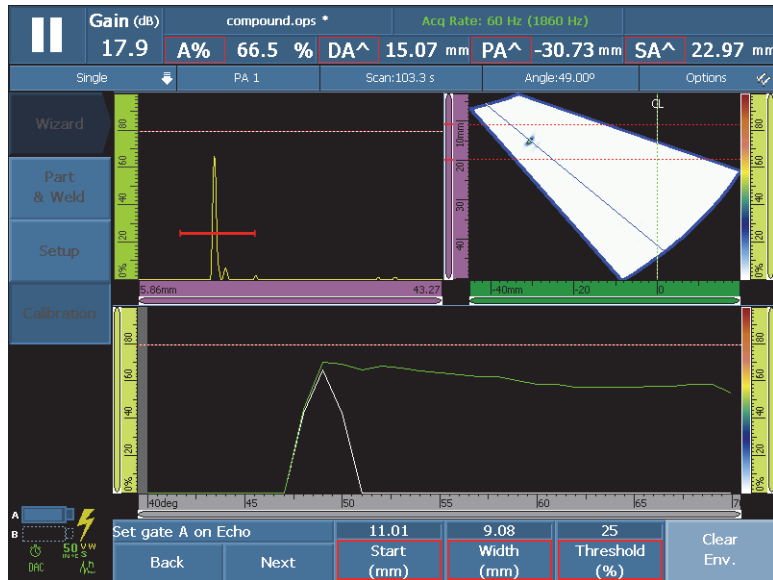
1. Place the probe on a calibration block, as shown in Figure 5-18 on page 147.



**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 302 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 149).
  - 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 149.



**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.

---

<b>TIP</b>
------------

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

---

## 5.5.2 TCG Calibration

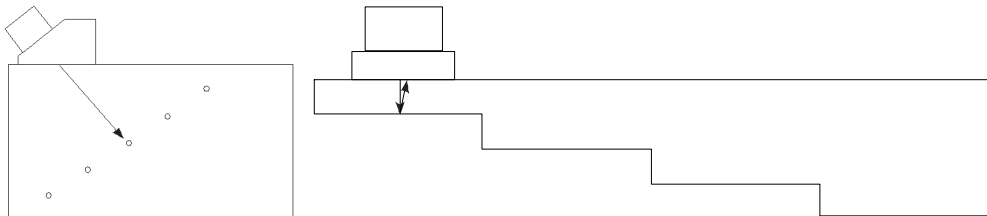
This section describes how to calibrate the TCG (time-corrected gain) sizing function. To perform this procedure, you need a calibration block with a number of identical holes at known depths or with multiple known thicknesses.

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

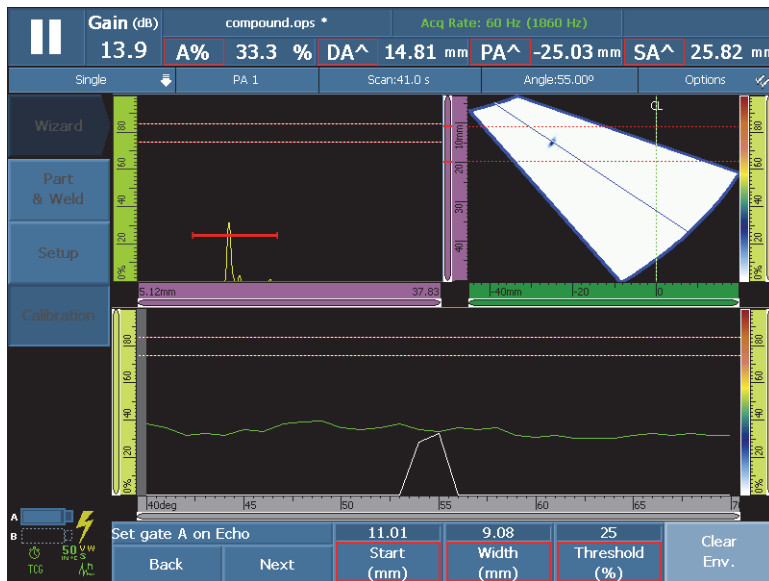
1. Place the probe on a calibration block, as shown in Figure 5-20 on page 150.



**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 302 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 152).
  - 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 152.



**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

---

<b>NOTE</b>
-------------

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 156):

---

<b>NOTE</b>
-------------

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.

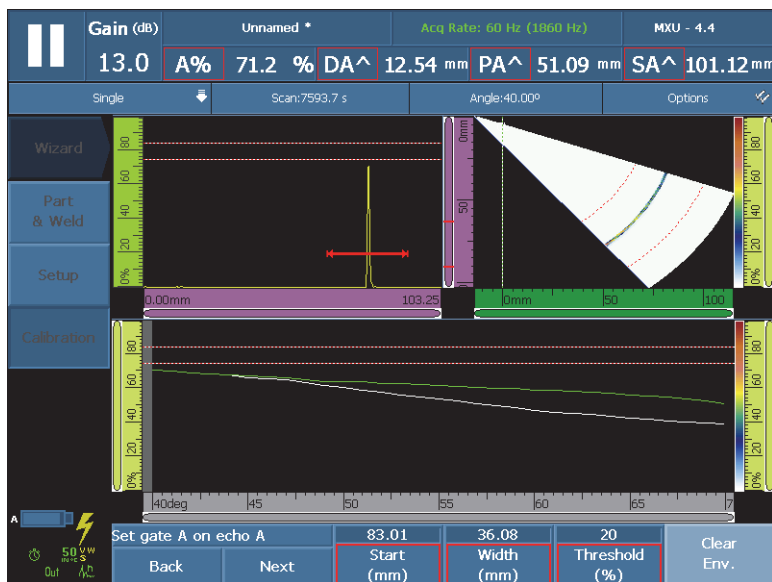


Figure 5-22 Set Gate A over the signal envelope

- 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 157, 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.

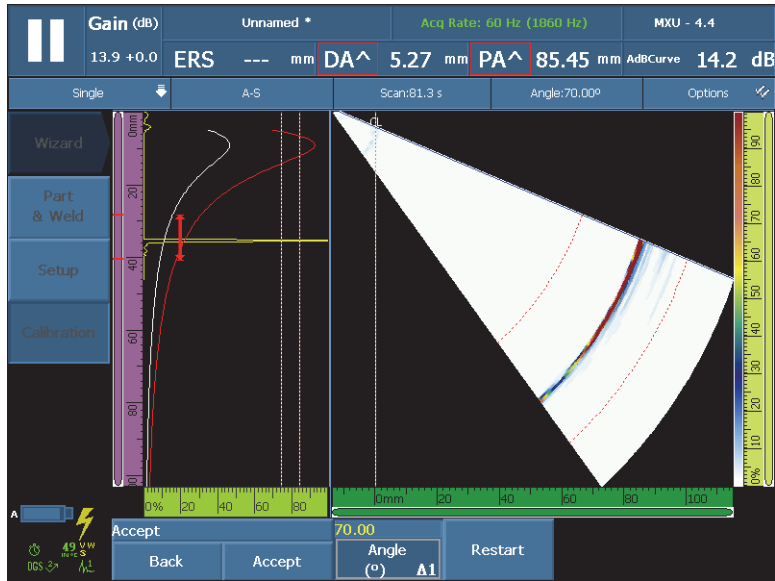


Figure 5-23 Example of DGS sizing curves

8. In the **Accept** step, proceed as follows:
  - a) If needed, select **Angle (°)** to set the desired angle and verify the calculated curve.
  - b) If the calibration is not satisfactory, select **Restart** to restart the calibration process.
  - c) 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 157).



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 158 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.

$$A - B - C = D \tag{1}$$

where:

A = Discontinuity Indication Level in dB (see “**AWS A**” on page 286)

B = Reference Indication Level in dB (see “**AWS B**” on page 286)

C = Attenuation Factor in dB (see “**AWS C**” on page 287)

D = Indication Rating in dB (see “**AWS D**” on page 287)

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°.

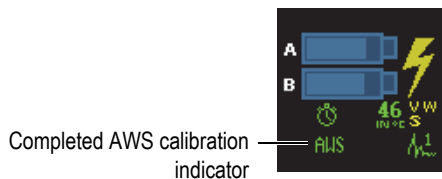
After setup creation, the next step consists of calibrating the sensitivity.

## 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 159).



**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

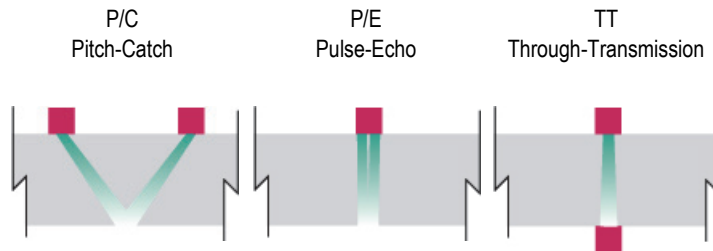
1. Select **UT Settings > General > Gain**.
2. Enter an appropriate gain value. For an appropriate gain value, you should typically view the peak amplitude in a gate at about 80 % of the full-scale view.
3. Select **Start**, and then enter the start position value of the displayed range.
4. Select **Range**, 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 164).



**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.

---

<b>NOTE</b>
-------------

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 167).

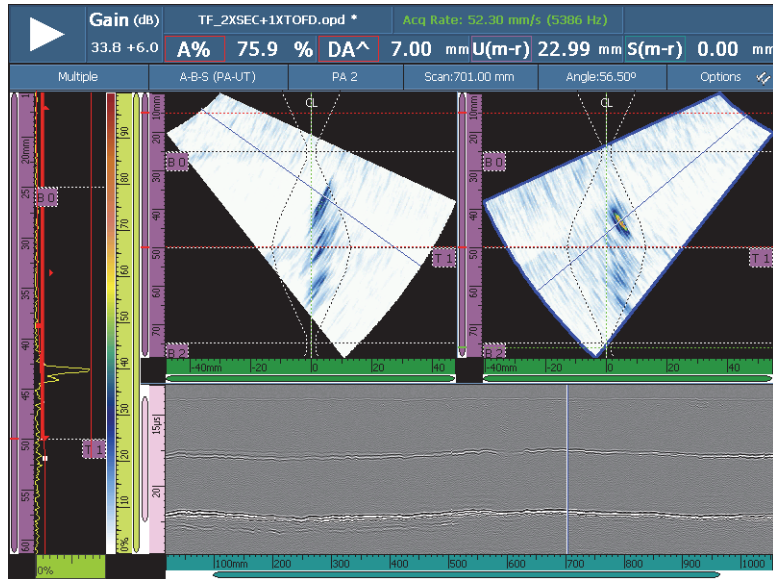


Figure 6-2 An example of the PA-UT Weld (A-B-S) layout

## 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 135.

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 126.
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/**.

---

<b>NOTE</b>
-------------

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<sup>^</sup>)**, **T (B/)**, or **T(A<sup>^</sup>-B/)**, and so on, according to the selections made under **Gates/Alarms > Thickness > Source**.

---

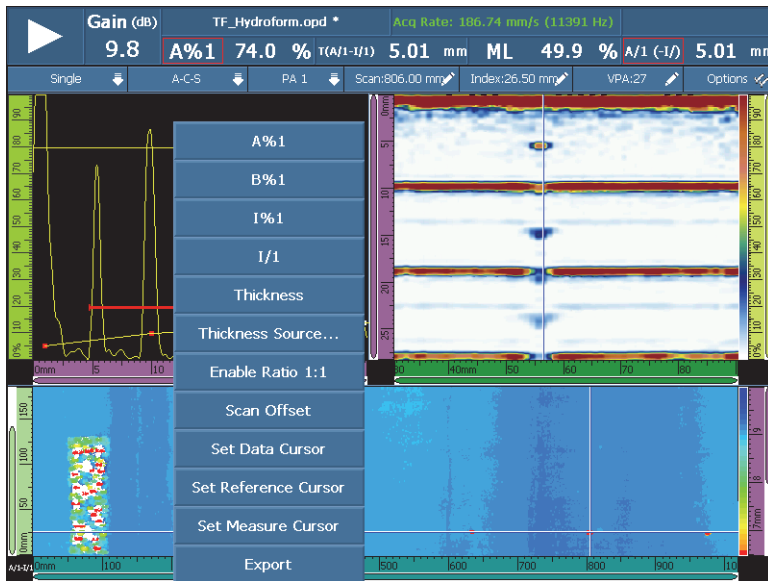
11. Select the **A-C** layout in the Layout section of the title bar.
12. To choose C-scan sources, tap and hold the C-scan until a contextual menu appears, where you can choose from several available sources.
13. Select **Thickness** to display a thickness C-scan.  
The thickness C-scan has been configured.

## 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 170).



**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 171). The **A-B-C-D Profile** layout is also available, but only in analysis mode.

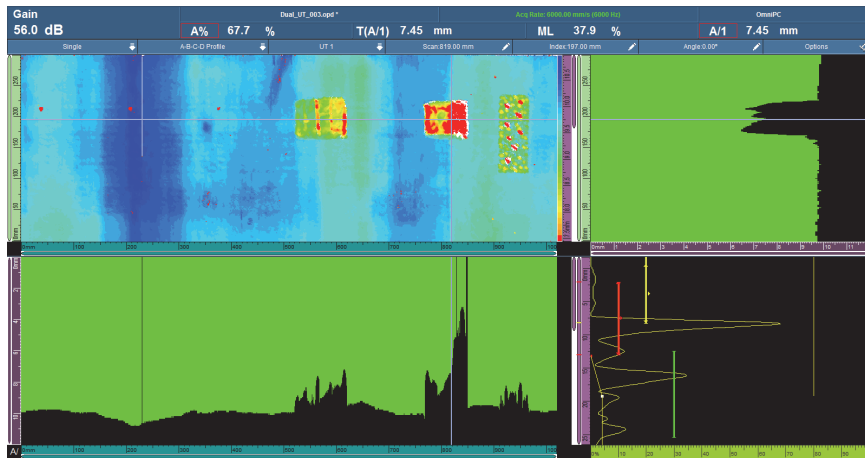


Figure 6-4 The Profile B-scan view

## 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 172).

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

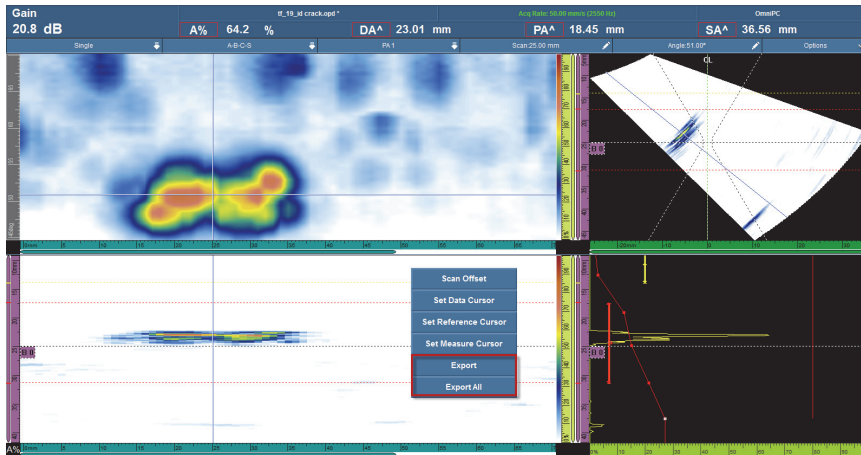


Figure 6-5 The Export function

## 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.

---

<b>NOTE</b>
-------------

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 147 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 302.
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 175).

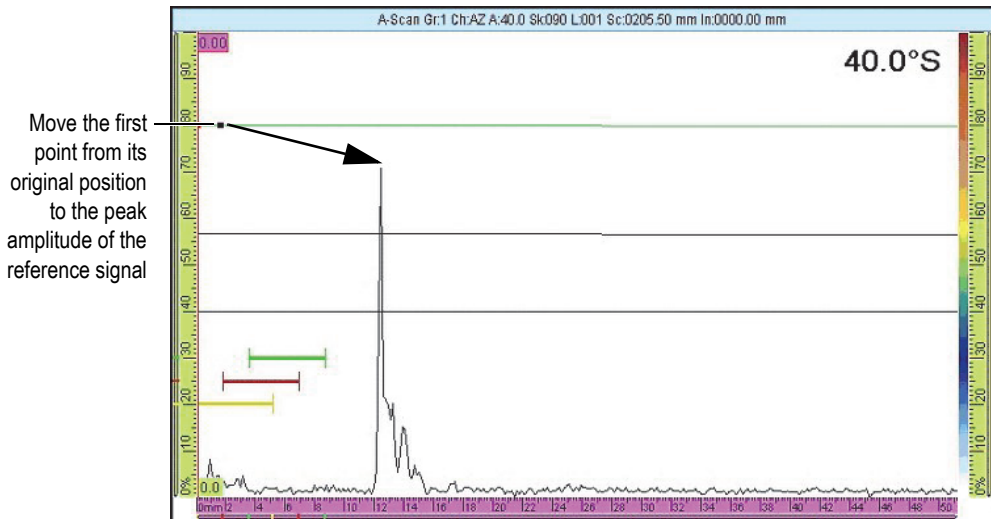


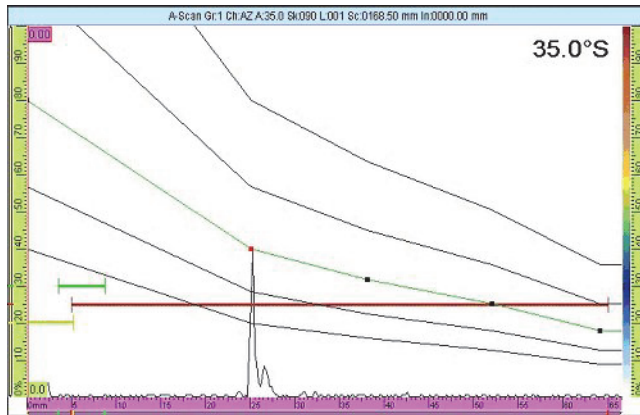
Figure 6-6 The first point of the DAC curve

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 176).



**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 177).

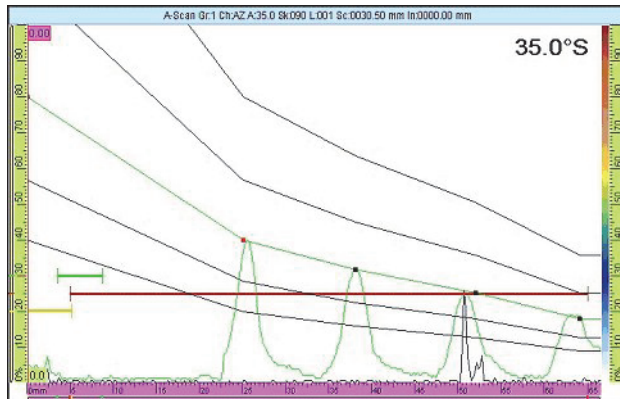


Figure 6-8 Verification of the DAC curve

---

**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/ $\mu$ 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 302 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.

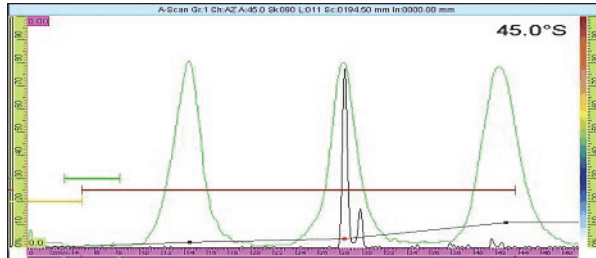
### To delete a point on the curve

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 179).



**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 62 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 180). You can easily choose a group of eight parameters that are appropriate to your application.

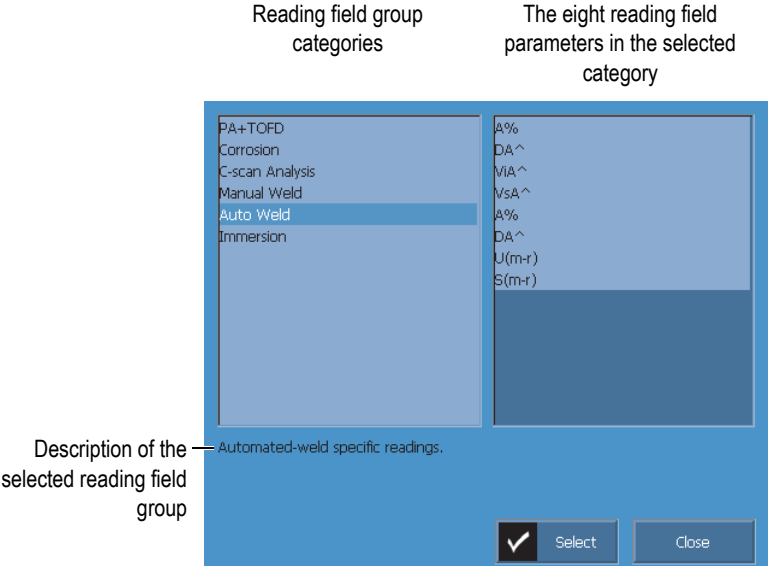


Figure 6-10 Dialog box for choosing predefined readings

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 291.

### 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 181).

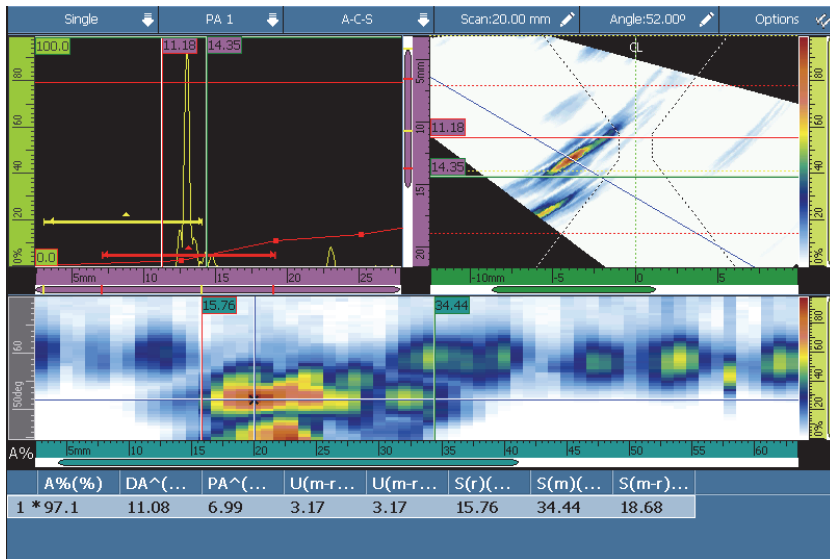


Figure 6-11 Example of an indication table

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 69 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 182).

Entry	Scan mm	Index mm	Group	Channel	A% %	DA <sup>^</sup> mm	PA <sup>^</sup> mm	SA <sup>^</sup> 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

Show Table

Add

1

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 182).

---

**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 48 and “Scans, Views, and Layouts” on page 69.

2. To access additional A-scan configuration options, select **Display > A-Scan**.  
For more details about these options, see “A-Scan Settings Submenu” on page 292.

### 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 184).

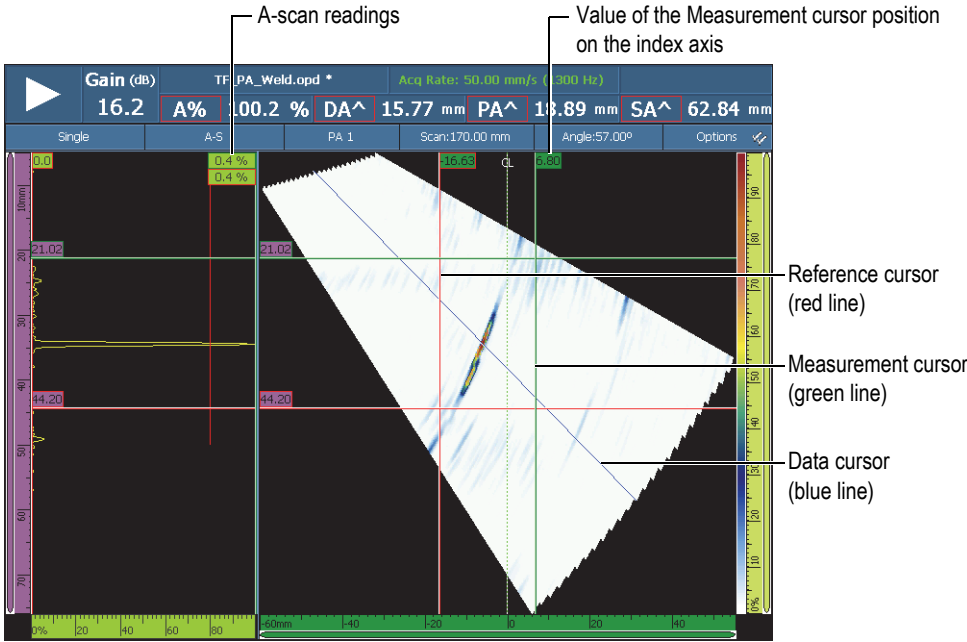



Figure 6-13 Example of cursors

---

### NOTE

In the cursor related data boxes (ex.: , 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 296 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.

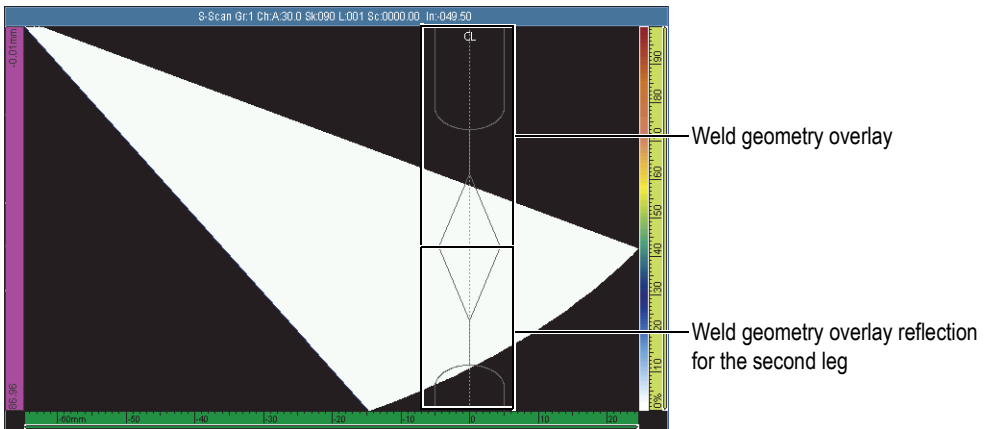


Figure 6-14 Example of a J-type weld geometry overlay

#### 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.

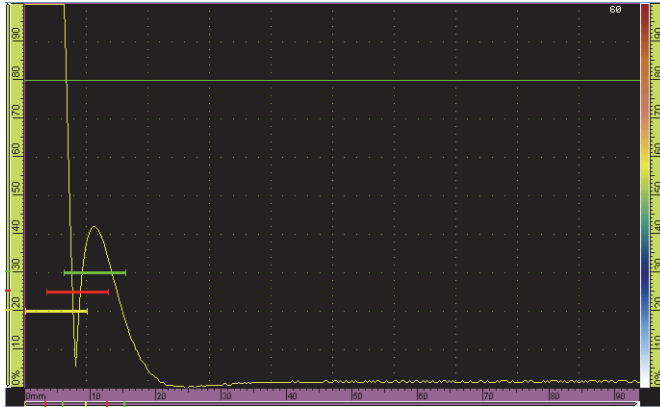
#### To display the leg overlay

- ◆ 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 188).



**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 292 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**

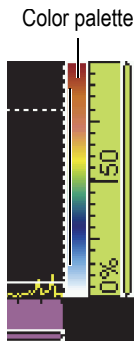
In order for the **Ratio 1:1** parameter button to be available, the inspection must be synchronized on two axes. To perform the synchronization, select **Scan > Inspection > Type = Raster Scan**.

## 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 189), 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 Evident 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 189).

### 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 332 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 191).

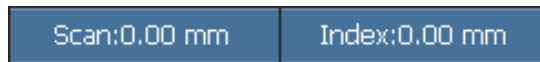


Figure 6-17 Scan and Index reading fields

- 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.

---

<b>NOTE</b>
-------------

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 193).

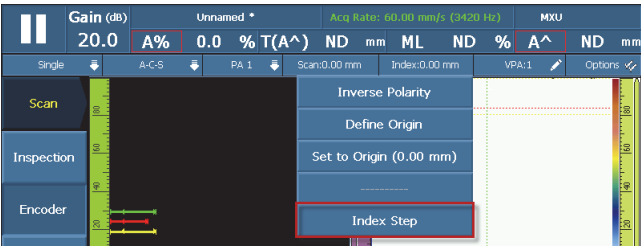


Figure 6-18 The Index Step function

### 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

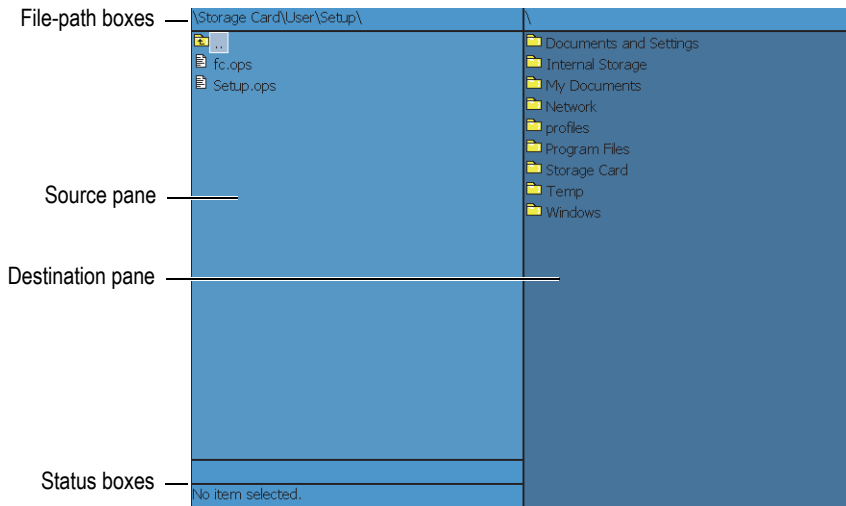
This section describes the File Manager utility. The File Manager interface is designed to provide as many management options as possible, even if no keyboard or mouse is connected to the instrument.

#### 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 196).



**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 197).



**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 196).

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 198).

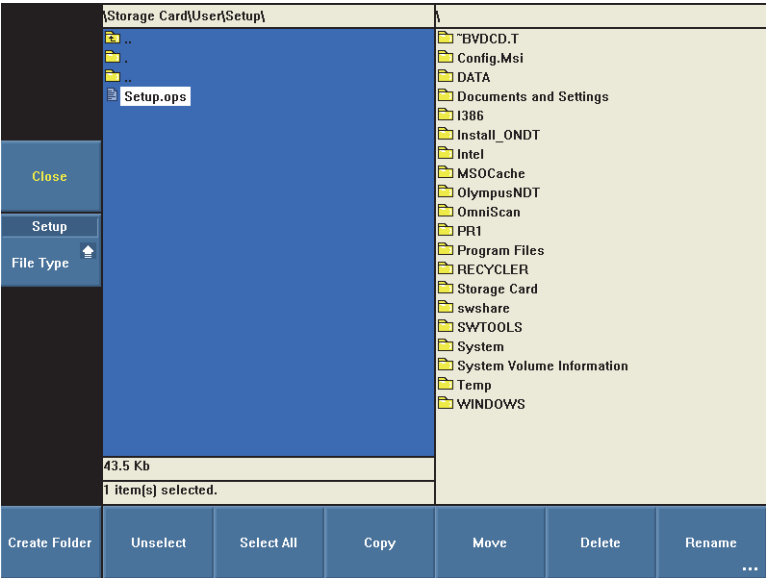


Figure 7-3 The source pane highlighted

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.

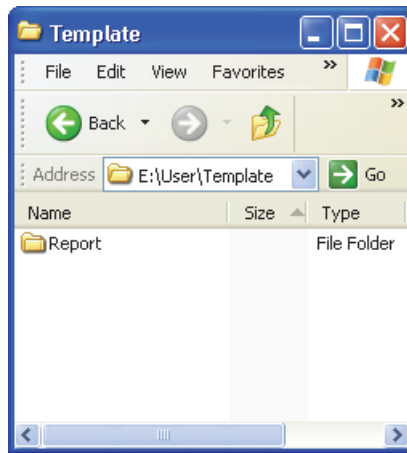
---

<b>IMPORTANT</b>
------------------

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.

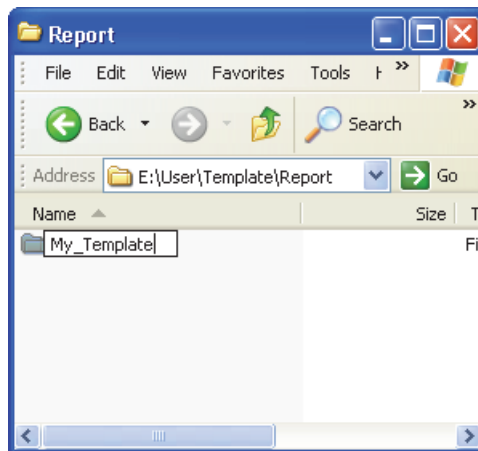
---

- b) If it does not already exist in the storage card, create the following folder:  
    \User\Template\Report\ (see Figure 7-4 on page 200).



**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 200).



**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 199.
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.

---

<b>TIP</b>
------------

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 203 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 203.
- 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 203.
- 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 203.
- 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 203.
- 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 203.
- 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 207.

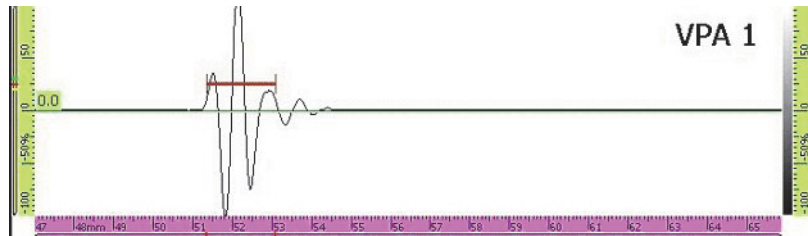


Figure 7-6 Gate A on the ball signal

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 208

For Windows XP:

- "Configuring the Computer Network Connection — Windows XP" on page 209
- "Creating a User Account on a Computer — Windows XP" on page 212
- "Creating a Shared Folder on a Computer — Windows XP" on page 215
- "Configuring OmniScan MX2 Network Connection — Windows XP" on page 221

For Windows 7:

- "Configuring the Computer Network Connection — Windows 7" on page 223
- "Creating a User Account on a Computer — Windows 7" on page 226
- "Creating a Shared Folder on a Computer — Windows 7" on page 229
- "Configuring the OmniScan MX2 — Windows 7" on page 234

Finally, the procedure entitled "Saving OmniScan Data on a Computer" on page 235 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**

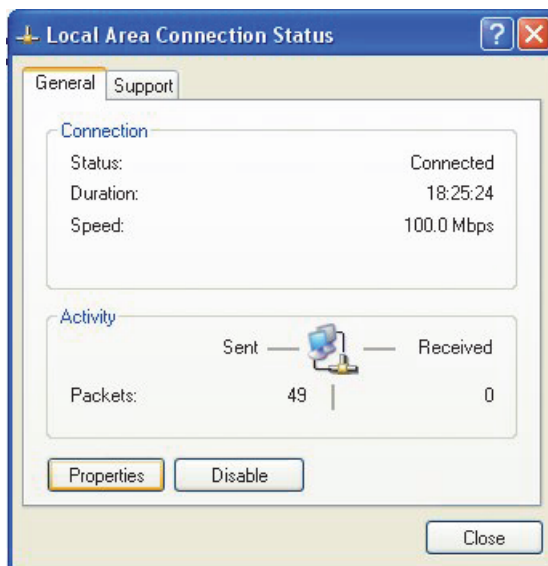
1. Connect a crosslink Ethernet (RJ-45) cable to the network port of the computer.
2. Connect the other end of the cable to the OmniScan MX2 Ethernet port.

**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**

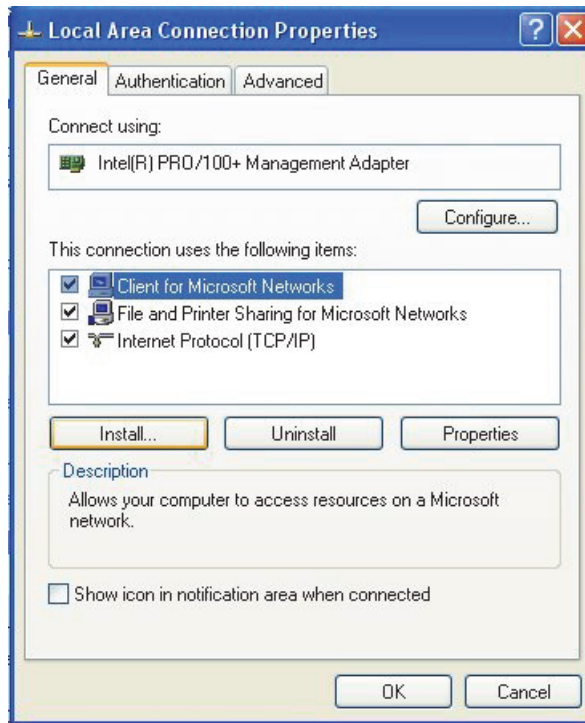
1. On the computer, click **Control Panel**.
2. Open **Network Connections**.
3. Double-click the connection used to establish the link with the OmniScan MX2.  
The **Local Area Connection Status** dialog box appears (see Figure 7-7 on page 210).



**Figure 7-7 The Local Area Connection Status dialog box (Windows XP)**

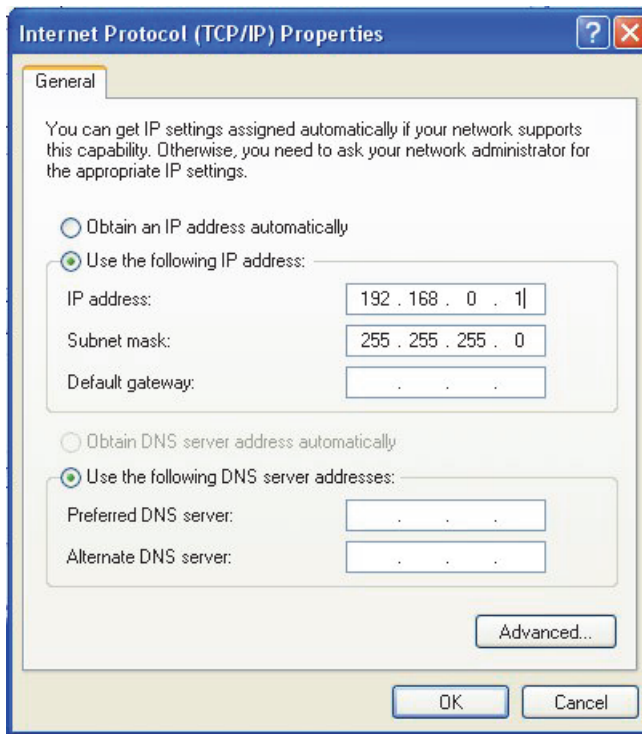
4. On the **General** tab, click **Properties**.

The **Properties** dialog box for the selected connection appears (see Figure 7-8 on page 211).



**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 212).



**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 221).
9. In the **Subnet Mask** text box, type an OmniScan-compatible Subnet Mask (see “Configuring OmniScan MX2 Network Connection — Windows XP” on page 221).
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 213).

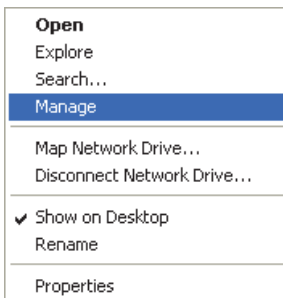


Figure 7-10 The shortcut menu for the My Computer icon

The **Computer Management** dialog box appears (see Figure 7-11 on page 213).

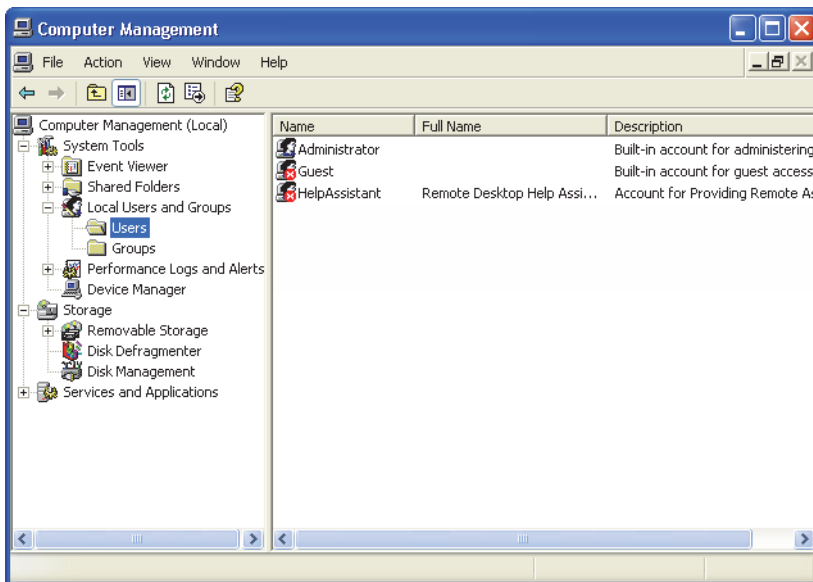


Figure 7-11 The Computer Management dialog box (Windows XP)

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 214).

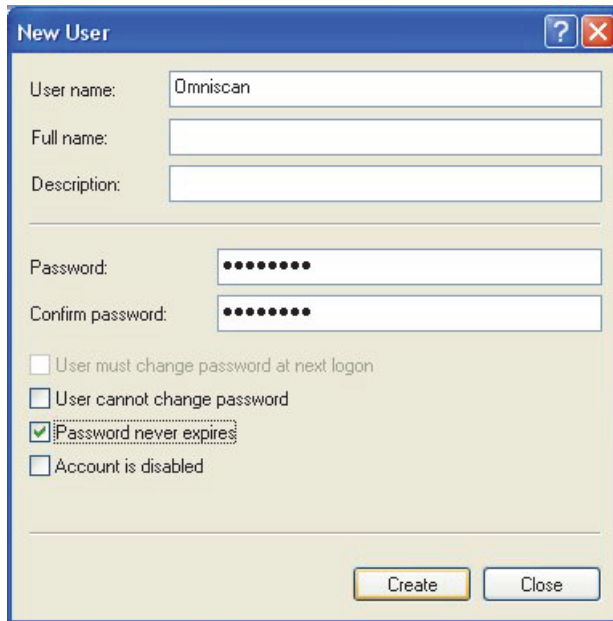


Figure 7-12 The New User dialog box (Windows XP)

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 “OmniScan” on the hard disk drive of the computer.
2. Right-click the **OmniScan** folder, and then click **Sharing and Security** on the shortcut menu.
3. In the **OmniScan Properties** dialog box, click the **Sharing** tab (see Figure 7-13 on page 215).

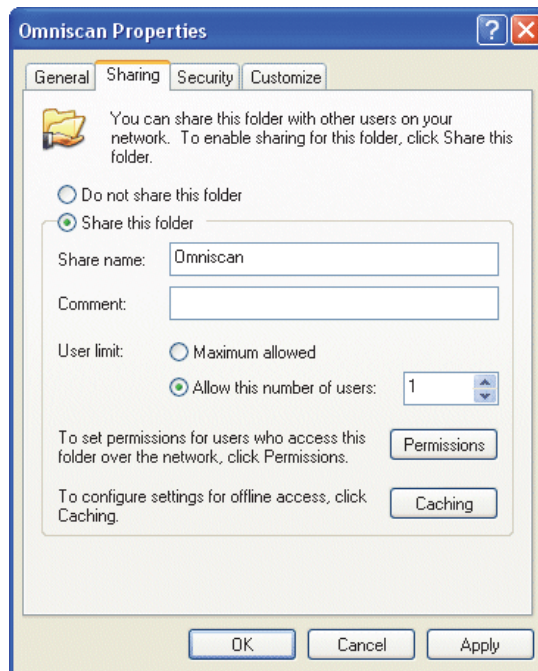


Figure 7-13 The Sharing tab of the OmniScan Properties dialog box (Windows XP)

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 216).

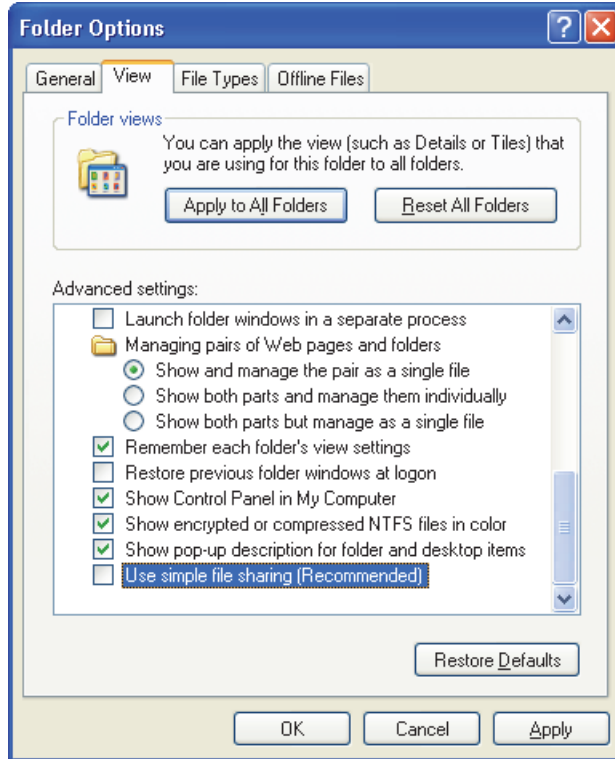
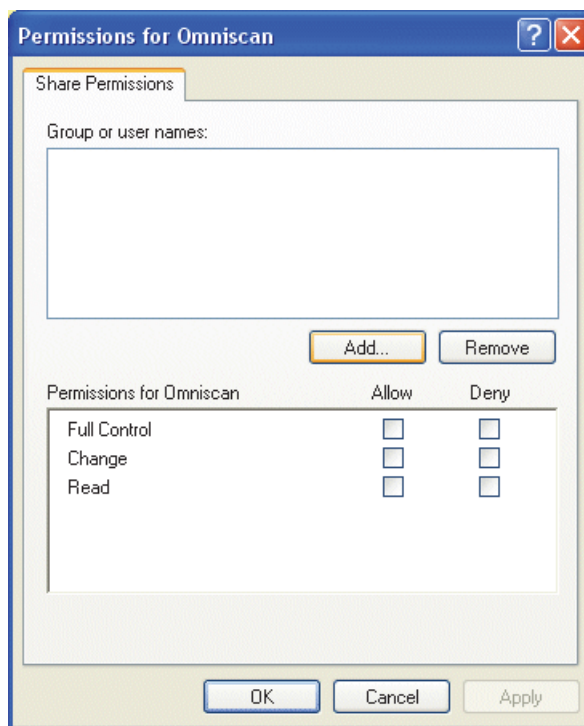


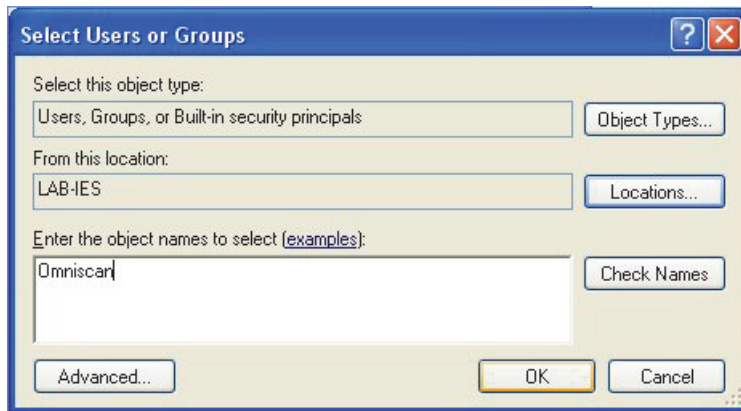
Figure 7-14 The Folder Options dialog box (Windows XP)

- 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 217).



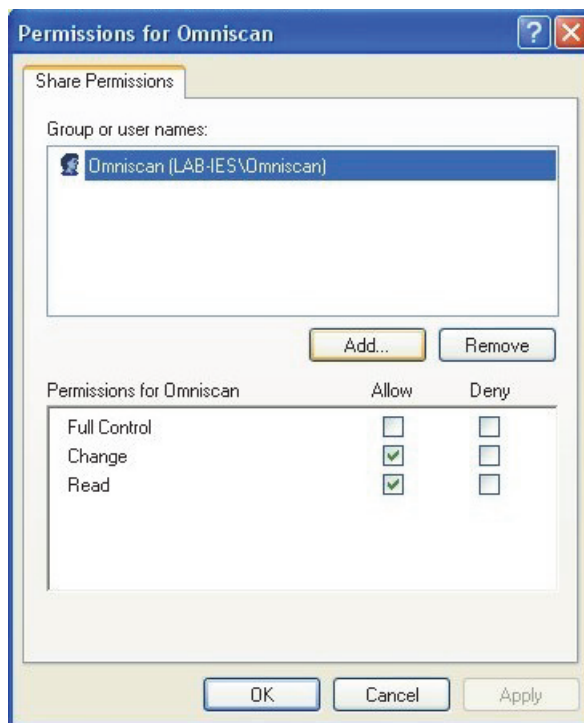
**Figure 7-15 The Permissions for Omniscan dialog box (Windows XP)**

8. In the **Select Users or Groups** dialog box, click the **Locations** button (see Figure 7-16 on page 218).



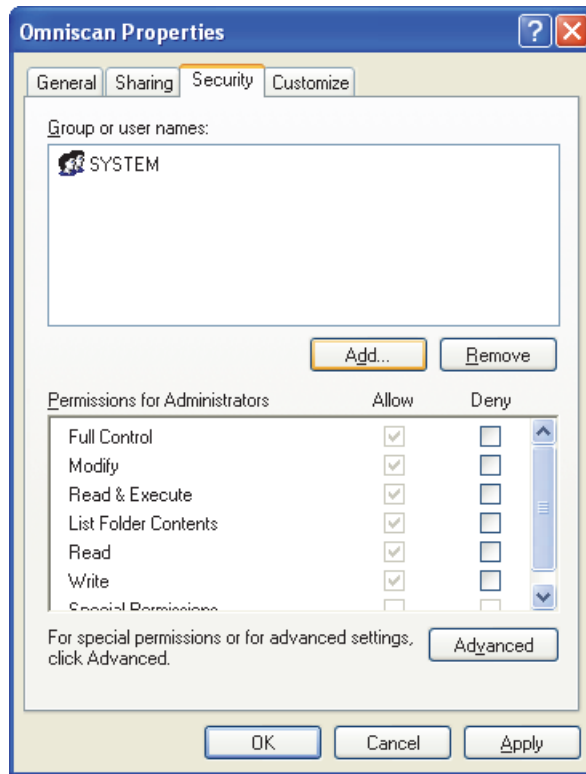
**Figure 7-16 The Select Users or Groups dialog box (Windows XP)**

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 218) 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 219).



**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 220).



**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 218).
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 218) 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 221):
  - 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**.

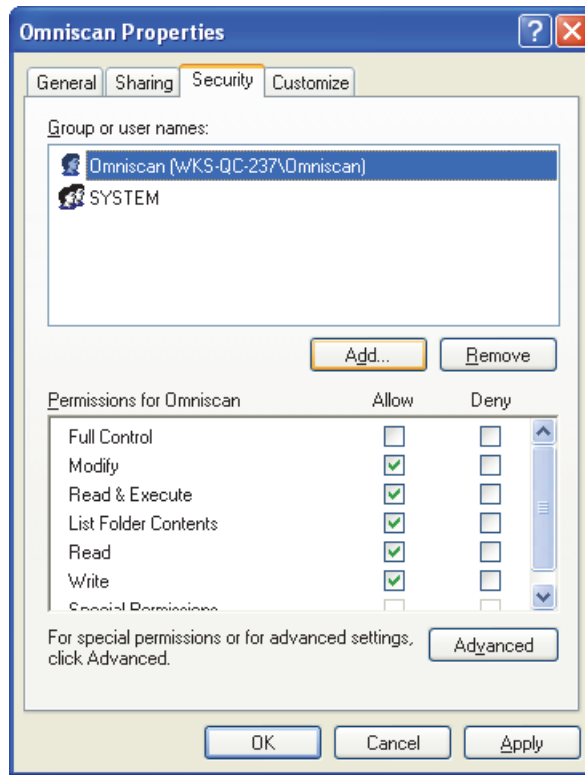


Figure 7-19 Security permissions for the Omniscan user

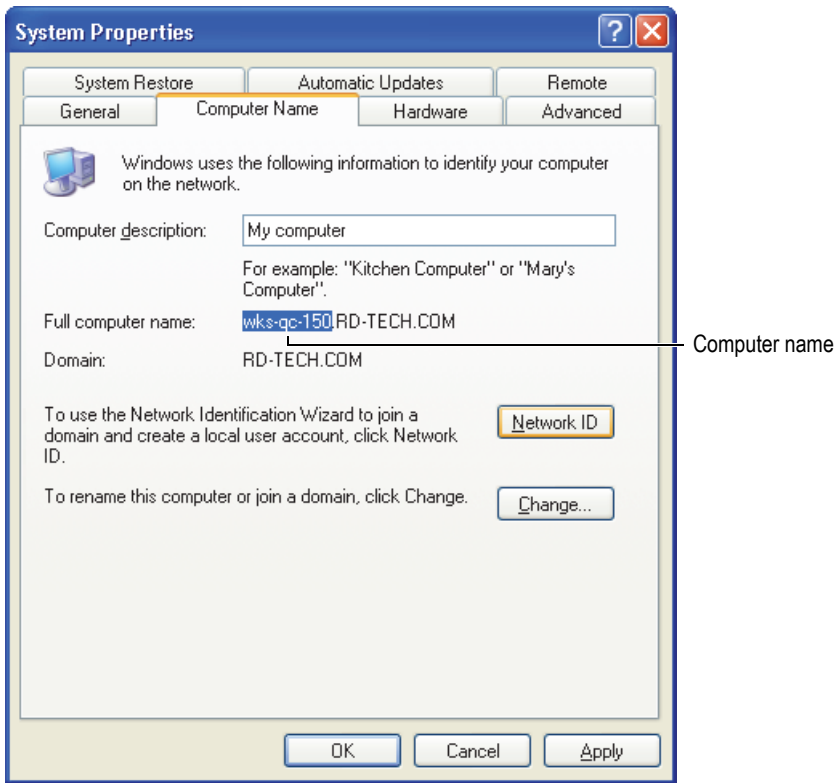
### 7.7.5 Configuring OmniScan MX2 Network Connection — Windows XP

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 223).



**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**.

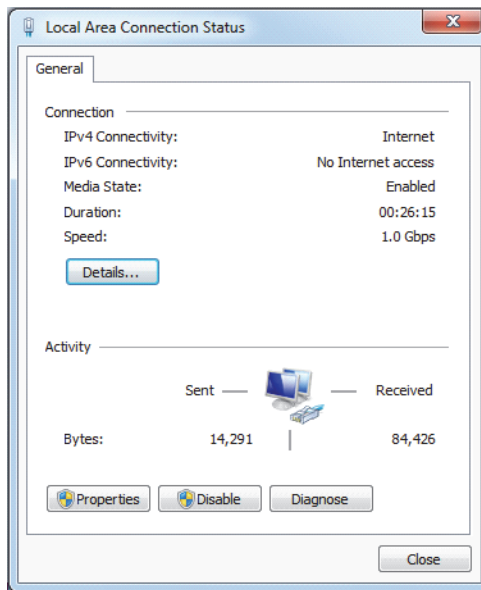
### 7.7.6 Configuring the Computer Network Connection — Windows 7

**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.

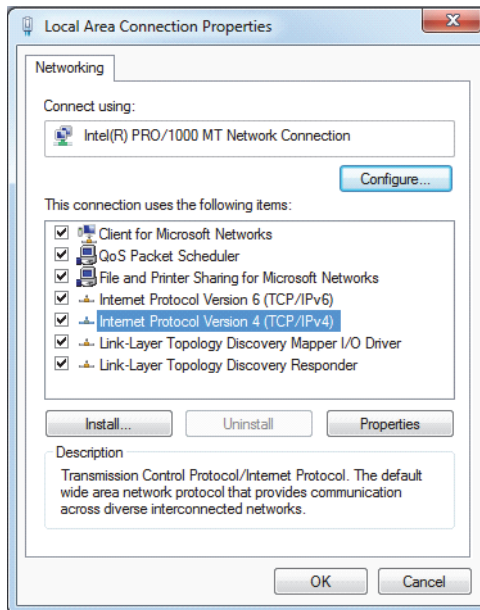
The **Local Area Connection Status** dialog box appears (see Figure 7-21 on page 224).



**Figure 7-21 The Local Area Connection Status dialog box (Windows 7)**

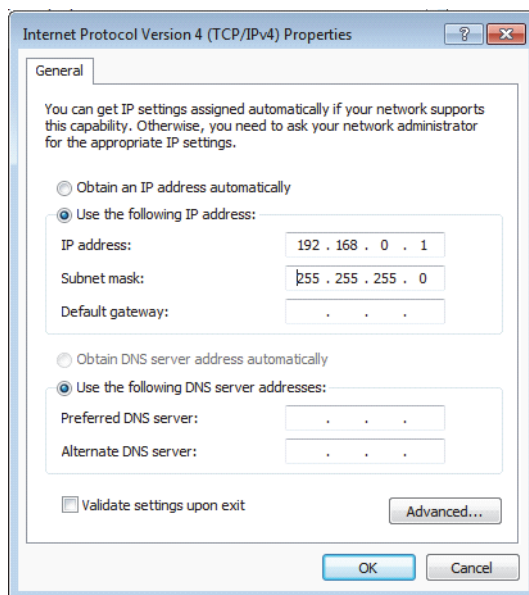
4. On the **General** tab, click **Properties**.

The Properties dialog box for the selected connection appears (see Figure 7-22 on page 225).



**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 226).



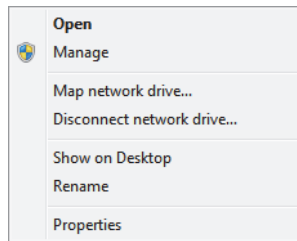
**Figure 7-23 The Internet Protocol Version 4 (TCP/IPv4) Properties dialog box (Windows 7)**

8. In the **IP address** box, type an OmniScan-compatible IP address (see “Configuring the OmniScan MX2 — Windows 7” on page 234).
9. In the **Subnet Mask** box, type an OmniScan-compatible subnet mask (see “Configuring the OmniScan MX2 — Windows 7” on page 234).
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**

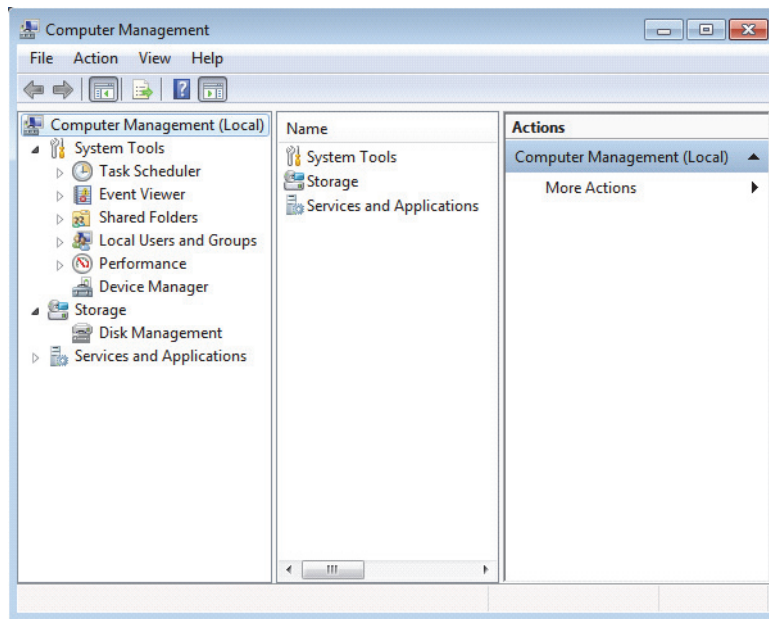
### **To create a user account on the computer**

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 227).



**Figure 7-24** The shortcut menu for the Computer icon (Windows 7)

The **Computer Management** dialog box appears (see Figure 7-25 on page 227).



**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 228).

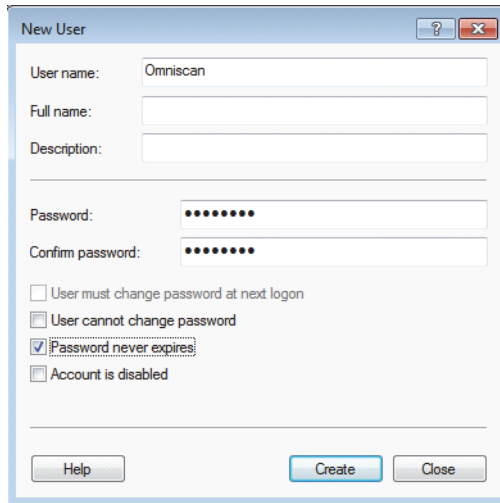


Figure 7-26 The New User dialog box (Windows 7)

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 229).
4. Click **Share**.

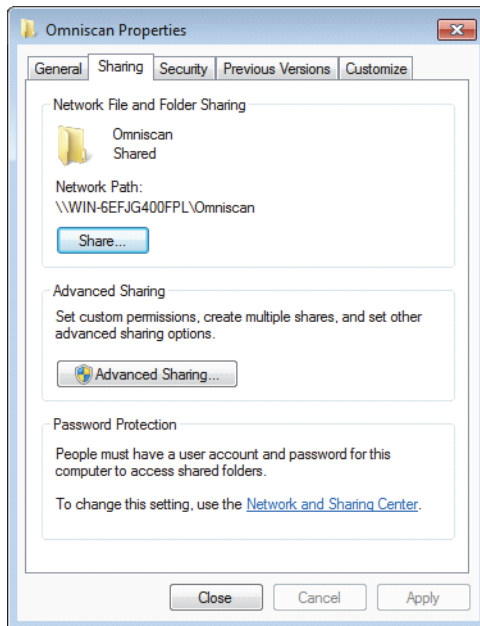
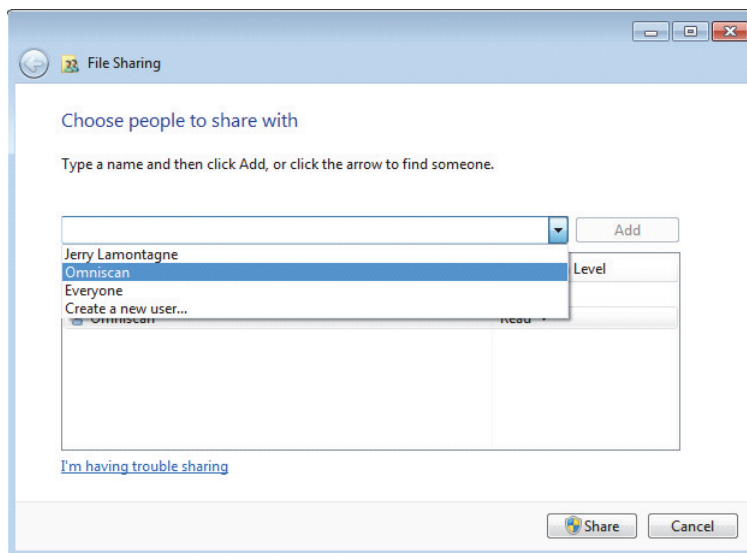


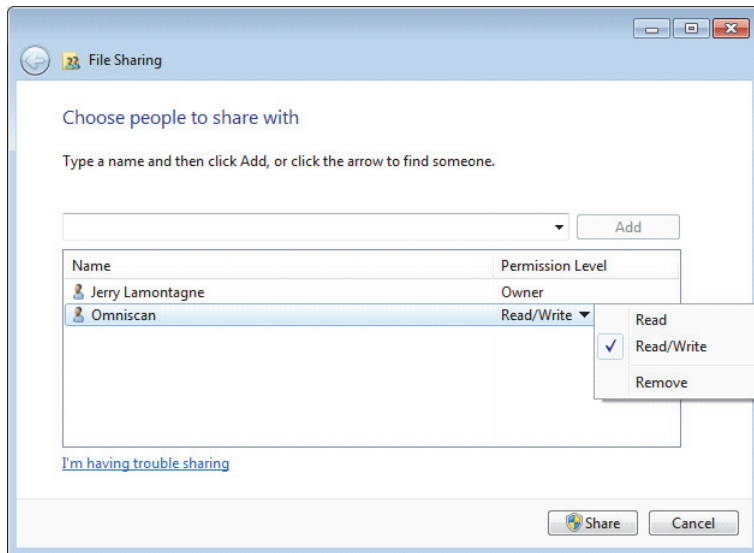
Figure 7-27 The Omniscan Properties dialog box (Windows 7)

5. In the **File Sharing** dialog box (see Figure 7-28 on page 230), 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 231).



**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 229).  
The **Advanced Sharing** dialog box appears (see Figure 7-30 on page 232).
9. Click the **Permissions** button.

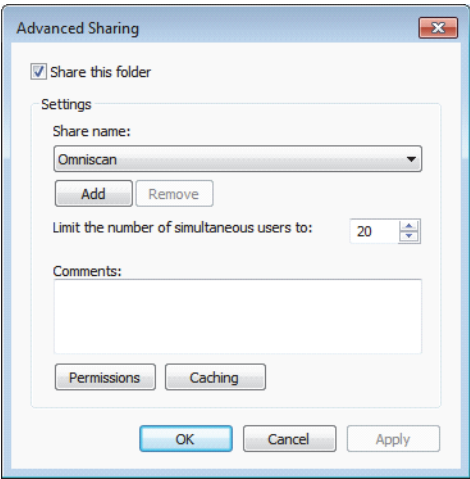


Figure 7-30 The Advanced Sharing dialog box (Windows 7)

10. In the **Permissions for Omniscan** dialog box, click **Add** (see Figure 7-31 on page 232).

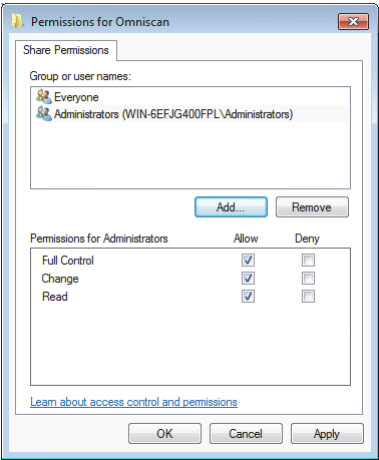
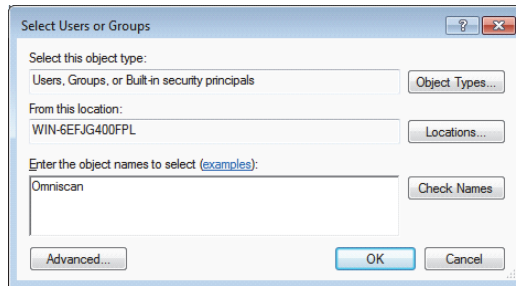


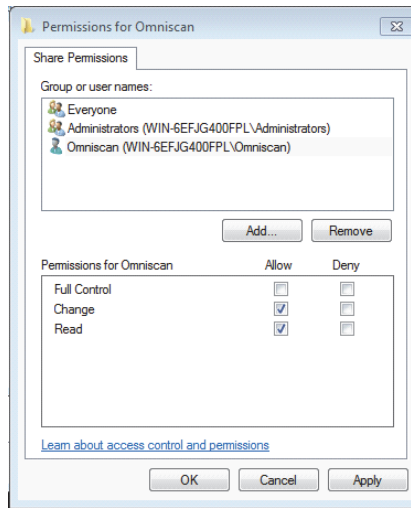
Figure 7-31 The Permissions for Omniscan dialog box (Windows 7)

11. In the **Select Users or Groups** dialog box, click the **Locations** button (see Figure 7-32 on page 233).



**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 233), 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 234).



**Figure 7-33 The Permissions for Omniscan dialog box (Windows 7)**

15. Close all open dialog boxes.

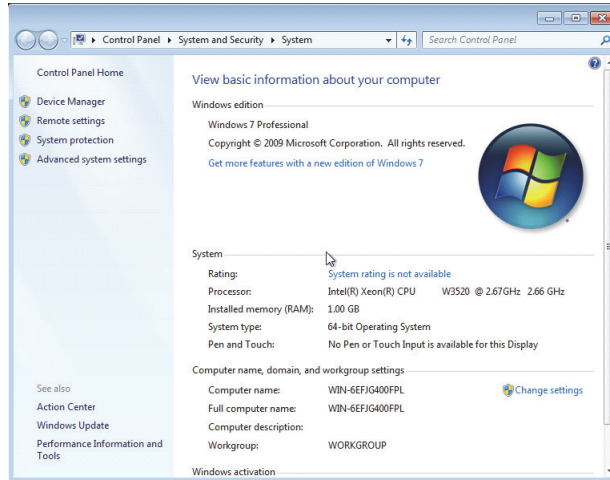
## 7.7.9 Configuring the OmniScan MX2 — Windows 7

### 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 235).



**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 208 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 208.
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.

---

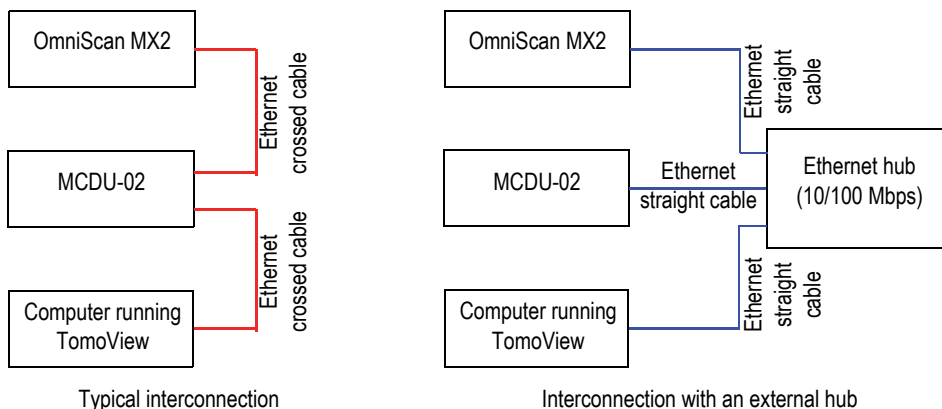
### NOTE

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 238).



**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 Evident 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.

---

<b>NOTE</b>
-------------

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 202).
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.

---

<b>TIP</b>
------------

“Online Help” on page 87 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 29 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 32 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 243) to open an existing report or preview and save a new report.
- **Format** (see “Format Category” on page 244) to define the content of the report.
- **User Fields** (see “User Fields Category” on page 246) 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 181.

**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 246.

**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 98 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 208
- “Connecting the OmniScan MX2 to a Network” on page 236

**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.

---

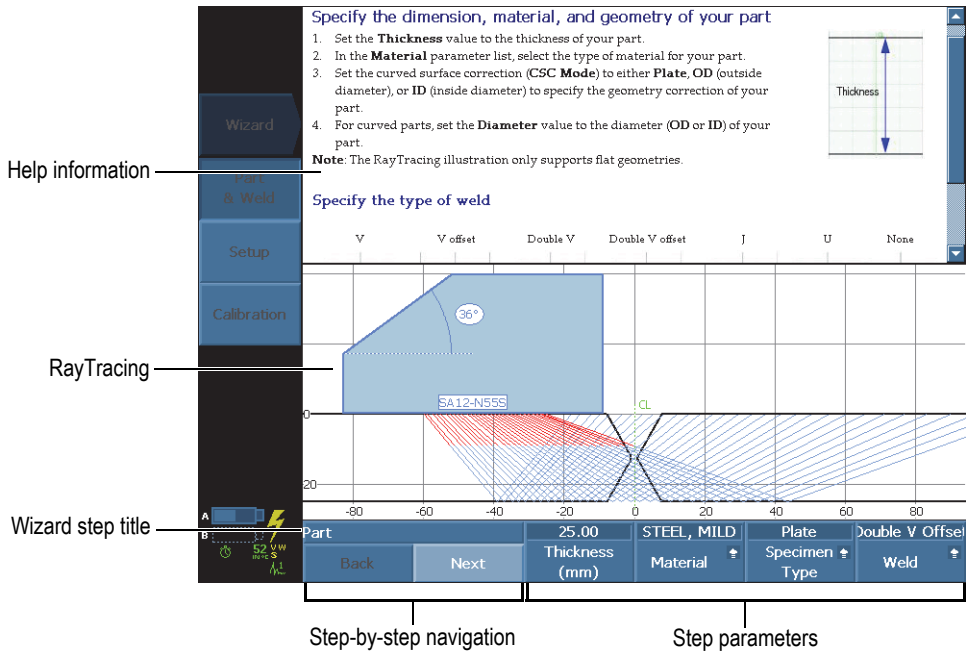
<b>NOTE</b>
-------------

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 251, 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.

---

<b>TIP</b>
------------

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**

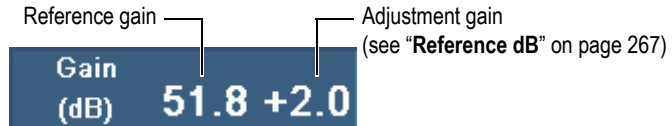
The **UT Settings** menu contains the OmniScan instrument ultrasound settings.

### **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 257).



**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 29 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**

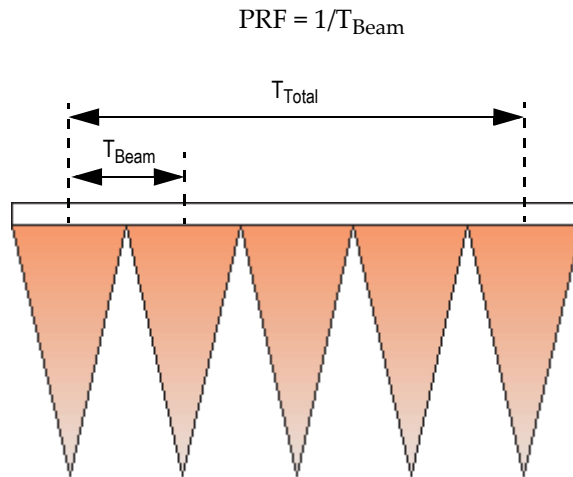
Calculates the recommended value for the current configuration (default value). You can enter a value manually.

For more details about the acquisition rate and PRF, see “Acquisition Rate and PRF” on page 259, “Acquisition Rate and PRF for a Multiple Group Configuration (OmniScan MX2 Only)” on page 260, “Acq Rate and PRF Indicator” on page 261, and “Maximizing the PRF value” on page 262.

**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  $T_{\text{Total}}$  and PRF is the inverse of  $T_{\text{Beam}}$ , which are expressed in the following calculations (illustrated in Figure 8-3 on page 260):

$$\text{Acq Rate} = 1/T_{\text{Total}}$$



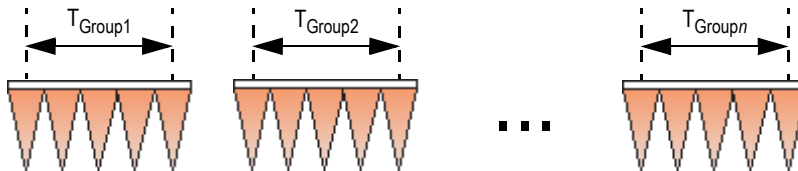
**Figure 8-3 The time intervals for the Acq Rate and PRF calculations (PA group)**

### **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 260, the acquisition rate for a multiple group configuration is calculated as follows:

$$Acq\ Rate = 1/(T_{Group1} + T_{Group2} + \dots + T_{Groupn})$$

$$PRF_{Beam} = 1/T_{Beam}$$



**Figure 8-4 The time intervals for Acq Rate in a multiple group configuration**

**Acq Rate and PRF Indicator**

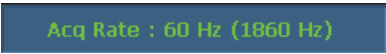
The **Acq Rate** indicator in the reading field section of the user interface displays the Acq Rate—expressed as a frequency [Hz] or speed [distance/s]—followed by the PRF value in parentheses (see Figure 8-5 on page 261). The syntax of the indicator is as follows:

**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



**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 261. See “Maximizing the PRF value” on page 262 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 261 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<sup>^</sup> - I<sup>^</sup>**) 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**

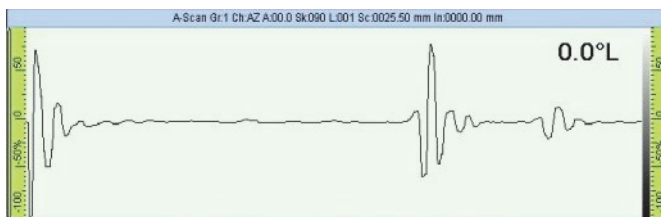
Used to select the appropriate filter value.

#### **Rectifier**

Used to select the rectification mode.

#### **RF** (radio frequency)

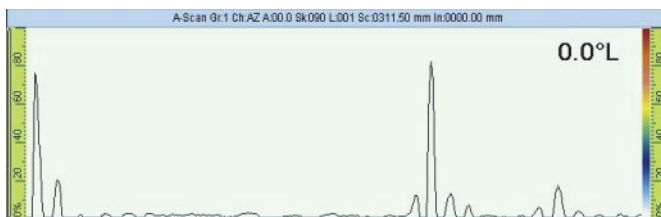
There is no rectification (see Figure 8-6 on page 264). Only available when **250 % mode** is turned off (see “250 % Mode Category” on page 340).



**Figure 8-6 RF mode**

**HW+** (half wave positive)

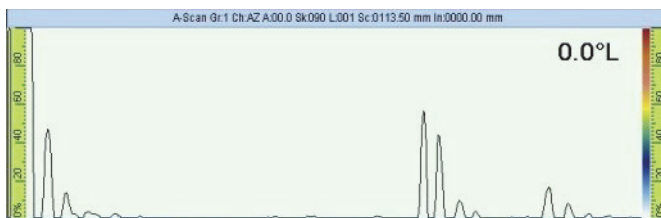
Only the positive values are rectified (see Figure 8-7 on page 264).



**Figure 8-7 HW+ rectification**

**HW-** (half wave negative)

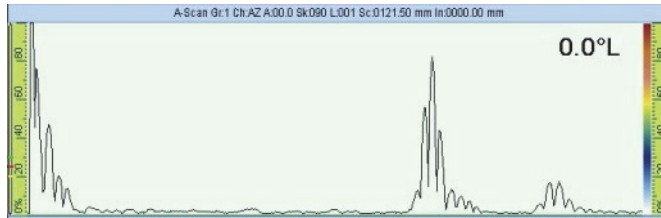
Only the negative values are rectified (see Figure 8-8 on page 264).



**Figure 8-8 HW- rectification**

**FW (full wave)**

The rectification values are all changed into absolute values (see Figure 8-9 on page 265).



**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 266).

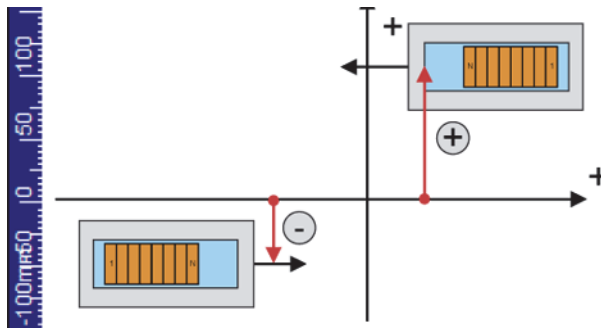


Figure 8-10 The scan offset

### Index Offset

The beam **Index Offset** (see Figure 8-11 on page 266) 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.

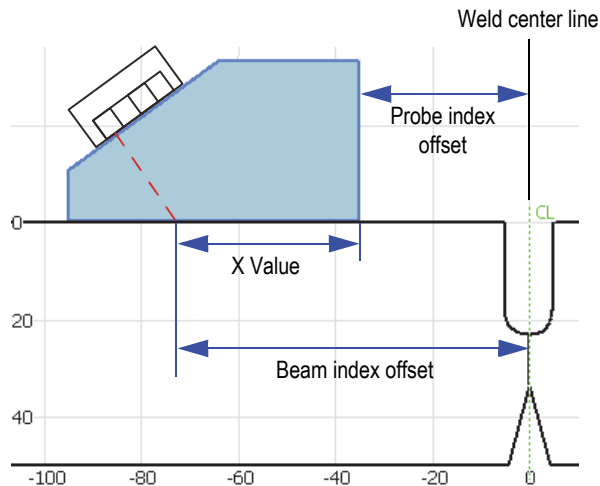


Figure 8-11 The beam index offset in a weld inspection example

**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%**.

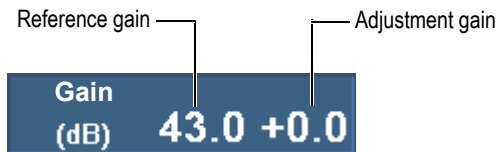
**Set to XX.X %**

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 268). 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 280 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 271).

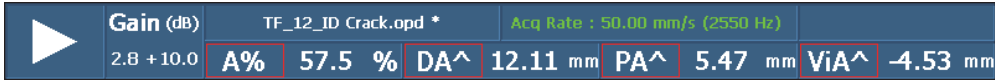


Figure 8-13 The four *s* at the top of the screen

### 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 272). 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 271). 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.

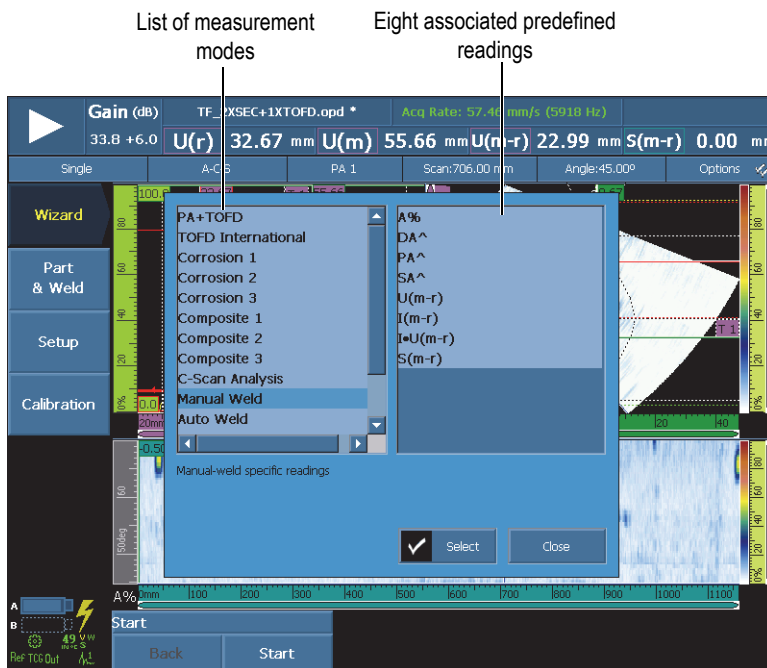


Figure 8-14 Dialog box for selecting measurement mode predefined readings

### Displayed Readings

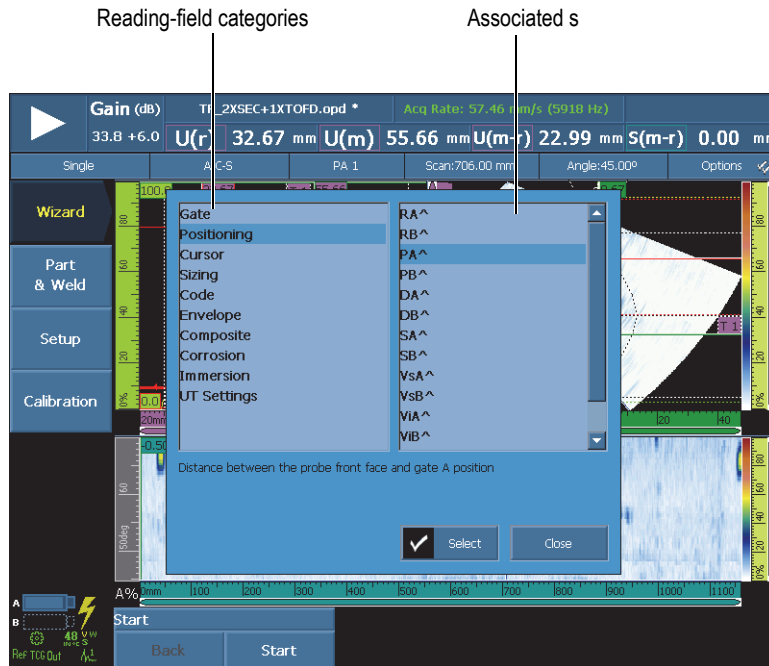
Used to select the list (**List 1/2** or **List 2/2**) of four readings you want to display at the top of the screen. The **Reading *n*** parameters on the right apply to the selected list.

### 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 273. 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 80 for details on outline colors.

---

<b>NOTE</b>
-------------

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 275).

#### **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 275 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).

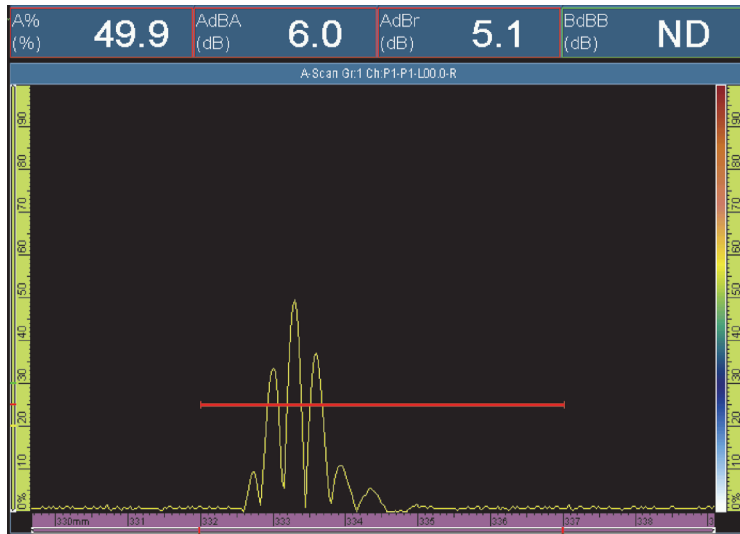
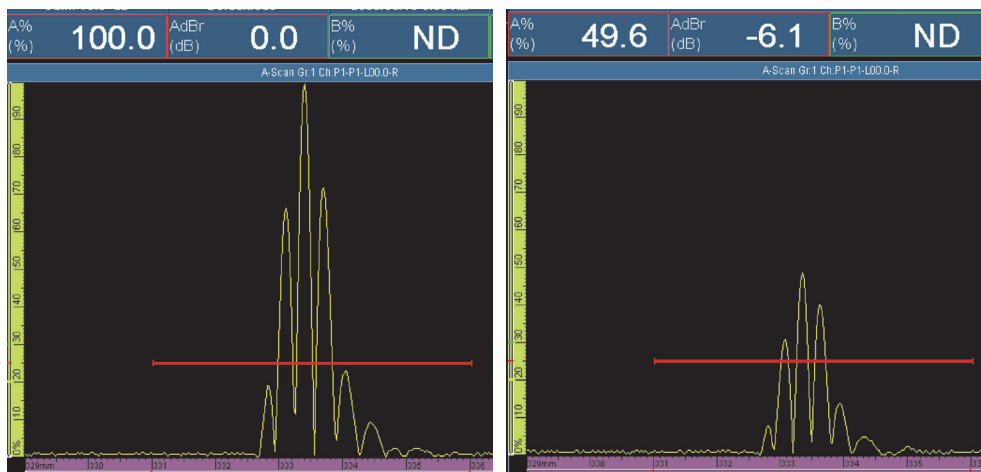


Figure 8-16 Example for the AdBA reading

### AdBr

Difference between the current signal amplitude in dB (*right* in Figure 8-17 on page 276) and the reference signal amplitude (*left* in Figure 8-17 on page 276) 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 276 (*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 277). The measurement taken depends on the gate mode that is selected (see “Gates Submenu” on page 307).

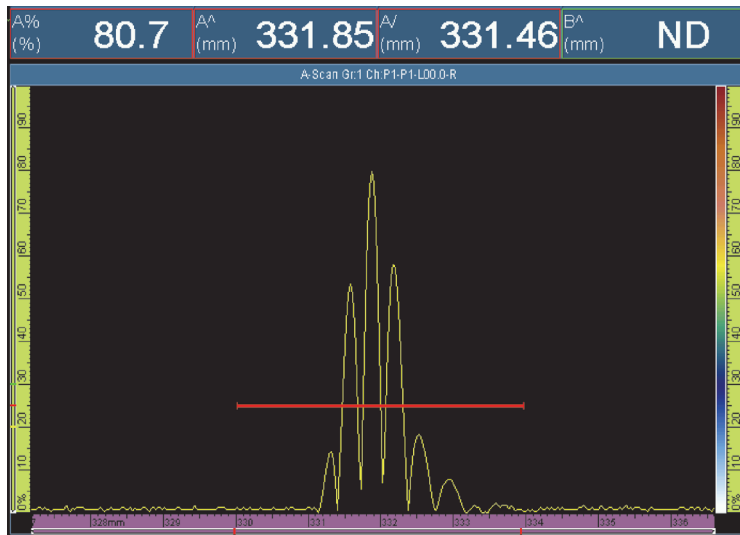


Figure 8-18 Example for the A%, A<sup>^</sup>, and A/ readings

#### B<sup>^</sup>

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 307).

#### I/I

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 307).

#### A%<sub>r</sub>

Difference between the current signal amplitude (%) and the reference signal amplitude detected in gate A.

#### B%<sub>r</sub>

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<sup>^</sup>**

Distance on the part surface between the beam exit point and the indication detected in gate A (see Figure 8-20 on page 279).

**RB<sup>^</sup>**

Distance on the part surface between the beam exit point and the indication detected in gate B (see the **RA<sup>^</sup>** definition).

**PA<sup>^</sup>**

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 279).

**PB<sup>^</sup>**

Distance on the part surface between the wedge (or probe) front face and the indication detected in gate B (see the **PA<sup>^</sup>** definition).

**DA<sup>^</sup>**

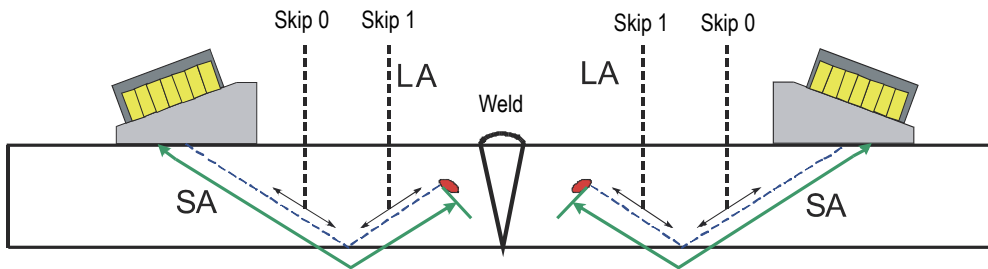
Depth in the part of the reflector producing the indication detected in gate A (see Figure 8-20 on page 279).

**DB<sup>^</sup>**

Depth in the part of the reflector producing the indication detected in gate B (see the **DA<sup>^</sup>** definition).

**SA<sup>^</sup>**

Sound path from the part entry point to the indication detected in gate A (see Figure 8-19 on page 278).



**Figure 8-19 Diagram of the LA and SA readings**

**SB<sup>^</sup>**

Sound path from the part entry point to the indication detected in gate B (see the **SA<sup>^</sup>** definition).

**VsA<sup>^</sup>**

Volumetric position of the indication detected in gate A in relation to the scan axis (see Figure 8-20 on page 279).

**VsB<sup>^</sup>**

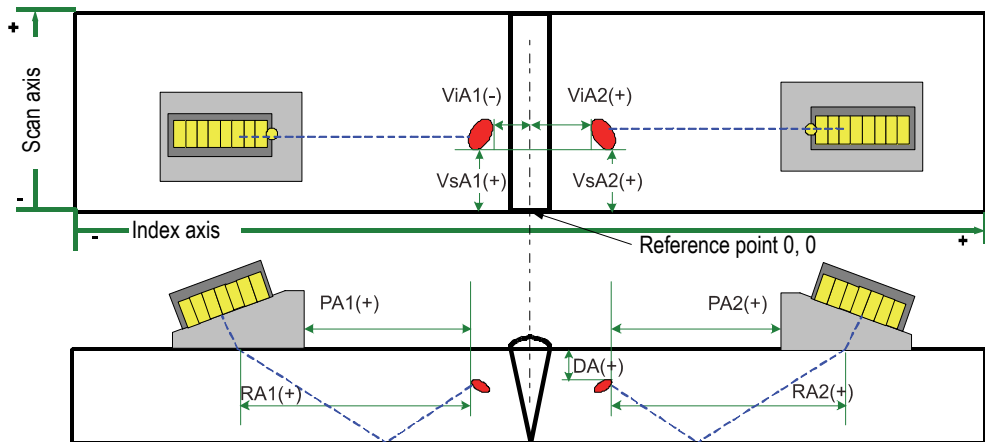
Volumetric position of the indication detected in gate B in relation to the scan axis (see the **VsA<sup>^</sup>** definition).

**ViA<sup>^</sup>**

Volumetric position of the indication detected in gate A on the index axis (see Figure 8-20 on page 279).

**ViB<sup>^</sup>**

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<sup>^</sup>**

Number of skips from the beam exit point to the reflector indication detected in gate A. In Figure 8-19 on page 278, the **LA<sup>^</sup>** value is 1, because the reflector is seen after one skip.

**LB<sup>^</sup>**

Number of skips from the beam exit point to the reflector indication detected in gate B (see the **LA<sup>^</sup>** definition).

---

<b>NOTE</b>
-------------

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 281).

**%(m)**

Amplitude value at the Measurement-cursor position (see Figure 8-21 on page 281).

**%(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 281).



**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 282).

**U(m)**

Position of the Measurement cursor on the ultrasound axis (see Figure 8-22 on page 282).

**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 282).



**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 283).



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)**

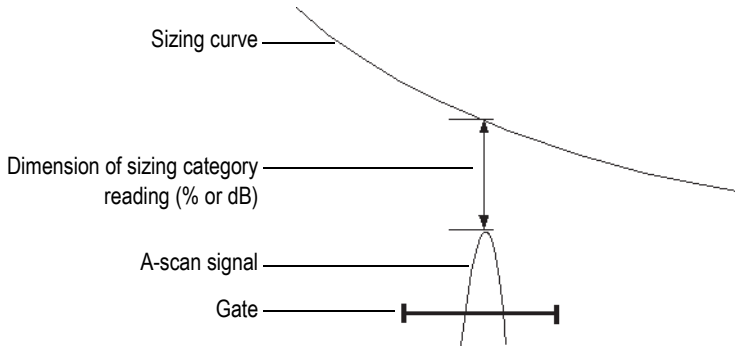
Difference between the Measurement and cursor in microseconds (Halfpath), excluding the beam delay in the wedge.

### 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 285).

#### 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 288). 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 288).

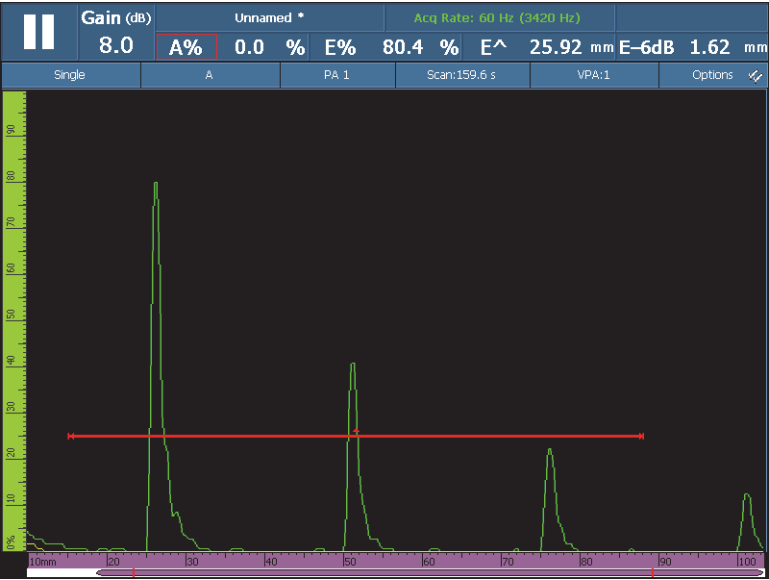


Figure 8-25 Example for the E%, E^, and E-6dB readings

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 290).

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 316). 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 290 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.

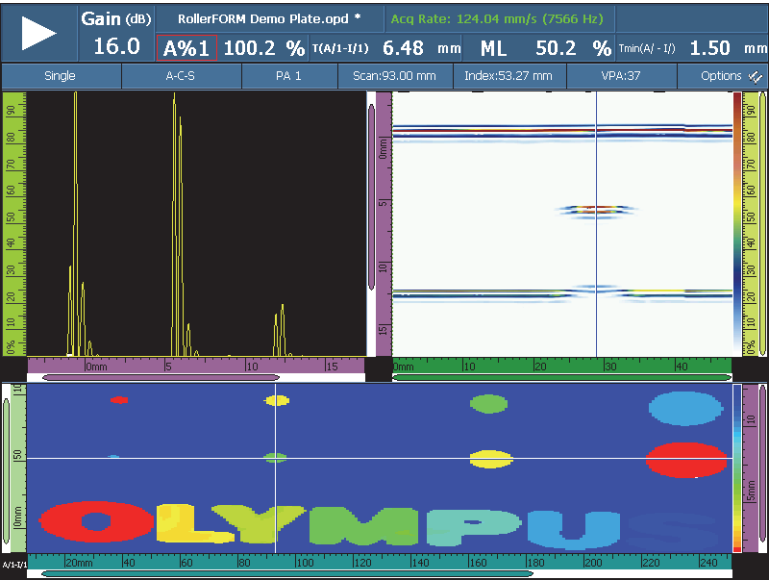


Figure 8-26 Example of corrosion inspection results, and T and ML readings

**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 291) presents detailed information related to the reflectors identified and recorded during the inspection, and it is used to create the inspection report.

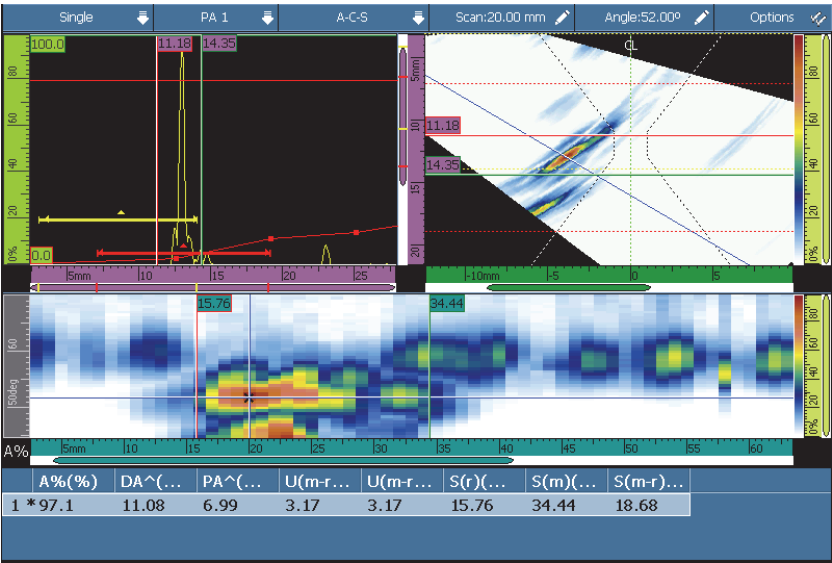


Figure 8-27 Example of an indication table

**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.

---

<b>TIP</b>
------------

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

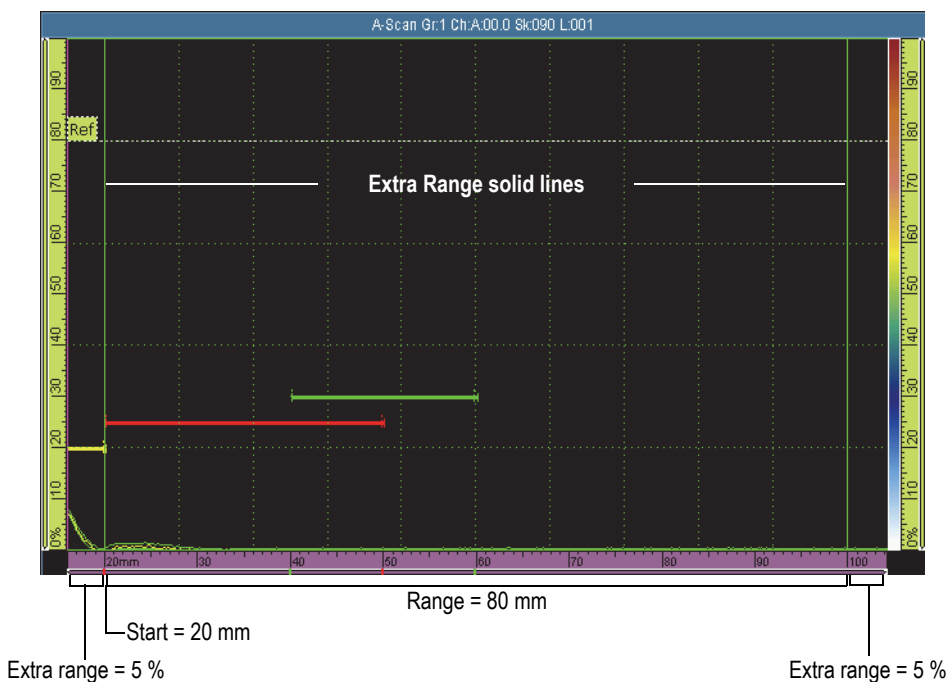
When the PRF value is above the display refresh rate, displays the A-scan built from the minimum thickness of all A-scans, including the ones that are not displayed (UT group only).

### 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 294, 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 296). 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.

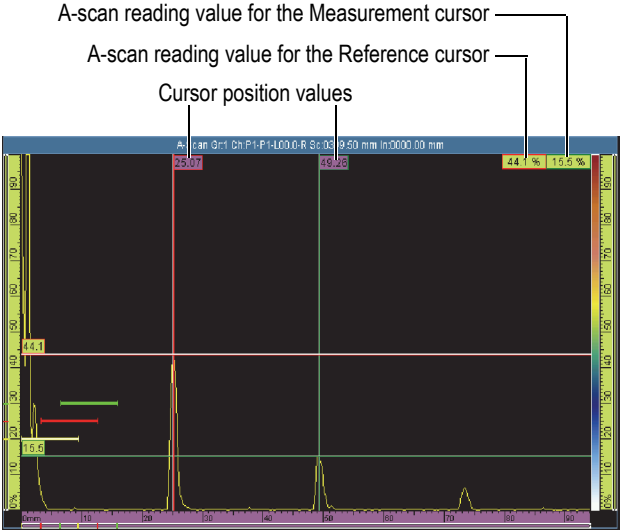


Figure 8-29 Cursor overlay example

**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 297).

**TIP**

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

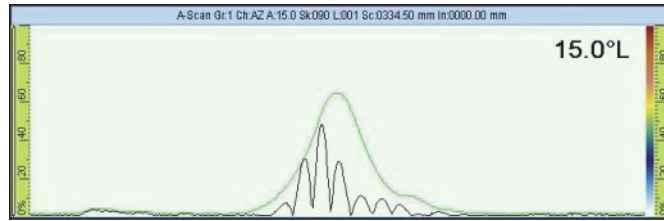


Figure 8-30 A-scan envelope curve

### 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.

---

<b>TIP</b>
------------

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 254 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 302.

---

<b>IMPORTANT</b>
------------------

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 302 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 304):

#### **Straight**

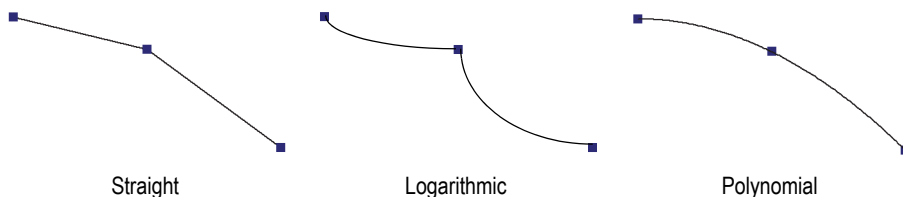
Linear interpolation (straight lines) between the DAC points.

#### **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 174 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 312 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 308).

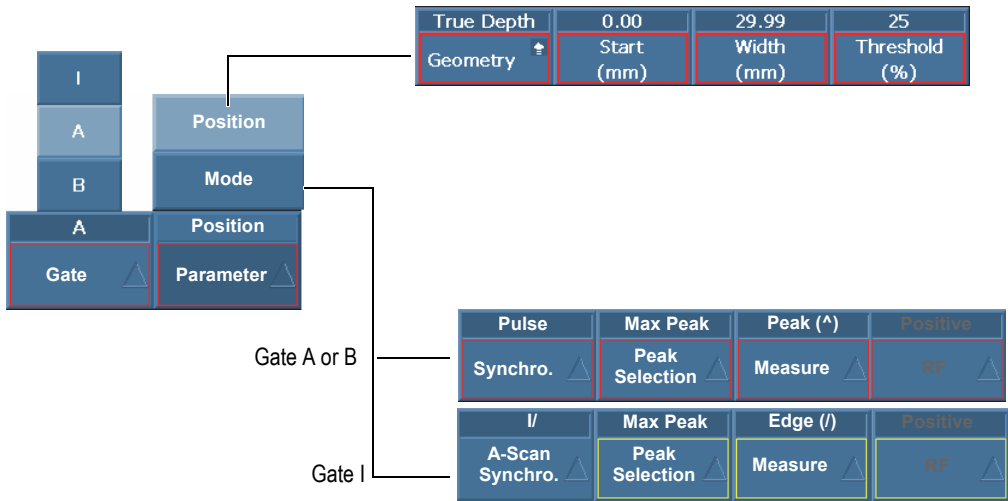


Figure 8-32 Available selection of gate parameters

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 310):

**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.

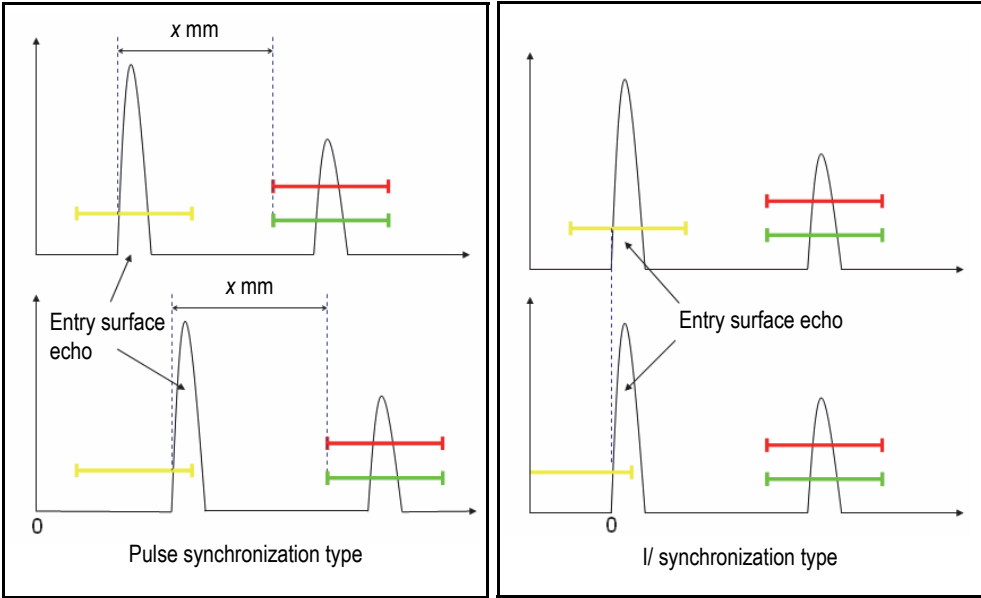


Figure 8-33 A-scan synchronization types for gate I

**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 311):

- 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.

---

<b>NOTE</b>
-------------

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 314.

### 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.

---

<b>NOTE</b>
-------------

For more details on alarms, see “Setting an Alarm” on page 173.

---

### 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 173.

**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.

---

<b>IMPORTANT</b>
------------------

---

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 317).

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**

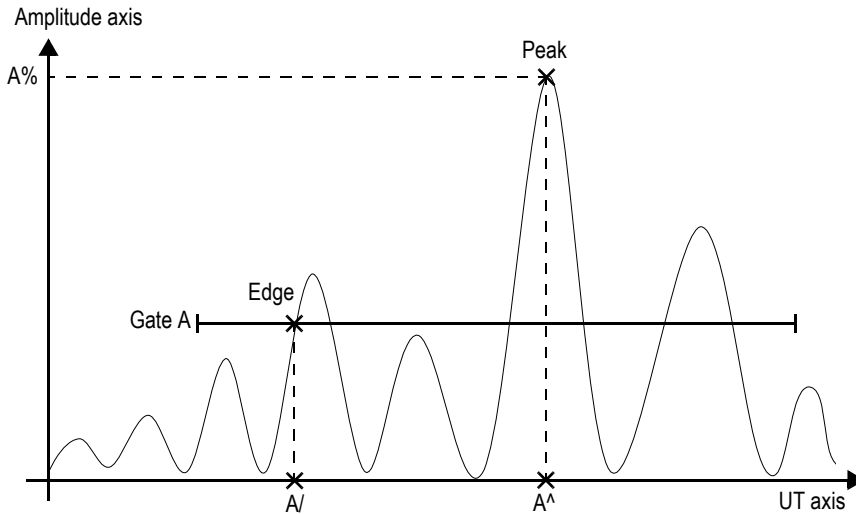
Crossing point on gate B; available when you select **Gates/Alarms > Gates > Parameter = Mode** and **Gates/Alarms > Gates > Measure = Edge(/)** for gate B.

**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.

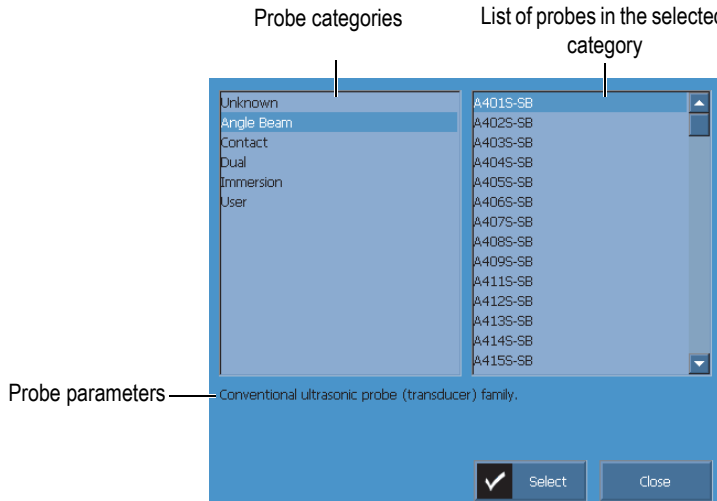
#### Select Transmitter (available in pitch-and-catch)

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 319). 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 319 for details).

**Edit Wedge:** Opens the Wedge Definition Manager, enabling you to define a new wedge (see “Wedge Definition Manager” on page 322 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)

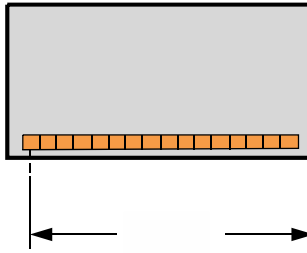
For circular probe elements, used to set the probe-element diameter.

**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 321).



**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.

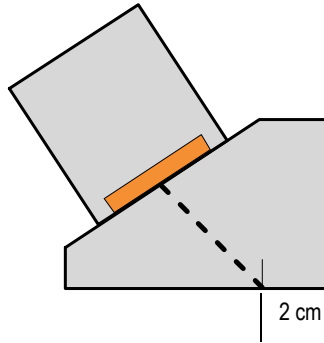
#### **Wave Type** (UT group only)

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 323). 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)

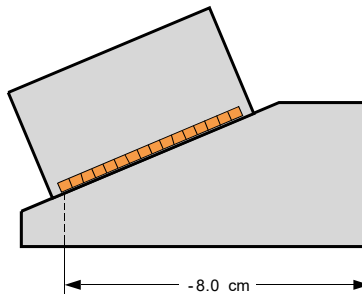
Used to set the orientation of the wedge (**Normal** or **Reverse**).

**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 323).



**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 325).

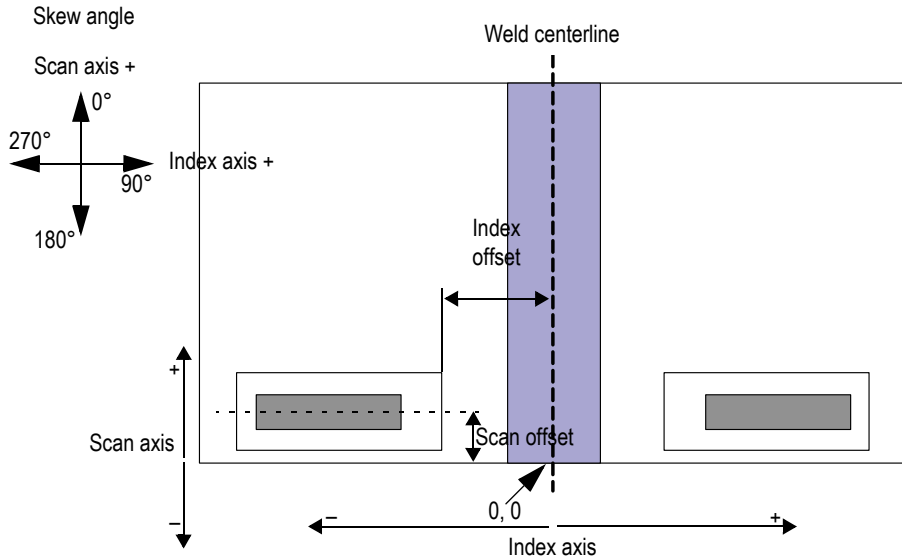
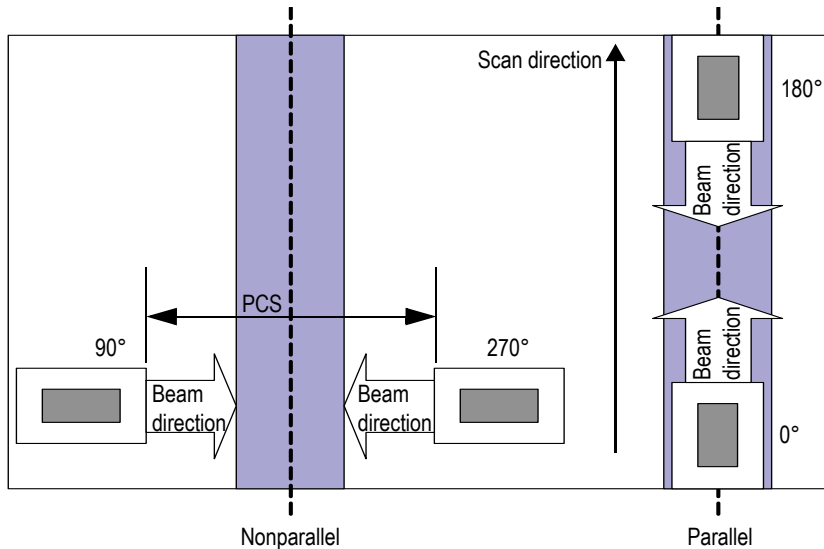


Figure 8-39 Scan and index offsets

## 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 326). 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 326).

#### 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

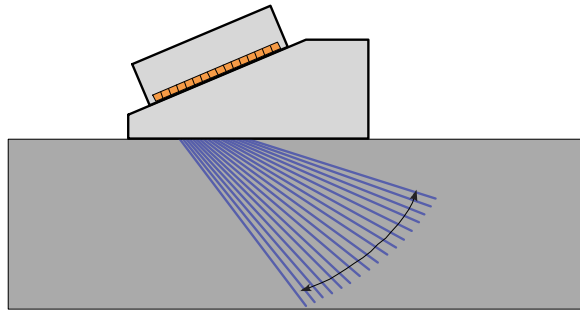
The **Configuration** submenu manages the parameters related to the scan types that will be performed by the probe.

### 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 328).



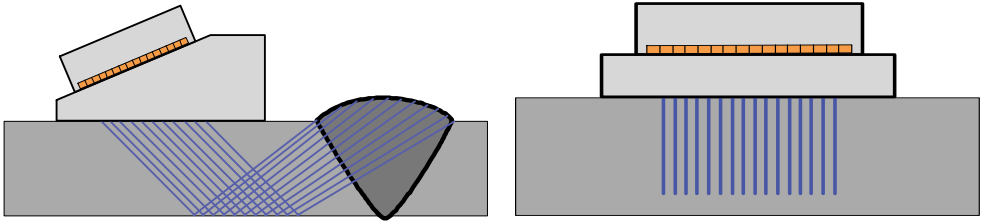
**Figure 8-41 Sectorial scanning**

### Linear

Provides a linear scan at a configurable angle (see Figure 8-42 on page 329). You can use this mode at a zero-degree angle if you do not need overlapping scans.

### Linear at 0°

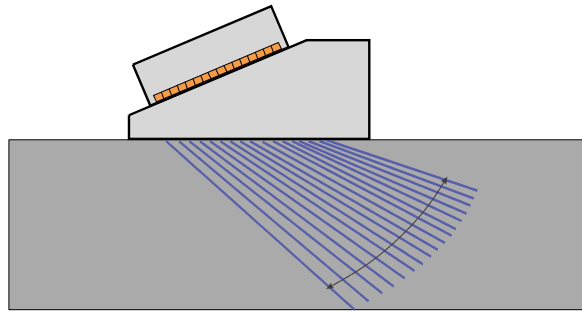
Provides a volume corrected linear sweep at zero degrees (see Figure 8-42 on page 329). Use this mode when you perform overlapping scans. This feature is available with one group only.



**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 329).



**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**.

---

<b>TIP</b>
------------

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.

---

<b>NOTE</b>
-------------

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

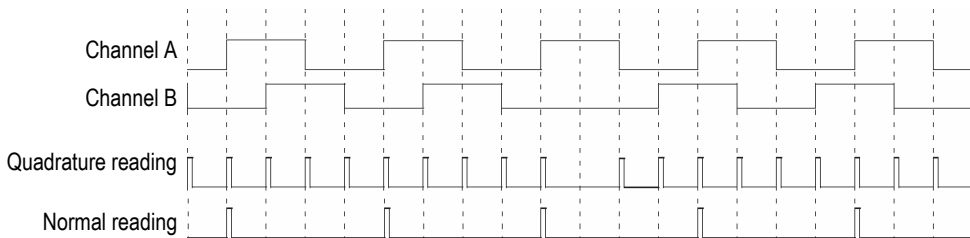
Used to select the type of the currently selected encoder. There are two encoder hardware inputs that can be configured independently. Each encoder input has two channels, A and B, enabling a dual-channel encoder for quadrature readings of the resolution.

## 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 333), 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 Evident 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 Evident 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 29 or Table 3 on page 32 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 338)
- **Gates** (see “Gates Category” on page 340)
- **250 % Mode** (see “250 % Mode Category” on page 340)
- **Interpolation** (see “Interpolation Category” on page 340)
- **Interleave** (see “Interleave Category” on page 340)

#### 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 341)
- **Screen** (see “Screen Category” on page 341)
- **System** (see “System Category” on page 342)
- **Network Settings** (see “Network Settings Category (OmniScan MX2 Only)” on page 342)
- **External Storage** (see “External Storage Category (OmniScan MX2 Only)” on page 343)
- **Help** (see “Help Category” on page 344) [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 91 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**

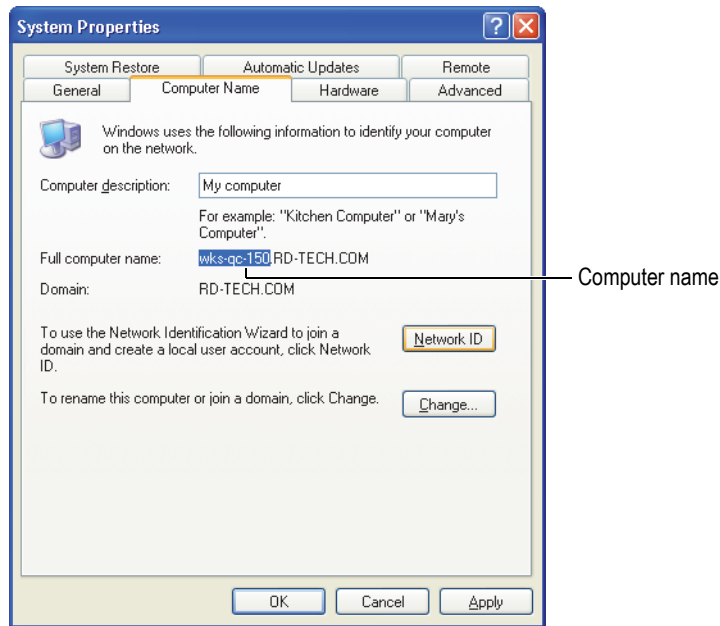
Applies the changes made to the parameters of the **Network** submenu.

**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 344. See "Configuring OmniScan MX2 Network Connection — Windows XP" on page 221 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 198 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 114 for a detailed procedure.

---

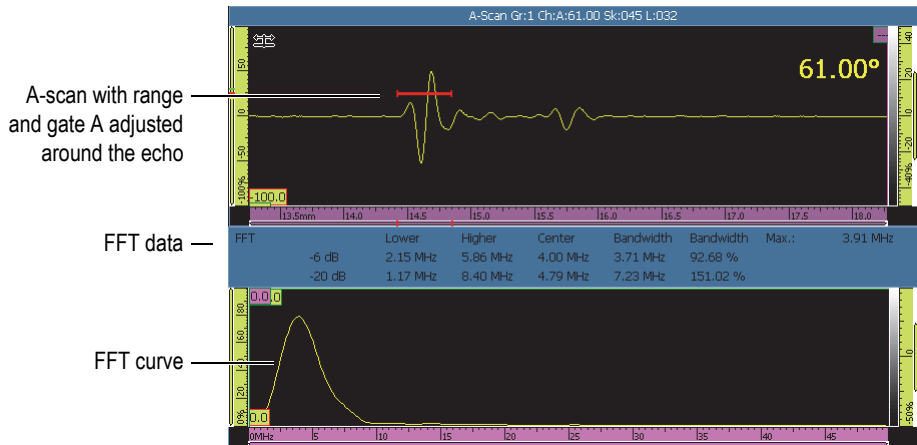
#### IMPORTANT

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 347). 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 347].

### **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 349).

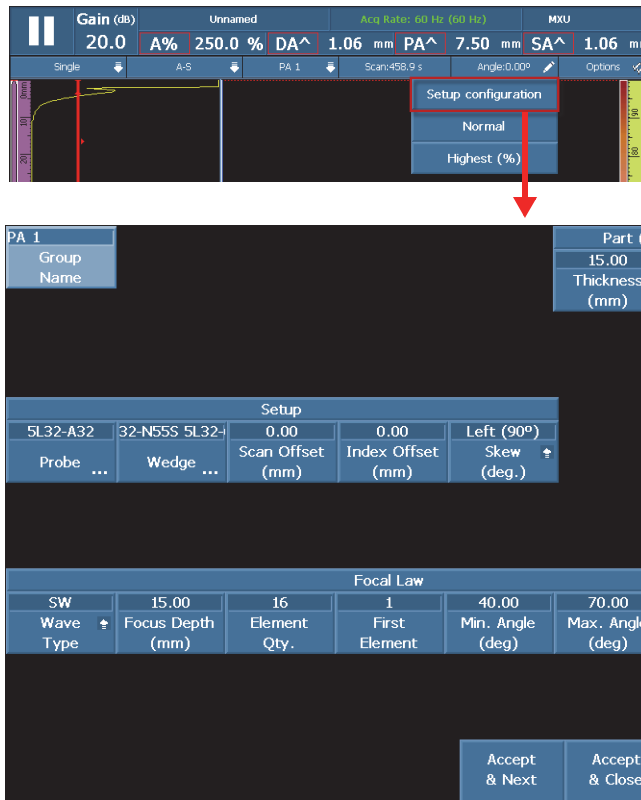



Figure 8-47 The Setup Configuration screen

### 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.

---

<b>NOTE</b>
-------------

- 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 351).



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 .....	20
Figure 1-2	Front panel controls of the OmniScan SX .....	21
Figure 1-3	The Shut Down button .....	22
Figure 1-4	Saving the setup .....	23
Figure 1-5	Requirements for UT probe connection .....	24
Figure 1-6	Requirements for PA probe connection .....	24
Figure 1-7	UT probe connection .....	25
Figure 1-8	PA probe connection .....	25
Figure 1-9	The main controls .....	26
Figure 1-10	Functions on MX2 keypad .....	29
Figure 1-11	Functions on SX keypad .....	31
Figure 1-12	OmniScan MX2: Example pop-up button for the Gain function key .....	33
Figure 1-13	Virtual keyboard with numeric keypad ( <i>left</i> ) or without ( <i>right</i> ) .....	35
Figure 1-14	Example of selection zone overlaps .....	36
Figure 1-15	The visual reference on the gate .....	38
Figure 1-16	The power indicator .....	39
Figure 1-17	OmniScan MX2 acquisition indicator light .....	40
Figure 1-18	OmniScan SX acquisition indicator light .....	40
Figure 2-1	Example screen image of the user interface .....	45
Figure 2-2	The title bar shortcut menus .....	46
Figure 2-3	The readings shortcut menu .....	53
Figure 2-4	The File Manager shortcut menu .....	54
Figure 2-5	The System shortcut menu .....	55
Figure 2-6	The acquisition parameters .....	55
Figure 2-7	The OmniScan MXU software window components .....	56
Figure 2-8	The menu hierarchy and identification syntax .....	57
Figure 2-9	Menu item groups .....	59
Figure 2-10	The reading fields and data screen .....	61
Figure 2-11	The parameters in the reading-field area .....	62
Figure 2-12	The information available in a reading field .....	62

Figure 2-13	The Gain value field .....	63
Figure 2-14	The Gain value field with an active reference .....	63
Figure 2-15	An example of the status indicators .....	64
Figure 2-16	OmniScan MX2 battery charge status example .....	66
Figure 2-17	OmniScan SX battery charge status example .....	66
Figure 2-18	The acquisition modes .....	68
Figure 2-19	Example of ultrasonic scan views .....	71
Figure 2-20	The RayTracing graphical elements .....	73
Figure 2-21	Example of RayTracing in the Setup wizard .....	74
Figure 2-22	Example of indications in the RayTracing view .....	76
Figure 2-23	The Multiple Group layouts .....	77
Figure 2-24	Example of multiple views with various rulers/scales .....	78
Figure 2-25	Three readings outlined in red, and one in purple .....	81
Figure 2-26	The purple color for ultrasound axis and mode button .....	83
Figure 2-27	Editing a parameter value with the increment/decrement step .....	86
Figure 2-28	The Compression feature symbol .....	87
Figure 3-1	File Name parameter button .....	94
Figure 3-2	Example of header note ( <i>top</i> ) and footer note ( <i>bottom</i> ) in a report .....	99
Figure 3-3	Example of Label and Content parameters in a report .....	99
Figure 4-1	Elements of a wizard step .....	104
Figure 4-2	The probe selection dialog box .....	106
Figure 4-3	The lists of wedges in the wedge selection dialog box .....	107
Figure 4-4	The probe reference point measurement .....	109
Figure 4-5	Probe-element pitch .....	109
Figure 4-6	Wedge angle .....	111
Figure 4-7	The primary offset measurement .....	111
Figure 4-8	The secondary offset .....	112
Figure 4-9	The height of the first element .....	112
Figure 4-10	Example of FFT curve and data for a 5 MHz probe .....	115
Figure 4-11	Example of a Probe Characterization section in a report .....	116
Figure 4-12	Example adaptor to connect UT probes to PA connector .....	118
Figure 5-1	Choice of parameters to calibrate in Select Calibration .....	122
Figure 5-2	Choice of parameters to calibrate — TOFD group .....	122
Figure 5-3	Example of reflector signals in a linear scan .....	127
Figure 5-4	Setting the gate on the second signal .....	128
Figure 5-5	Velocity indicator (V) in green after velocity calibration .....	128
Figure 5-6	Building the envelope for the wedge delay calibration .....	130
Figure 5-7	Wedge delay indicator (W) after calibration for UT group .....	131
Figure 5-8	The velocity (V) and wedge delay (W) indicators .....	135
Figure 5-9	Building the envelope for the wedge-delay calibration .....	137
Figure 5-10	Wedge delay indicator (W) after calibration for all focal laws .....	138
Figure 5-11	Building the envelope for the first angle/VPA .....	139

Figure 5-12	Wedge delay indicator ( $W_p$ ) after calibration for 2 or 3 focal laws .....	140
Figure 5-13	Reference reflector signal before and after sensitivity calibration .....	141
Figure 5-14	Building the envelope for the sensitivity calibration .....	143
Figure 5-15	Sensitivity indicator (S) in green after calibration .....	144
Figure 5-16	Building the envelope for the sensitivity calibration of the first angle ....	146
Figure 5-17	Sensitivity indicator ( $S_i$ ) after calibration for 2 or 3 focal laws .....	146
Figure 5-18	Calibration blocks for the DAC calibration .....	147
Figure 5-19	Set Gate A on Echo for a PA group .....	149
Figure 5-20	Calibration blocks for the TCG calibration .....	150
Figure 5-21	Set Gate A on Echo for a PA group .....	152
Figure 5-22	Set Gate A over the signal envelope .....	156
Figure 5-23	Example of DGS sizing curves .....	157
Figure 5-24	Completed DGS calibration status indicator .....	157
Figure 5-25	Completed AWS calibration status indicator .....	159
Figure 6-1	Choice of transmission/reception modes .....	164
Figure 6-2	An example of the PA-UT Weld (A-B-S) layout .....	167
Figure 6-3	The Export C-scan function .....	170
Figure 6-4	The Profile B-scan view .....	171
Figure 6-5	The Export function .....	172
Figure 6-6	The first point of the DAC curve .....	175
Figure 6-7	The DAC curve on the next signal .....	176
Figure 6-8	Verification of the DAC curve .....	177
Figure 6-9	An envelope with peaks at 80 % .....	179
Figure 6-10	Dialog box for choosing predefined readings .....	180
Figure 6-11	Example of an indication table .....	181
Figure 6-12	Selecting an entry in the indication table .....	182
Figure 6-13	Example of cursors .....	184
Figure 6-14	Example of a J-type weld geometry overlay .....	187
Figure 6-15	A green grid on the A-scan view .....	188
Figure 6-16	The palette on the right side of the A-scan view .....	189
Figure 6-17	Scan and Index reading fields .....	191
Figure 6-18	The Index Step function .....	193
Figure 7-1	The File Manager areas .....	196
Figure 7-2	The parent folder icon .....	197
Figure 7-3	The source pane highlighted .....	198
Figure 7-4	The Report folder path .....	200
Figure 7-5	The path for the custom report files .....	200
Figure 7-6	Gate A on the ball signal .....	207
Figure 7-7	The Local Area Connection Status dialog box (Windows XP) .....	210
Figure 7-8	The Properties dialog box for selected connection (Windows XP) .....	211
Figure 7-9	The Internet Protocol (TCP/IP) Properties dialog box (Windows XP) .....	212

Figure 7-10	The shortcut menu for the My Computer icon .....	213
Figure 7-11	The Computer Management dialog box (Windows XP) .....	213
Figure 7-12	The New User dialog box (Windows XP) .....	214
Figure 7-13	The Sharing tab of the Omniscan Properties dialog box (Windows XP) .....	215
Figure 7-14	The Folder Options dialog box (Windows XP) .....	216
Figure 7-15	The Permissions for Omniscan dialog box (Windows XP) .....	217
Figure 7-16	The Select Users or Groups dialog box (Windows XP) .....	218
Figure 7-17	The Permissions for Omniscan dialog box (Windows XP) .....	219
Figure 7-18	The Security tab of the Omniscan Properties dialog box (Windows XP) .....	220
Figure 7-19	Security permissions for the Omniscan user .....	221
Figure 7-20	The computer name found in the System Properties dialog box .....	223
Figure 7-21	The Local Area Connection Status dialog box (Windows 7) .....	224
Figure 7-22	The Local Area Connection Properties dialog box (Windows 7) .....	225
Figure 7-23	The Internet Protocol Version 4 (TCP/IPv4) Properties dialog box (Windows 7) .....	226
Figure 7-24	The shortcut menu for the Computer icon (Windows 7) .....	227
Figure 7-25	The Computer Management dialog box (Windows 7) .....	227
Figure 7-26	The New User dialog box (Windows 7) .....	228
Figure 7-27	The Omniscan Properties dialog box (Windows 7) .....	229
Figure 7-28	The File Sharing dialog box (Windows 7) .....	230
Figure 7-29	Setting Permissions in the File Sharing dialog box (Windows 7) .....	231
Figure 7-30	The Advanced Sharing dialog box (Windows 7) .....	232
Figure 7-31	The Permissions for Omniscan dialog box (Windows 7) .....	232
Figure 7-32	The Select Users or Groups dialog box (Windows 7) .....	233
Figure 7-33	The Permissions for Omniscan dialog box (Windows 7) .....	234
Figure 7-34	Computer name in System Properties dialog box (Windows 7) .....	235
Figure 7-35	Interconnections between OmniScan MX2, MCDU, and computer .....	238
Figure 8-1	Wizard step components .....	251
Figure 8-2	Gain value display .....	257
Figure 8-3	The time intervals for the Acq Rate and PRF calculations (PA group) ....	260
Figure 8-4	The time intervals for Acq Rate in a multiple group configuration .....	260
Figure 8-5	The Acq Rate indicator in the reading field section .....	261
Figure 8-6	RF mode .....	264
Figure 8-7	HW+ rectification .....	264
Figure 8-8	HW- rectification .....	264
Figure 8-9	FW rectification .....	265
Figure 8-10	The scan offset .....	266
Figure 8-11	The beam index offset in a weld inspection example .....	266
Figure 8-12	The Gain value field with an active reference .....	268
Figure 8-13	The four s at the top of the screen .....	271

Figure 8-14	Dialog box for selecting measurement mode predefined readings .....	272
Figure 8-15	Reading field selection dialog box .....	273
Figure 8-16	Example for the AdBA reading .....	275
Figure 8-17	Example for the AdBr reading .....	276
Figure 8-18	Example for the A%, A <sup>^</sup> , and A/ readings .....	277
Figure 8-19	Diagram of the LA and SA readings .....	278
Figure 8-20	Diagram of the RA, PA, DA, ViA, and VsA readings .....	279
Figure 8-21	Example for the %(r), %(m), and %(m-r) readings .....	281
Figure 8-22	Example for the U(r), U(m), and U(m-r) readings .....	282
Figure 8-23	Example for the %(U(r)) reading .....	283
Figure 8-24	Illustration of the sizing category readings .....	285
Figure 8-25	Example for the E%, E <sup>^</sup> , and E-6dB readings .....	288
Figure 8-26	Example of corrosion inspection results, and T and ML readings .....	290
Figure 8-27	Example of an indication table .....	291
Figure 8-28	Example of an extra range set to "5% - 5%" .....	294
Figure 8-29	Cursor overlay example .....	296
Figure 8-30	A-scan envelope curve .....	297
Figure 8-31	Illustration of Curve Type interpolation choices .....	304
Figure 8-32	Available selection of gate parameters .....	308
Figure 8-33	A-scan synchronization types for gate I .....	310
Figure 8-34	Description of the source codes for gate A .....	317
Figure 8-35	The probe selection dialog box .....	319
Figure 8-36	The probe reference point measurement .....	321
Figure 8-37	The wedge reference point measurement .....	323
Figure 8-38	The primary offset measurement .....	323
Figure 8-39	Scan and index offsets .....	325
Figure 8-40	The parallel and nonparallel scan types .....	326
Figure 8-41	Sectorial scanning .....	328
Figure 8-42	Linear scanning with an angle (left) and at zero degrees (right) .....	329
Figure 8-43	Compound scanning .....	329
Figure 8-44	Quadrature and normal reading of the resolution .....	333
Figure 8-45	Computer name in System Properties dialog box (Windows XP) .....	344
Figure 8-46	FFT curve and data example .....	347
Figure 8-47	The Setup Configuration screen .....	349
Figure 8-48	The Scan Configuration parameters .....	351



---

## List of Tables

---

Table 1	Main control functions for various contexts .....	27
Table 2	Key shortcuts for OmniScan MX2 .....	29
Table 3	Key shortcuts for OmniScan SX .....	32
Table 4	OmniScan power indicator meaning .....	39
Table 5	Acquisition indicator meaning .....	41
Table 6	Choosing a menu from the menu list .....	41
Table 7	Choosing a submenu from a menu .....	42
Table 8	Choosing a parameter from a submenu .....	42
Table 9	Choosing a value from a parameter list .....	43
Table 10	Going back one level or canceling a selection .....	43
Table 11	Entering a value in an edit field .....	43
Table 12	Opening a shortcut menu .....	44
Table 13	The status indicators and their meanings .....	64
Table 14	Battery charge indicator variations .....	67
Table 15	Basic data views .....	70
Table 16	Ruler/scale colors .....	79
Table 17	Parameter button types .....	84
Table 18	Reflector, probe, and calibration block types .....	124
Table 19	RGB color codes .....	203
Table 20	Color of the Acq Rate indicator .....	261
Table 21	DAC curve properties relative to standards .....	302
Table 22	Example of the influence of the Measure parameter .....	311

