OmniScan MXU Software

User's Manual

Software Version 5.17

10-001244-01EN — Rev. 13
April 2024

This instruction manual contains essential information on how to use this 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, INC.
3415, rue Pierre-Ardouin, Quebec (Quebec) G1P 0B3 Canada

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 5.17
Part number: 10-001244-01EN
Rev. 13
April 2024

Printed in Canada

Table of Contents

List of Abbreviations

Acq.	acquisition
AIM	Acoustic Influence Map
AOD	axial outside diameter
AWS	American Welding Society
BP	band pass
COD	circumferential outside diameter
CSC	curved-surface correction
DAC	distance-amplitude correction
DC	direct current
DGS	distance gain size
DHCP	Dynamic Host Configuration Protocol
DNS	Domain Name System
ERS	equivalent reflector size
FBH	flat-bottom hole
FMC	full matrix capture
FSH	full-screen height
FW	full wave
HAZ	heat affected zone
HAZ	heat-affected zone
HP	high pass
HW-	half wave negative
HW+	half wave positive
IP	Internet protocol
L Velocity	longitudinal velocity

LED	light-emitting diode
ML	material loss
ND	no detection (of signal)
NS	no synchronization
P/C	pitch-catch
P/E	pulse-echo
PA	phased array
PCI	phase coherence Imaging
PRF	pulse repetition frequency
pts/λL	points per wavelenght for longitudinal wave
pts/λT	points per wavelength for transversal wave
PW	pulse-width
PWI	plane wave imaging
RCS	Remote Collaboration Service
RF	radio frequency
RGD	red, green, blue
SDH	side-drilled hole
T Velocity	transversal velocity
TCG	time-corrected gain
TFM	total focusing method
USB	Universal Serial Bus
UT	ultrasonic testing
VPA	virtual probe aperture

Important Information — Please Read Before Use

Intended Use

The OmniScan MXU software is for the OmniScan X3 flaw detector, which is used in nondestructive inspections on industrial and commercial materials.

WARNING

Do not use the OmniScan X3 flaw detector for any purpose other than its intended use. It must never be used to inspect or examine human or animal body parts.

Instruction Manual

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

Keep this instruction manual in a safe, accessible location.

IMPORTANT

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

Instrument Compatibility

CAUTION

Always use equipment and accessories that meet Evident specifications. Using incompatible equipment could cause equipment malfunction and/or damage, or human injury.

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 signal word might appear in the documentation of the instrument:

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 note signal words could appear in the documentation of the instrument:

IMPORTANT

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

NOTE

The NOTE signal word calls attention to an operating procedure, practice, or the like, that 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 that provides hints on how to effectively use the capabilities of the product.

Safety

Before turning on the instrument, verify that the correct safety precautions have been taken (see the following warnings). In addition, note the external markings on the instrument, which are described under “Safety Symbols.”

Warnings

WARNING

General Warnings

- Carefully read the instructions contained in this instruction manual and in the *OmniScan X3 User's Manual* prior to turning on the instrument.
- Keep this instruction manual 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 *Evident Terms and Conditions* available at https://evidentscientific.com/evident-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 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 https://www.evidentscientific.com/service-and-support/service-centers/.

Introduction

The OmniScan MXU software runs on the innovative, portable OmniScan X3 flaw detector. Its ultrasonic inspection functions make it suitable for numerous nondestructive testing applications. The software combines conventional ultrasonic testing (UT), phased array (PA), and total focusing method (TFM) operation modes.

In addition to this document, the following documents are relevant to the OmniScan X3 flaw detector operation:

OmniScan X3 — User's Manual

Provides a detailed description of the OmniScan X3 flaw detector. Refer to this document for operating instructions, maintenance, connections, specifications, and typical accessories.

OmniScan X3 — Getting Started Guide

A short leaflet containing essential information on how to quickly start operating the OmniScan X3 flaw detector.

1. Instrument Overview

The OmniScan X3 flaw detector features front panel controls for easy and efficient operation of the OmniScan MXU software. Figure 1-1 on page 20 shows the OmniScan X3 front panel and the available controls and indicators.

NOTE

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

Figure 1-1 Front panel controls of the OmniScan X3 flaw detector

Table 1 Front control panel controls description

Item number	Description
1	Display touch screen
2	Alarm indicator lights
3	Help key
4	Main controls: Accept key, Cancel key, and scroll knob
5	Zoom key
6	Play key
7	Pause key
8	Save key

Table 1 Front control panel controls description *(continued)*

Item number	Description
9	Power key
10	Power indicator key
11	Acquisition indicator light

1.1 Turning On and Off the OmniScan X3

This section explains how to turn on and off the OmniScan X3 flaw detector. The OmniScan MXU software shuts down automatically when you turn off the OmniScan X3 flaw detector.

To turn on the OmniScan X3

1. Press and hold the Power key (⏻) for one second.

   The system starts up, performs a memory check, and the launch screen appears (see Figure 1-2 on page 22).

NOTE

If the system encounters a problem during the start-up phase, the power indicator light indicates the nature of the problem using a color code (for details, refer to the *OmniScan X3 User's Manual*).

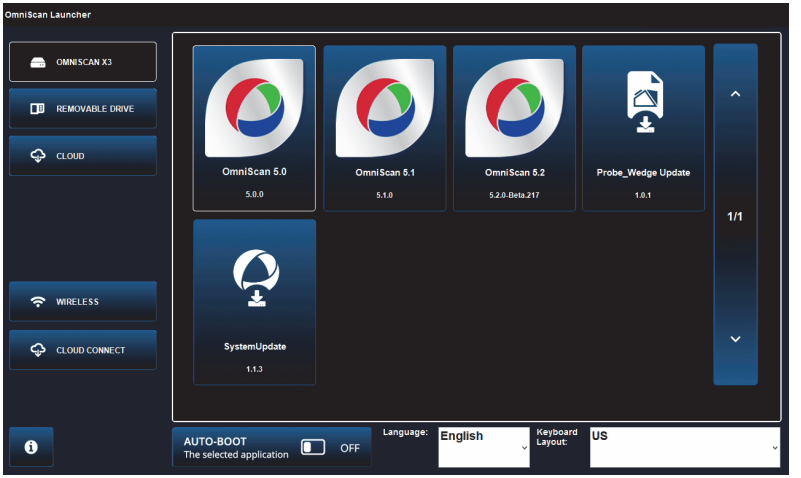


Figure 1-2 OmniScan Launcher

2. Tap to start the desired application and/or to configure the following:
   - **OmniScan Launcher** (applications)—if more than one application is available. The compatible file type has the .wrp file extension.
   - **OMNISCAN X3** (hard drive)—a series of buttons appears on the screen. To delete an application, tap and hold it until a message confirms the deletion. Applications must be on the hard drive to be executed.
   - **REMOVABLE DRIVE**—displayed only if a USB key or SD card is connected. Tap on an application to transfer it to the hard drive.

IMPORTANT

Prior to use, format the removable drives to one of the supported file systems, NTFS or exFAT.

   - **CLOUD**—displayed only if the **CLOUD CONNECT** is configured. This option gives you access to the official version of the system (MXU, System Update and Probe_Wedge Update). Tap on an application to transfer it to the hard drive.
   - **WIRELESS**—To activate the **WIRELESS** function, you need to plug the wireless LAN dongle into the instrument and, in the Wireless Properties,

check the Wireless Enabled option and select and configure your wireless Internet network.

- **CLOUD CONNECT**—To activate **CLOUD CONNECT**, you must enable the **WIRELESS** function. Tap **CLOUD CONNECT**, check **Enable** in the **Cloud Settings**, and make sure that the **Ready** and **Enable** statuses are **Yes**.
- —The information button shows the installed versions of the **Platform Compatibility**, **Low Level**, and **System**.
- **AUTO-BOOT**—Toggle this to ON to set the OmniScan X3 flaw detector to automatically boot up using the selected application (OmniScan X.X) on subsequent start-ups.
- **Language**—This option enables you to change the language of the software. You must change the language before starting the application.
- **Keyboard Layout** —This option enables you to change the keyboard language of the software. You must change the keyboard language before starting the application.

If you always choose the same application, you can skip the application selection step for future restarts by selecting **Always boot the selected application** below the software buttons.

To regain the ability to choose the application at start-up, select **Preferences > System**, and then **Manual boot**.

To turn off the OmniScan X3

1. Press and hold the Power key ( ) for 3 seconds.
2. Tap the **Shut Down** button on the confirmation window to turn off the OmniScan X3 flaw detector.

IMPORTANT

If the OmniScan X3 does not react after a short press of the Power key ( ) (or after selecting **Shut Down**), press and hold the Power key ( ) for at least five seconds. This initiates a power-down sequence. However, your setup will NOT be saved with this method.

CAUTION

Never attempt to turn off the OmniScan X3 flaw detector by removing all power sources, because this could cause a faulty start-up the next time you turn it on.

1.2 Installing Software

The OmniScan MXU software can easily be updated. You can download the latest MXU software version at: https://www.olympus-ims.com/en/service-and-support/downloads/ or by using the **CLOUD** option. From the Internet, extract the contents of the *.zip file on a USB key or SD card, and then insert it into the OmniScan X3 instrument. The file needs to be on the root directory of the removable drive to be detected. From the **CLOUD**, select the application to copy to the instrument. On the launcher screen, tap the inserted media folder and select the application to copy to the instrument. After the copying has been completed, the newly installed software appears in the OmniScan X3 main folder.

1.3 Main Controls

The three main controls shown in Table 2 on page 24 enable full operation of the OmniScan MXU software.

Table 2 Main controls for OmniScan X3 flaw detector

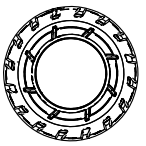
Image	Name	Description
	Scroll knob	Rotate the scroll knob clockwise or counterclockwise to select a desired software button, or change a parameter value.

Table 2 Main controls for OmniScan X3 flaw detector *(continued)*

Image	Name	Description
	Accept key	Press the Accept key to activate the current selection, and move to the next level in the menu hierarchy. In an alphanumeric parameter value field, pressing the Accept key twice (or tapping the parameter twice) opens the software keyboard.
	Cancel key	Press the Cancel key to cancel the current selection, and return to the previous level in the menu hierarchy.

1.4 Function Keys

The function keys are located on the keypad on the right side of the OmniScan X3 front panel (see Figure 1-1 on page 20). Table 3 on page 25 summarizes how to use the function keys to activate different software functions.

Table 3 Key functions for OmniScan X3 flaw detector

Image	Name	Function
	Zoom	Used to enter and exit Zoom mode. For details, see “Using the Zoom, Pan, Gates, and Print Screen” on page 41.
	Play	Used to restart the inspection data acquisition, and/or the encoders, depending on the configuration on the **Scan > Inspection** menu.
	Pause	Used to toggle between the inspection and analysis modes.
	Save	Used to save the report, data, or image, depending on the configuration on the **File name** menu.

1.5 Indicators

There are three types of LED indicators on the instrument front panel that turn on, off, and flash different colors (see Figure 1-1 on page 20):

- Power LED—green if the instrument is "on", but flashes red during a critical power situation. (Refer to the *OmniScan X3 User's Manual* for full status descriptions, for example, orange states during charging.)
- Acquisition LED—turns orange during analysis mode and off during inspection.
- Alarm LEDs (3)—turn red when an associated (gate) alarm is triggered.

1.6 File Formats

From MXU 5.11 and forward the file format used in the OmniScan X3 is the *.nde* file format rather than the legacy *.odat* format.

Note that *.nde* is an open file format that allows data to be accessed with no proprietary software.

The *.odat* file format will still be supported in MXU versions 5.11 and up, but no new *.odat* files will be created.

Set up files created before MXU 5.11 are supported. However, new data files will use the *.nde* format. If an *.odat* file is edited and saved using MXU 5.11 and up, it will remain in the *.odat* file format.

NOTE

Added analysis features are not available for *.odat* files in MXU versions 5.11 and up.

2. OmniScan Interface

The main components of the OmniScan MXU software user interface are shown in Figure 2-1 on page 27.

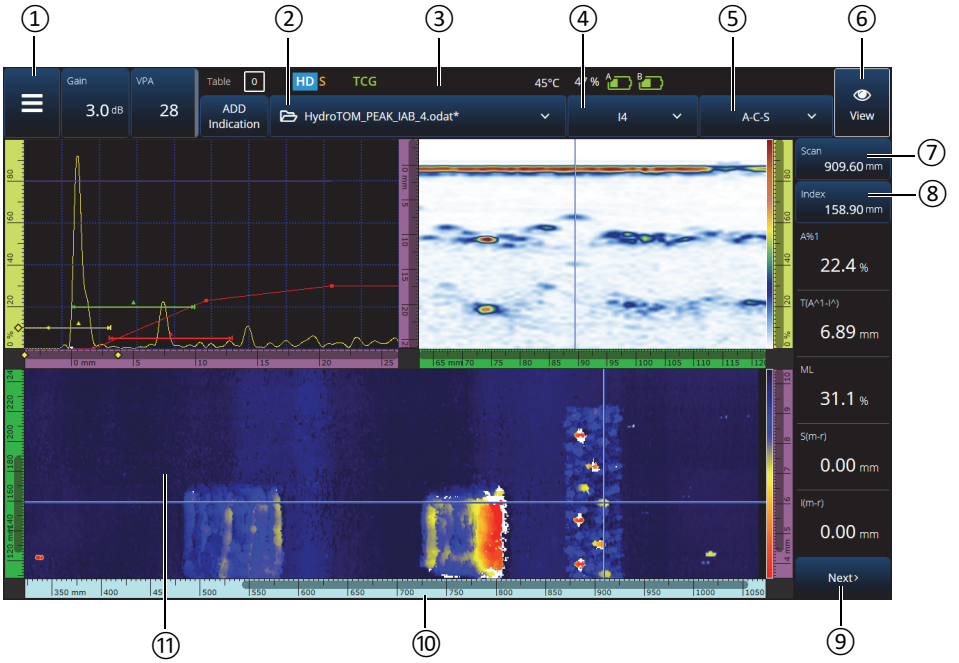


Figure 2-1 OmniScan MXU interface components

Table 4 OmniScan MXU interface components

Item number	Description
1	Main menu
2	File menu
3	Status indicator
4	Focal law groups menu
5	Layout menu
6	View menu
7	Scan position indicator and control
8	Index position indicator and control
9	Readings menu (scroll for more)
10	Ruler (scale)
11	Data screen

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 operations in version 5.1 (see “Preferences” on page 93).

2.1 OmniScan MXU Software Navigation

Figure 2-2 on page 29 shows the three menu levels of the OmniScan MXU software, and describes the syntax used throughout this manual to systematically select the menu and submenu, and to optionally enter or select a parameter value. For example,

☰ > 🔔 **Gates & Alarms > Gate Main > Start** signifies that you first select the ☰ Main menu, then 🔔 **Gates & Alarms** menu, followed by **Gate Main** submenu, and finally the **Start** parameter.

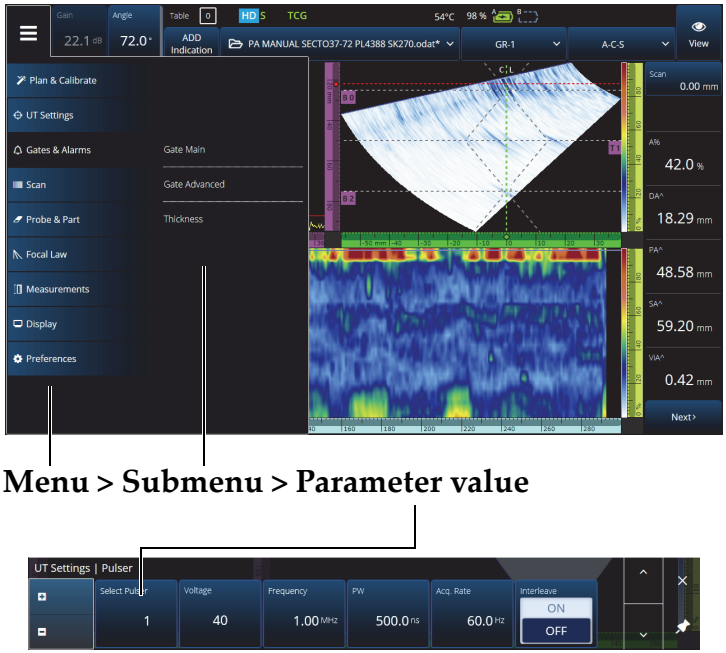


Figure 2-2 Menu hierarchy and identification syntax

The menu temporarily appears horizontally over the data screen area, with submenu selections to the right. When selected, the parameter submenu appears over the data screen. It is possible to scroll to another submenu using the arrow buttons (▲▼). The submenu can be hidden by tapping the close button (✕), or it can be pinned (📌) to the side of the screen (see Figure 2-3 on page 30).

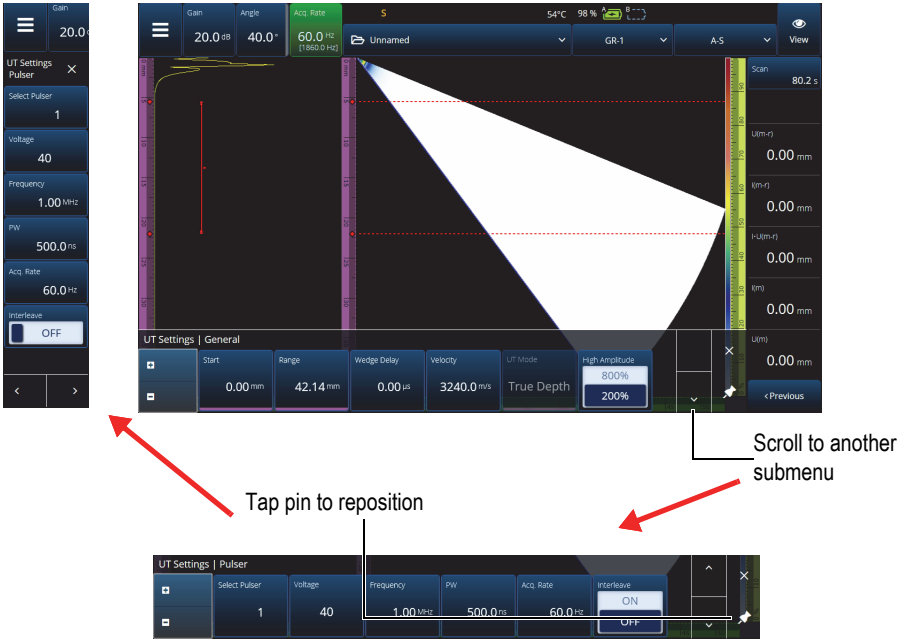


Figure 2-3 Scroll and reposition a parameter submenu

2.2 Gain

The **Gain** value applied to all focal laws of the current group appears in the upper-left corner of the screen. Figure 2-4 on page 30 shows the information displayed in the **Gain** value field.

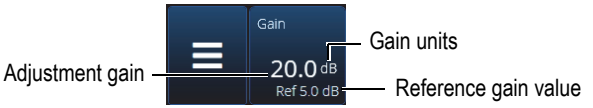


Figure 2-4 Gain value field

The **Gain** value field presents two values after the **UT Settings > Advanced > Reference dB** parameter is turned **On** (in TFM, **Reference dB** is turned on in **TFM Settings > General**). 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.

2.3 Status Indicators

The current status of the OmniScan X3 flaw detector is indicated at the top of the screen (see Figure 2-5 on page 31). Table 5 on page 31 provides a list of the status indicators and their meanings.

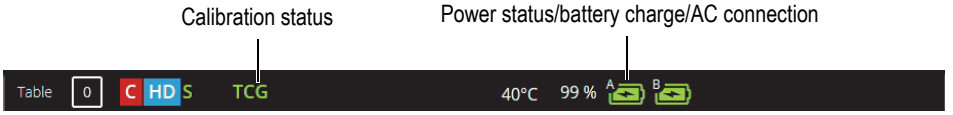


Figure 2-5 Example of the status indicators

Table 5 Status indicators and their meanings

Indicator	Meaning
C	The number of data points on the inspected area exceeds the number of available pixels (see “Compression (TOFD Only)” on page 123).
HD	High definition: Used to see the data scale and ruler on the instrument correctly with the resolution of the screen (1280 × 768). Seeing the HD icon tells you that there is no compression in the scan axis (in the case of one-line scans) or both the scan axis and index axis (in the case of raster scans).
TCG (green)	The time-corrected gain (TCG) is applied (“TCG/DAC Calibration” on page 170).
DAC (green)	The DAC curve is applied to the current group.
DGS (green)	The DGS curve is applied to the current group.

Table 5 Status indicators and their meanings *(continued)*

Indicator	Meaning
	Blinking: The GPS is acquiring the device's location. Steady: The geolocation is active and the position is acquired.
[52]°C	The OmniScan X3 flaw detector's internal temperature in degrees Celsius.
[④]	The wireless LAN is active.
	Connected to the cloud (with notifications).
S (green)	The sensitivity is calibrated.
W (green)	The wedge delay is calibrated.

2.4 Battery Status Indicators

The battery status indicators at the top of the screen indicate the amount of power remaining in the batteries:

- The percentage of remaining power is displayed next to the indicators. The OmniScan X3 flaw detector must be turned on for approximately 15 minutes before it is able to accurately display this information.
- The bar length in the battery status indicator represents the approximate amount of power remaining in each battery (for example, 70%).

IMPORTANT

The maximum ambient temperature for OmniScan X3 battery discharging is 45 °C (the maximum OmniScan X3 operating temperature).

NOTE

If you attempt to turn on the OmniScan X3 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 X3 flaw detector.

Figure 2-6 on page 33 provides details about the variations of the battery charge indicator.

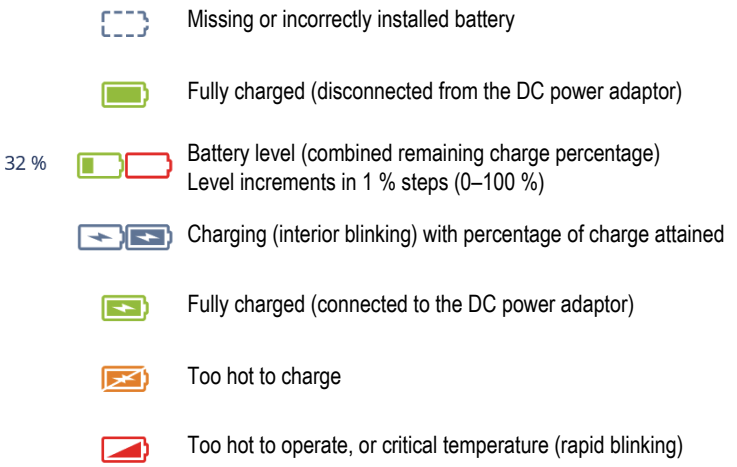


Figure 2-6 Battery indicator variations

The battery status indicator in MXU and the battery status indicator on the batteries may differ. This is because the OmniScan MXU software is more conservative with the remaining charge. Figure 2-7 on page 34 shows the equivalence between the software and hardware battery indicators.

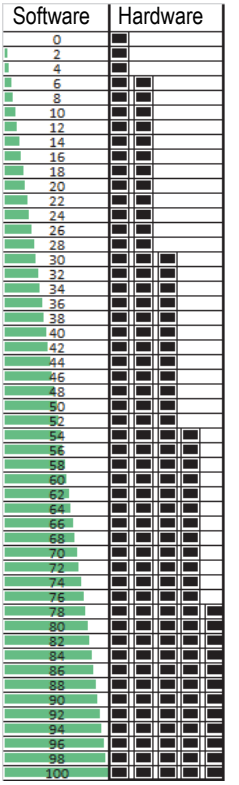


Figure 2-7 Battery charge display in MXU vs. hardware indicator

2.5 Data Screen

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

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 120). 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 of ultrasonic probe beams that use the same parameters (also referred to as a "beam set"), 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.

Table 6 on page 35 lists the basic ultrasonic scan views, which are illustrated in Figure 2-8 on page 36.

Table 6 Basic ultrasonic scan 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
S-scan	End	Ultrasound versus index

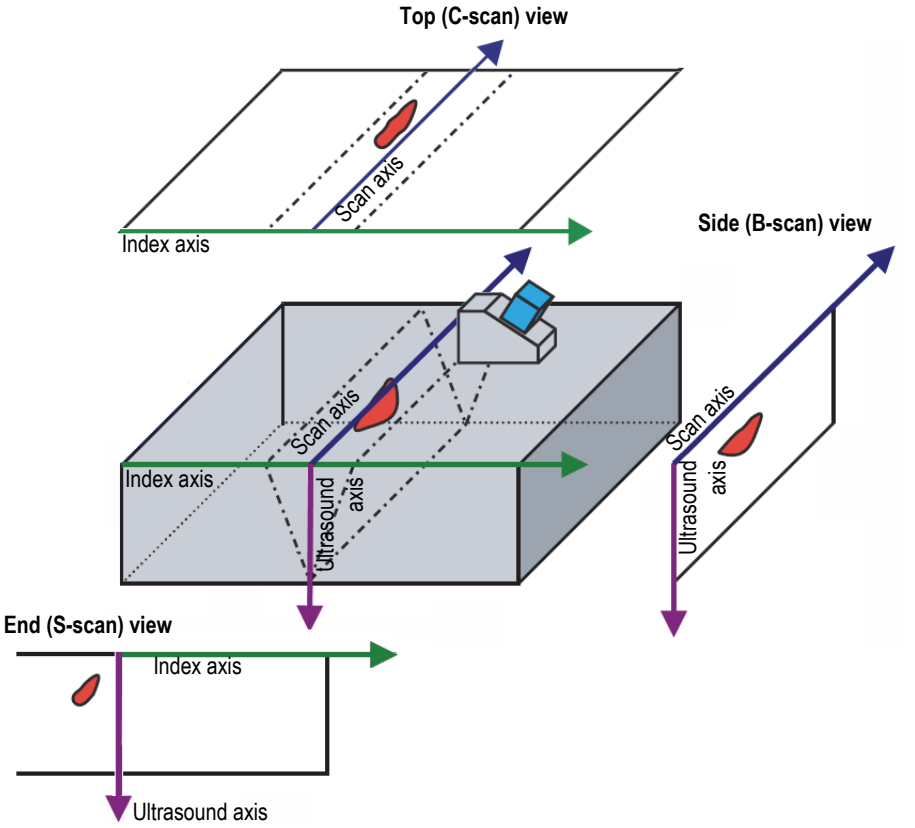


Figure 2-8 Example of ultrasonic scan views

The scans and views, which are available on the Layout menu, can be further described as follows:

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 (ultrasound path), or a waveform. A peak in the signal corresponds to the echo of a reflector or a discontinuity in the part. In TFM, the A-scan is constructed from the TFM grid and is not generated by a single beam like in standard PA.

B-scan (side view)

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 (top view)

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).

S-scan (PA group only)

2-D representation of ultrasonic 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 (TFM group only)

2-D representation of the ultrasonic data acquired with TFM. This view displays the amplitude color-coded on an ultrasound-index plan. The size of each axis is defined by the **Zone** parameters. The view will be displayed according to the geometry of the part, so curved parts will be displayed on curved axes.

Top view (TFM group only)

2-D representation of ultrasonic data acquired with TFM. This view displays the maximum amplitude of the full ultrasound range on a scan-index plan.

Side view (TFM group only)

2-D representation of ultrasonic data acquired with TFM. This view displays the maximum amplitude in a projection on the scan-ultrasound plan.

Your layout selection can combine the most useful views (see Figure 2-9 on page 38).

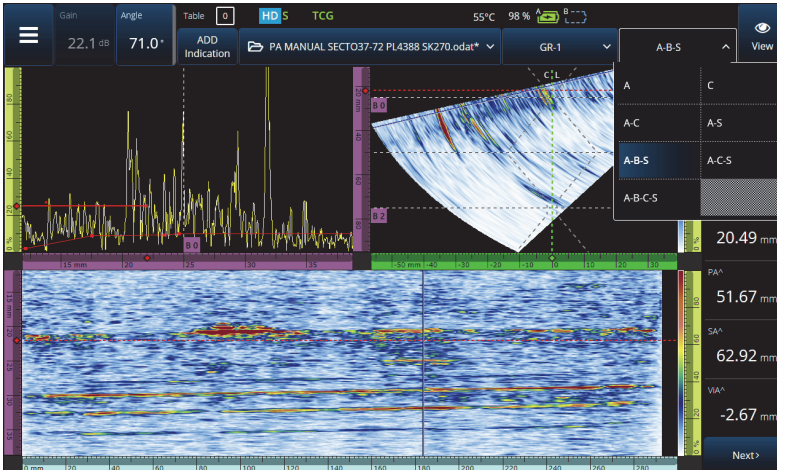


Figure 2-9 Layout menu

To select a layout

1. Tap the Layout menu.
2. Select the layout(s) to be displayed.
3. To toggle between Single group layout or Multiple groups layout, tap the View button, and select either **Single** or **Multiple**.

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

- A (A-scan)
- B (B-scan)
- C (C-scan)
- S (S-scan)
- End view (TFM group)
- Top view (TFM group)
- Side view (TFM group)

When multiple groups are displayed, combinations of the above-mentioned layouts are possible.

NOTE

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

TIP

If you have a multiple group setup, you may want to rename the groups in the scan plan.

2.6 Using the Touch Screen

You can interact with the OmniScan MXU software using the touch screen, but, if you prefer, it is also possible to connect a mouse and/or a keyboard through the USB ports.

To use the touch screen

- Simply tap once on the touch screen to do a left click.
- Tap and hold your finger on the touch screen to do a right click. Many shortcuts (see Table 57 on page 125) are available using a tap and hold (or right-click).

Instead of using the virtual keyboard or numerical keypad, you can simply input values using a physical keyboard connected to the instrument.

IMPORTANT

In some cases, gate or cursor selection zones overlap. If you attempt to select a cursor or gate at the point where they overlap, they will be selected in this order of priority: Reference cursor, Measurement cursor, Data cursor, gate A, gate B, and gate I.

2.6.1 Entering or Editing Values

You can use the virtual keyboard, arrows, or scroll knob to enter or edit numerical parameter values.

To enter or edit values

1. Tap the parameter (see Figure 2-10 on page 40).
2. Rotate the scroll knob to change the value, and then press the Accept key ( ).

   OR

   Tap to display the numerical keypad, then enter the value, and tap the Accept button ( ).

   Alternatively to accept, press another key or button, or tap any layout view.

To revert to the previous value, press the Cancel key ( ) on the instrument or the Cancel button ( ) on the virtual keyboard.

TIP

You can display the numerical keypad by tapping twice on the numerical parameter you wish to change. Also, you can change the scroll knob increment using and .

Figure 2-10 Parameter adjustment using up/down arrows or keypad

2.6.2 Using the Zoom, Pan, Gates, and Print Screen

To use the zoom

1. Press the Zoom key ( ) to turn on (or off) zoom mode (see Figure 2-11 on page 41).
2. Adjust the zoom:
   - Tap the screen twice at the corners of the area you want to zoom.

     OR

     Press twice on the Zoom key to reset the zoom.

     OR

     Tap the view location you wish to zoom, and then use the scroll knob to create a concentric zoom that is centered on the tapped position.

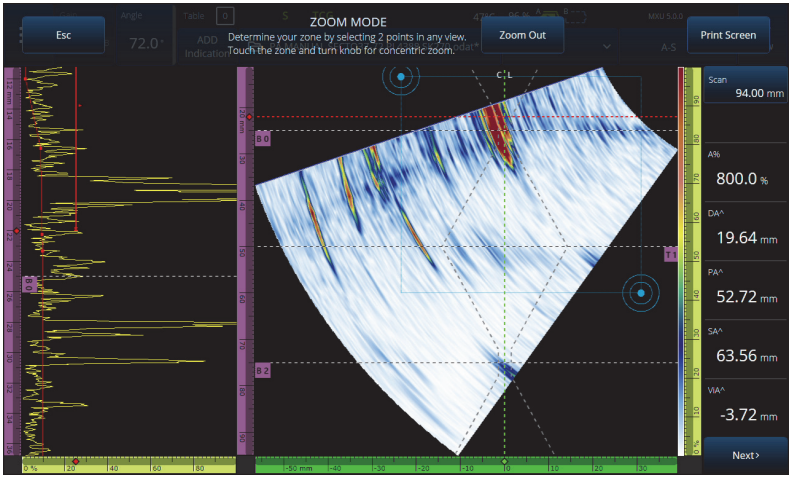


Figure 2-11 Zoom example

To pan within a zoomed view

- Click on the ruler corresponding to the axis you want to pan. Either use the knob to pan the view, or input the center position of the window in the field **Center**.

To adjust the gates

1. To adjust the gate **Start**, tap the left end of the gate.
2. To adjust the gate **Threshold**, tap the middle of the gate.
3. To adjust the gate **Width**, tap the right end of the gate.

NOTE

When a gate is short, it might be impossible to tap a specific zone. In this case, gate **Start** and **Width** controls are almost at the same position on the screen. Use the Gate menu to adjust a gate if tapping a specific zone is too difficult (see Figure 2-12 on page 42).

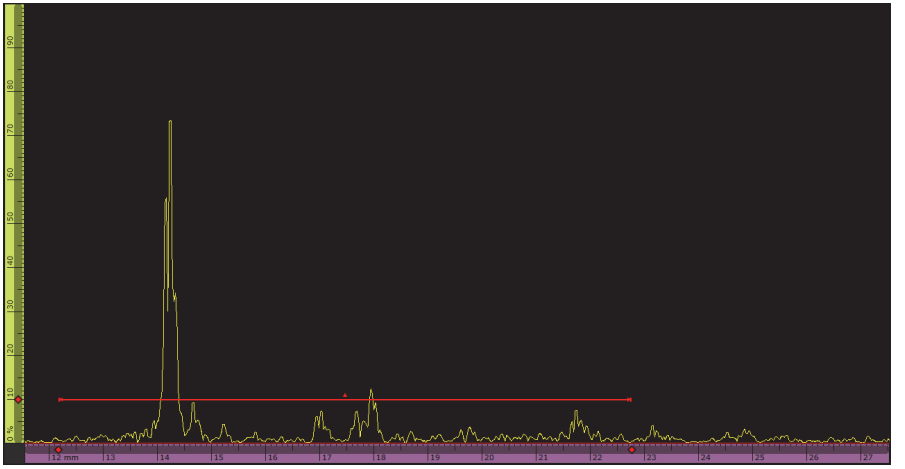


Figure 2-12 Visual reference on the gate

To use Print Screen

- Press the Zoom key ( ) to turn on zoom mode (see Figure 2-11 on page 41), and then tap **Print Screen** on the display.

NOTE

After tapping **Print Screen**, you have two to three seconds to make any screen adjustments or to open any temporary menus before the screen image is taken.

2.6.3 Pop-Up Buttons and Menus

Some buttons or menus enable pop-ups; for example, for parameter values, file names or probe/wedge library items (see Figure 2-13 on page 43).

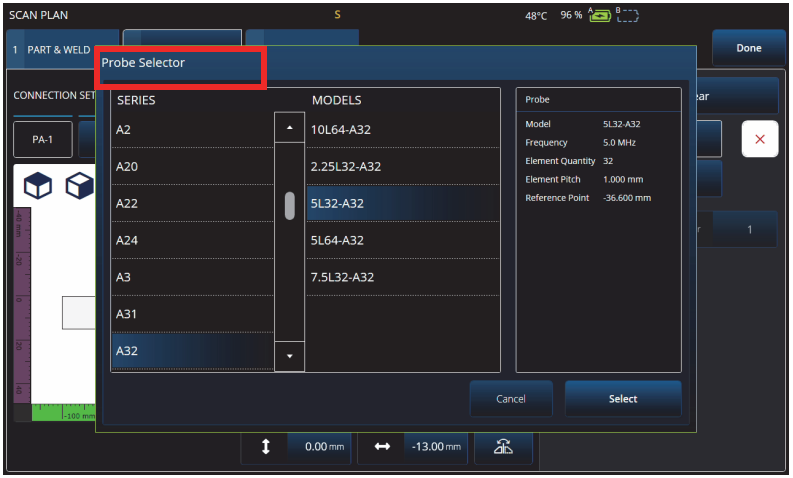


Figure 2-13 Pop-up menu example

2.7 Main Menu Organization

The ☰ Main menu provides a range of submenus for inspection configuration (see Figure 2-14 on page 44 and Table 7 on page 44).

NOTE

Depending on your chosen configuration, the menu can change from **UT Settings** to **TFM Settings**.

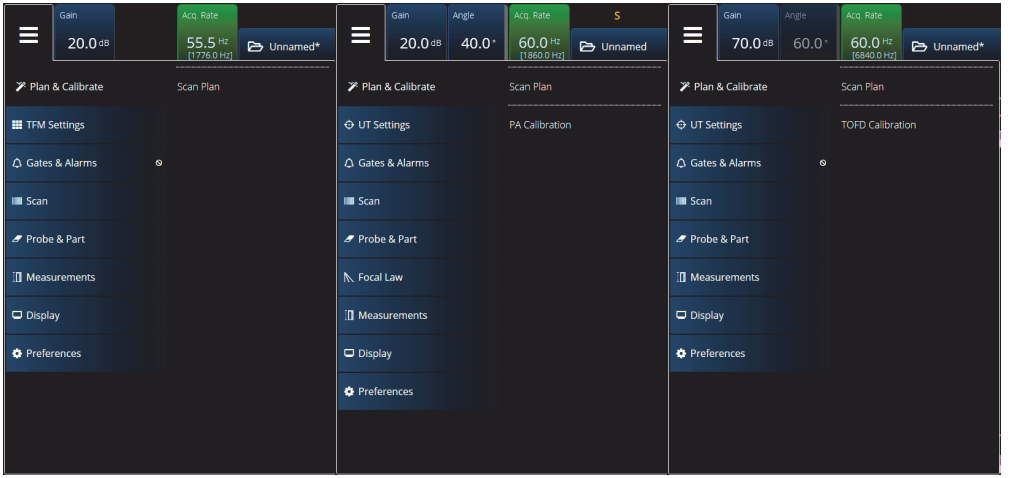


Figure 2-14 Main menu

Table 7 Main menu options

Menu	Description
Plan & Calibrate	Use this menu to create a complete application setup. The **Scan Plan** and **Calibration** wizards guide you in creating your setup.
UT Settings	This menu contains the parameters regularly modified during inspection, such as the gain and pulser/receiver parameters. (Available only for PA/UT inspections.)
TFM Settings	This menu provides settings for the total focusing method and full matrix capture. (Available only for TFM inspections.)

Table 7 Main menu options *(continued)*

Menu	Description
Gates & Alarms	This menu contains parameters for configuring gates, alarms, and output signals.
Scan	Use this menu to set the parameters related to the scanning, such as encoders and the area to scan.
Probe & Part	Use this menu to define probes and wedges and to adjust the parameters related to the probe position or part thickness previously defined in the scan plan.
Focal Laws	Use this menu to adjust the parameters related to the focal laws originally defined with the **Focal Law** wizard.
Measurements	This menu contains the parameters related to various measurement tools.
Display	This menu contains the parameters related to the data views and the information visible on the screen.
Preferences	Use this menu to set instrument configuration parameters when you start using the instrument. For example, the measurement unit (millimeters or inches) and the date and time.

2.7.1 UT Settings

The **UT Settings** menu enables you to access the **General**, **Pulser**, **Receiver**, **Beam,** and **Advanced** parameters.

2.7.1.1 General

Using the **General** parameter, you can see and modify the **Start**, **Range**, **Wedge Delay**, **Velocity**, **UT Mode**, and **High Amplitude** options. To access these options, go to **UT Settings > General** (see Figure 2-15 on page 46 and Table 8 on page 46).

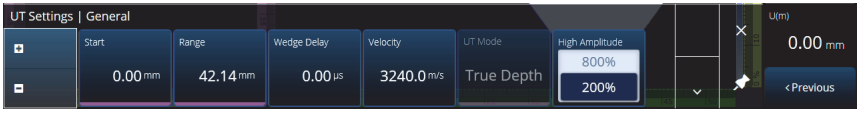


Figure 2-15 UT Settings — General

Table 8 UT Settings — General

Option	Description
Start	Used to set the starting location of the ultrasound axis (expressed in units of distance or time, according to the **UT Mode** setting.).
Range	Used to set the length of the ultrasound axis (expressed in units of distance or time, according to the **UT Mode** setting).
Wedge Delay	Used to set the delay applied to all focal laws in the group (expressed in µs (microseconds)).
Velocity	Used to set the velocity of ultrasound in the material (expressed in m/s (meter per second) or in µs (microseconds)).
UT Mode	UT: Used to change the ultrasound axis representation: **Time**, **Sound Path**, and **True Depth**. TOFD is set to **Time** and PA is set to **True Depth** in read-only.
High Amplitude	Used to switch between 200 % and 800 % mode. Data is encoded on 16 bits, so 200 % yields more precision, while 800 % provides greater tolerance to high amplitude variations.

2.7.1.2 Pulser

Using the **Pulser** parameter, you can see and modify the **Select Pulser**, **Voltage**, **Frequency**, **Velocity**, **PW, Acq. Rate**, and **Interleave** options. To access these options, go to **UT Settings > Pulser** (see Figure 2-16 on page 47 and Table 9 on page 47).

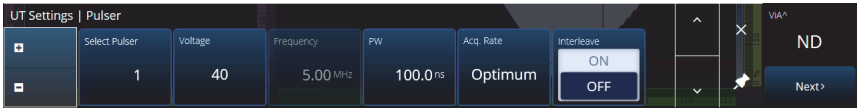


Figure 2-16 UT Settings — Pulser

Table 9 UT Settings — Pulser

Option	Description
Select Pulser	PA Connector: Displays the number of your starting pulser. UT Connector: Displays P1 or P2 according to the connector defined in the scan plan.
Voltage	PA Connector: Used to set the voltage of the pulser to either 40 (default value), 80, or 115. The OmniScan X3 has unipolar voltage (negative square pulse), while the OmniScan X3 64 has bipolar voltage (negative and positive square pulse). On the OmniScan X3 64, the voltage values are given in Peak-to-Peak voltage (Vpp) and range from 10 Vpp to 160 Vpp. Equivalent voltage are typically stronger in bipolar than in unipolar UT Connector: Used to set the voltage of the pulser to either 85 (default value), 155, or 295.
Frequency	Displays the probe frequency value. The value can be edited if the probe selected in the scan plan is **Unknown**.
PW	Used to select the pulse-width (PW) value. Select **Auto** to automatically adjust the pulse width according to the probe frequency. Select **Edit** to modify the value manually.

Table 9 UT Settings — Pulser *(continued)*

Option	Description
Acq. Rate	Used to set the value of the acquisition rate (Acq. Rate). The **Acq. Rate** value is defined for all groups and defines the repetition frequency of all channels. The product of **Acq. Rate** × **Scan Resolution** is equal to the scan speed if the inspection is set to **Time**, and is equal to the **Max. Scan Speed** for an inspection set to **Encoder** mode. If the scanning movement is faster than the **Max. Scan Speed**, data could be missed, which will be indicated by black lines. 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 attain. You can also choose one of the following presets: **Auto Max.:** Uses the maximum available **Acq. Rate** value. An acquisition rate that is too high may generate ghost echoes in some specimen. **Default:** The default value is 120 Hz. If the maximum available **Acq. Rate** is lower than 120, the default value is then set at this lower value. **Edit:** You can enter a value manually. **NOTE** 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 TTotal and PRF is the inverse of TBeam, which are expressed in the following calculation: Acq. Rate = 1/TTotal. With a multiple group configuration, the acquisition rate takes into account the emission of pulses for all groups.

Table 9 UT Settings — Pulser *(continued)*

Option	Description
Interleave	Set this parameter to **ON** to enable (**OFF** is the default) interleaving of the focal law firing sequence, which delays the appearance of ghost echoes.

2.7.1.3 Receiver

Using the **Receiver** parameter, you can see and modify the **Filter**, **Rectifier**, **Video Filter**, **Averaging**, and **Reject** options. To access these options, go to **UT Settings > Receiver** (see Figure 2-17 on page 49 and Table 10 on page 49).

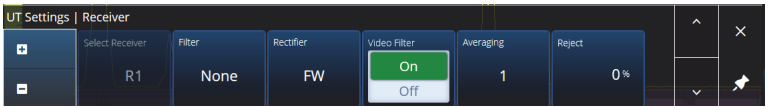


Figure 2-17 UT Settings — Receiver

Table 10 UT Settings — Receiver

Option	Description
Receiver	The value mirrors the value of the Pulser (read-only) if the group is a PA group or a UT pulse-echo group. The value can be edited only if the group is a UT on PA in pitch-catch configuration.

Table 10 UT Settings — Receiver *(continued)*

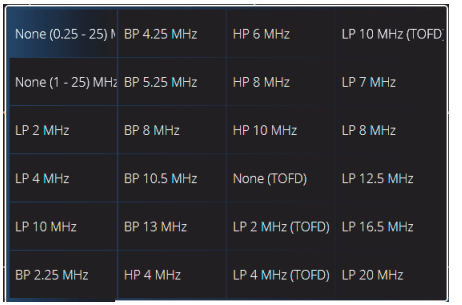
Option	Description
Filter	Used to select the appropriate filter value, such as TOFD or **LP (**low pass), **HP** (high pass), and **BP** (band pass).  

Table 10 UT Settings — Receiver *(continued)*

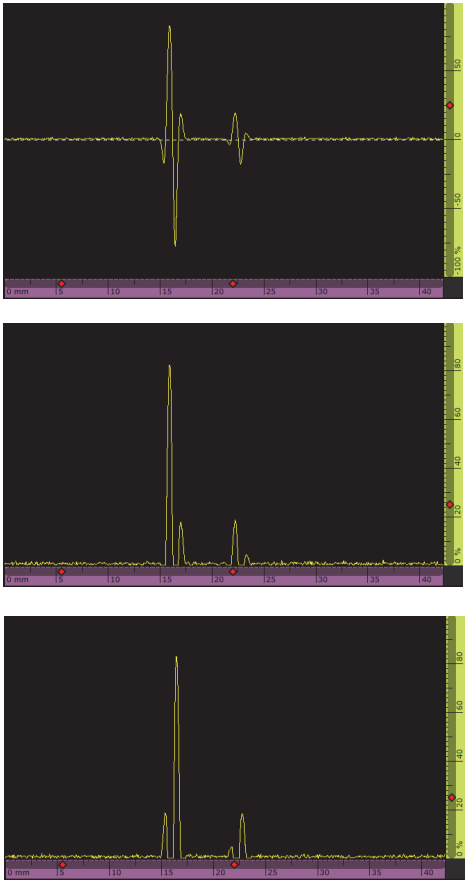
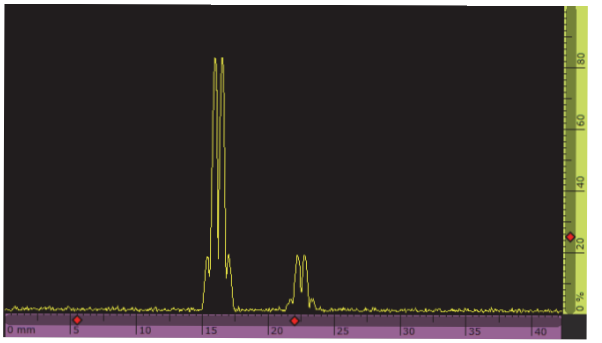
Option	Description
Rectifier	Used to set rectification of the A-scan signal. The four options are RF (radio frequency) non-rectified, HW+ (half wave positive), HW- (half wave negative), and FW (full wave), which are displayed below respectively. 

Table 10 UT Settings — Receiver *(continued)*

Option	Description
Rectifier *(continued)*	
Video Filter	PA/UT: 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 unavailable in RF mode.
Averaging	Used to select an averaging value (1, 2, 4, 8, or 16) 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. For TOFD, averaging values of 32 and 64 are also possible.
Reject	Signal amplitude inferior to the specified value is forced to 0 %. The default value is set to 0 %.

2.7.1.4 Beam

Using the **Beam** parameter, you can see and modify the **Scan Offset**, **Index Offset**, **Skew, Beam Delay**, **Gain Offset**, and **Refracted Angle** options. To access these options, go to **UT Settings > Beam** (see Figure 2-18 on page 53 and Table 11 on page 53).

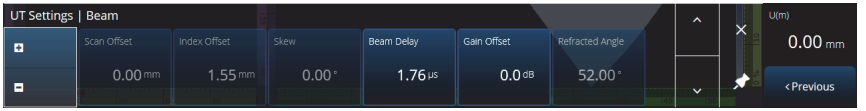


Figure 2-18 UT Settings — Beam

Table 11 UT Settings — Beam

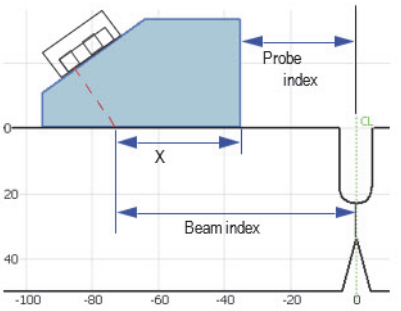
Option	Description
Scan Offset	PA/UT/TOFD: Displays the value calculated in the scan plan. The beam **Scan Offset** is the additional scan offset of the current beam with respect to the probe scan offset defined in **Probe & Part/Position**. The total scan offset for a specific beam is the Probe Scan Offset + Beam Scan Offset.
Index Offset	PA/UT/TOFD: The beam **Index Offset** is the difference between the 0-position marked on the part for inspection and the beam exit point 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. 
Skew	PA: Additional beam skew relative to the probe orientation (typically 90° or 270°). When the beam skew indicates 0°, it means the beam skew is aligned with the probe skew.

Table 11 UT Settings — Beam *(continued)*

Option	Description
Beam Delay	PA: 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 (expressed in µs (microseconds)).
Gain Offset	PA: Displays 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 (expressed in dB (decibels)).
Refracted Angle	PA/TOFD: Displays the angle of the ultrasonic beam in the material. UT: Sets the angle of the ultrasonic beam in the material. The nominal value appears between brackets.
Reference Point	UT/TOFD: Sets the distance between the front edge of the wedge and the beam exit point. The nominal value appears between brackets.

2.7.1.5 Advanced

Using the **Advanced** parameter, you can see and modify the **Ref. Amplitude**, **Auto 80 %**, **Reference dB**, **Point Quantity, Compression**, and **Effective Digitizing Frequency** options. To access these options, go to **UT Settings > Advanced** (see Figure 2-19 on page 54 and Table 12 on page 55).

Figure 2-19 UT Settings — Advanced

Table 12 UT Settings — Advanced

Option	Description
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 value for Auto XX % gain adjustment and also sets the height of the reference line if activated**.**
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. 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.
Point Quantity	**PA/UT:** Used to set the number of A-scan points to be stored. Increase this value to adjust the **Compression** factor to the desired value. Increasing the **Point Quantity** will first increase the range until the **Compression** can decrease, making sure the real UT range is never below the requested UT range (defined 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. **TOFD:** Displays the number of A-scan points to be stored. By default, this value is fixed and depends on the ultrasound range. 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.

Table 12 UT Settings — Advanced *(continued)*

Option	Description
Compression	**PA/UT**: Displays the value of the A-scan compression. Depending on the inspection range and number of points, a compression value greater than 1 may be required. For example, a value of 6 will keep the maximum value of every 6 consecutive acquisition points in time. No maxima are missed. **TOFD:** The compression is forced to 1 in TOFD and is read-only.
Effective Digitizing Frequency	The **Effective Digitizing Frequency** is set to 100 MHz, which means that a data point is acquired at every 0.01 µs of the analog waveform. This value cannot be changed by the user.
Net Digitizing Frequency	The **Net Digitizing Frequency** is the result of the **Effective Digitizing Frequency** divided by the **Compression**. The result is used for code compliance. The value in brackets [ ] is the time interval between each A-scan point.

2.7.2 TFM Settings

The **TFM Settings** menu enables you to access the **General**, **Pulser**, **Zone,** and **Advanced** parameters.

2.7.2.1 General

Using the **General** parameter, you can see and modify the **L Velocity**, **T Velocity**, **Reference dB**, and **Envelope** options. To access these options, go to **TFM Settings > General** (see Figure 2-20 on page 56 and Table 13 on page 57).

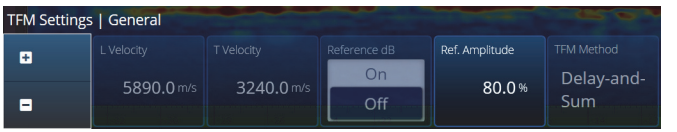


Figure 2-20 TFM Settings — General

Table 13 TFM Settings — General

Option	Description
L Velocity	Velocity of longitudinal waves in the material of the inspected part. The material type and longitudinal wave velocity are normally set during the group creation as part of the scan plan process.
T Velocity	Velocity of transversal waves in the material of the inspected part. The material type and transversal wave velocity are normally set during the group creation as part of the scan plan process.
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. The gain applied 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.
Ref. Amplitude	Set the reference amplitude in percentage.
TFM Method	You can only change the TFM method on OmniScan X3 64 flaw detectors. The choices are either **Delay-And-Sum** or **Phase Coherence Imaging (PCI)**. The TFM method can be applied to groups independently. For more information on PCI, see “Phase Coherence Imaging (PCI)” on page 210. For all OmniScan X3 models, the TFM method by default is **Delay-And-Sum.**

2.7.2.2 Pulser

Using the **Pulser** parameter, you can see and modify the **Voltage**, **Frequency**, **PW**, and **Acq. Rate Mode** options. To access these options, go to **TFM Settings > Pulser** (see Figure 2-21 on page 58 and Table 14 on page 58).

Figure 2-21 TFM Settings — Pulser

Table 14 TFM Settings — Pulser

Option	Description
Select Pulser	Indicates which element on the probe to be used as the first element of the pulser.
Voltage	Voltage of the pulser. On an OmniScan X3 64 flaw detector, you can select 10 Vpp, 20 Vpp, 40 Vpp, 80 Vpp, 120 Vpp, or 160 Vpp. On an OmniScan X3 flaw detector you can select 40 V (default value), 80 V, or 115 V.
Frequency	Probe-frequency value. To modify the frequency, select the **Probe & Wedge Manager** or change the probe in the scan plan.
PW	Pulse-width (PW) value. The pulse width is automatically adjusted according to the probe frequency.

Table 14 TFM Settings — Pulser *(continued)*

Option	Description
Acq. Rate	Used to set the value of the acquisition rate (**Acq. Rate**). The **Acq. Rate** value is defined for all groups and defines the repetition frequency of all channels. The product of **Acq. Rate** × **Scan Resolution** is equal to the scan speed if the inspection is set to **Time**, and is equal to the **Max. Scan Speed** for an inspection in **Encoder** mode. If the scanning movement is faster than the **Max. Scan Speed**, data could be missed and is indicated by black lines. 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. **Default** (default value) Sets the **Acq. Rate** to the minimum value between 120 Hz and the maximum acquisition rate available. **Edit** You can enter a value manually.

Table 14 TFM Settings — Pulser *(continued)*

Option	Description
Sparse	Adjusts the pulser sparsity of the FMC acquisition. The default Sparse value is set so a minimum of 16 elements are pulsed. For a 64-element probe, the default value is set to 1/4, For a probe with 16 elements or less, the default value is set to **Full Matrix**. The user can always change the value during setup. In a **Full Matrix** configuration (default), every element pulses and receives. Choosing a different **Sparse** value changes the number of pulsers that will be activated for pulsing, but all the elements are still used for receiving. The options are **Full Matrix**, **1/2**, **1/3**, **1/4**, **1/8**, and **1/16**. With a 32-element probe, for example, a value of **1/2** means that 16 elements will be pulsed and all 32 elements will receive. Some options may not be displayed because a minimum of 4 pulsers is required (for example, with a 16-element probe, **1/8** and **1/16** are absent). Changing the **Sparse** value will, in most cases, increase the maximum **Acq. Rate**, but may result in a lower signal-to-noise ratio (SNR).

2.7.2.3 Receiver

Using the **Receiver** parameter, you can define the filter to be applied on the TFM signal. To access this option, go to **TFM Settings > Receiver** (see Figure 2-22 on page 60 and Table 15 on page 61).

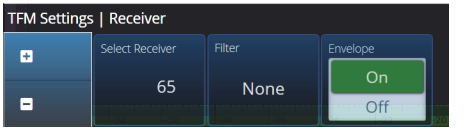
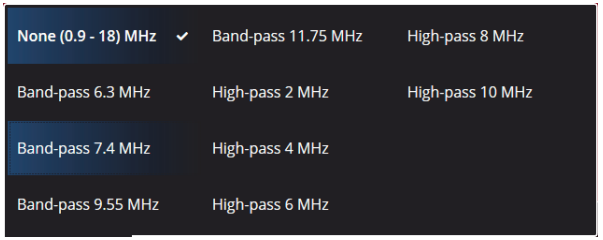


Figure 2-22 TFM Settings — Receiver

Table 15 TFM Settings — Receiver

Option	Description
Select Receiver	Indicates which element on the probe to be used as the first element of the receiver.
Filter	Select the appropriate filter value to be applied on the TFM signal. 
Envelope	Used to turn the **Envelope ON** (by default) or **OFF**. The envelope can be applied to groups independently. The TFM envelope is produced by combining and extracting the norm of two signals: the real component of the elementary A-scan acquired through FMC and the Hilbert-transform imaginary component. The processing removes the signal oscillations in the TFM image and enables a more robust maximum amplitude measurement. While calculating the envelope increases the computation burden on the software, it enables you to decrease the grid resolution and, as a result, increase the maximum **Acq. Rate.**

2.7.2.4 Wave Set and Zone

Using the **Zone** parameter, you can see and modify the **Min. Index**, **Max. Index**, **Min. Depth**, and **Max. Depth** options. To access these options, go to **TFM Settings > Wave Set and Zone** (see Figure 2-23 on page 62 and Table 16 on page 62).

Figure 2-23 TFM Settings — Wave Set and Zone

Table 16 TFM Settings — Wave Set and Zone

Option	Description
Wave Set	Displays the type of wave set selected in the scan plan.
Min. Index	Used to set the limit for the left side of the TFM zone (orange outline in the scan plan representation). For weld inspections, zero is in the middle of the weld.
Max. Index	Used to set the limit for the right side of the TFM zone (orange outline in the scan plan representation). For weld inspections, zero is in the middle of the weld).
Min. Depth	Used to set the upper limit for the TFM zone (orange outline in the scan plan representation.
Max. Depth	Used to set the lower limit for the TFM zone (orange outline in the scan plan representation).

2.7.2.5 Zone Resolution

Using the **Zone Resolution** parameter, you can see and modify the **Resolution**, **pts/λL**, **pts/λT**, and **Amplitude Fidelity** options. To access these options, go to **TFM Settings > Zone Resolution** (see Figure 2-24 on page 62 and Table 17 on page 63).

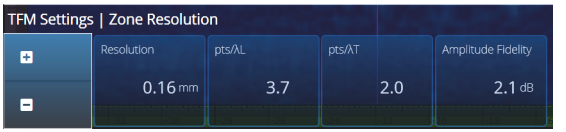
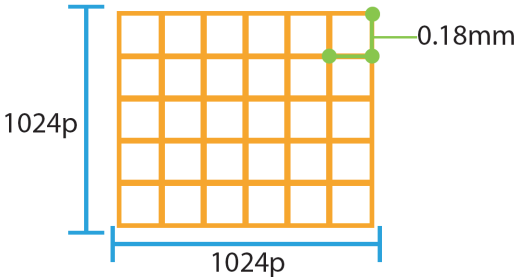


Figure 2-24 TFM Settings — Zone Resolution

Table 17 TFM Settings — Zone Resolution

Option	Description
Resolution	Used to set the distance between two pixels inside the TFM zone. Adjust the grid resolution to obtain a code-compliant **Amplitude Fidelity.** 
pts/λL	Displays the number of points per longitudinal wavelength, which is determined by the grid resolution setting.
pts/λT	Displays the number of points per transversal wavelength, which is determined by the grid resolution setting.
Amplitude Fidelity	Displays the maximum possible amplitude variation (in dB) caused by the grid resolution itself. This model is based on empirical observations and takes into account the horizontal and vertical axes.

2.7.2.6 Aperture

Using the Aperture parameter, you can see the pulser and receiver settings as they were set up in the scan plan.

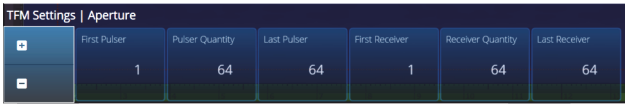


Figure 2-25 TFM Settings — Aperture

Table 18 TFM Settings — Aperture

Option	Description
First Pulser	Indicates the element No. used as the first element in the pulser.
Pulser Quantity	Indicates the number of elements used for the pulser.
Last Pulser	Indicates the element No. used as the last element in the pulser.
First Receiver	Indicates the element No. used as the first element in the receiver.
Receiver Quantity	Indicates the number of elements used for the receiver.
Last Receiver	Indicates the element No. used as the last element in the receiver.

2.7.3 Gates & Alarms

The **Gate & Alarms** menu enables you to access the **Gate Main**, **Gate Advanced**, **Alarm**, **Output**, and **Thickness** parameters.

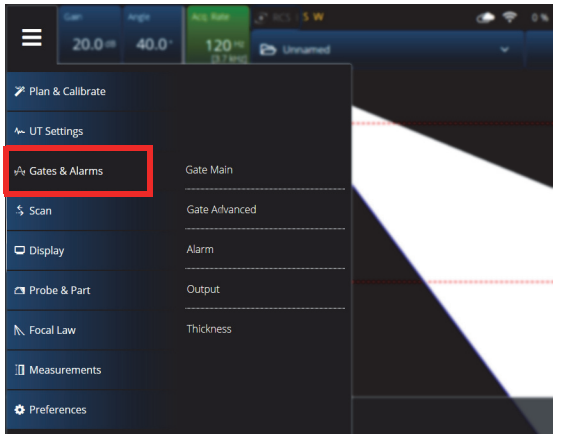


Figure 2-26 Gates & Alarms

2.7.3.1 Gate Main

Using the **Gate Main** parameter, you can see and modify the **Select Gates**, **Activation**, **Geometry, Start, Width**, and **Threshold** options. To access these options, go to **Gate & Alarms > Gate Main** (see Figure 2-27 on page 65 and Table 19 on page 65).

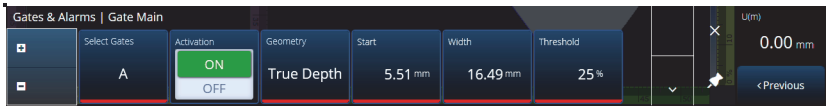


Figure 2-27 Gates & Alarms PA — Gate Main menu

Table 19 Gates & Alarms PA — Gate Main menu

Option	Description
Select Gates	Used to select which gate parameters will be edited. You can choose between **A**, **B,** or **I**.
Activation	Used to set the gate on the screen **ON** or **OFF**.

Table 19 Gates & Alarms PA — Gate Main menu *(continued)*

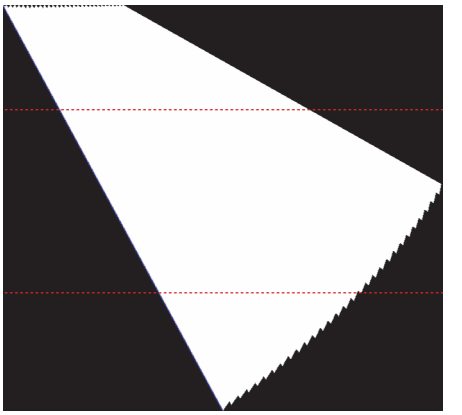
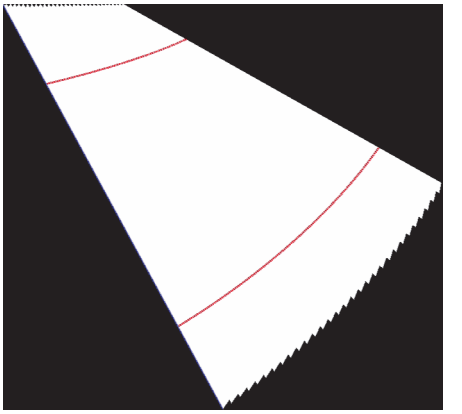
Option	Description
Geometry	Used to set the type of gate: **True Depth** or **Sound Path**.  **True Depth** sets the gate according to the depth in the material.  **Sound Path** sets the gate according to the distance traveled in the material.

Table 19 Gates & Alarms PA — Gate Main menu *(continued)*

Option	Description
Start	Used to set the starting position of the selected gate. This position is relative to the gate synchronization. The real position of the gate is the synchronization position plus the gate start position. If the gate is not synchronized, then the Start is relative to the zero on the ultrasound axis.
Width	Used to set the width of the gate (expressed in mm or in.).
Threshold	Used to set the height of the gate in the A-scan. This parameter determines the amplitude of a signal in the gate for detection.

2.7.3.2 Gate Advanced

Using the **Gate Advanced** parameter, you can see and modify the **Select Gates**, **Synchro**, **Peak, Measure**, and **Signal Polarity** options. To access these options, go to **Gate & Alarms > Gate Advanced** (see Figure 2-28 on page 67 and Table 20 on page 67).

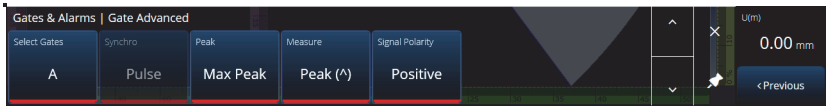


Figure 2-28 Gates & Alarms — Gate Advanced

Table 20 Gates & Alarms — Gate Advanced

Option	Description
Select Gates	Used to select which gate parameters will be edited. You can choose between **A**, **B,** or **I**.

Table 20 Gates & Alarms — Gate Advanced *(continued)*

Option	Description
Synchro (for **Gate A** and **Gate B)**	Used to specify the synchronization type of the selected gate. **Pulse**: Synchronizes at the beginning of the pulse. It is the only selection available when using a group type other than **Linear at 0°**. **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**. Gate **I** must be active to use this option. **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**. This option is available for gate **B** only if you have selected **Measure** = **Peak** for gate **A**. **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**. This option is available for gate **B** only if you have selected **Measure** = **Edge** for gate **A**.
A-scan Synchro	Used to specify the A-scan synchronization type. **Pulse**: Synchronizes at the beginning of the pulse. The ultrasound axis considers the **Wedge Delay** and **Beam Delay**, so the zero should be at the surface of the piece if the right side was selected in the scan plan. It is the only option available when using a group type other than **Linear at 0°**. **I/**: Synchronizes the zero of the ultrasound axis on the signal at the first crossing in the gate **I**. Gate **I** must be active to have this option. Wedge and beam delays are forced to 0 when choosing this option.

Table 20 Gates & Alarms — Gate Advanced *(continued)*

Option	Description
Peak	If the value of the **Measure** is set to **Peak (^)**, the **Peak** setting enables you to choose if the readings are related to **First Peak** or **Max Peak**. When **Max Peak** is selected for a specific gate (**A**, **B**, or **I**), the data, readings, and parameters displayed correspond only 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 correspond only to the first peak crossing this particular gate.
Measure	Used to set the measure type of the current gate. **Peak (^)**: The data, readings, and parameters displayed correspond to either the **Max Peak** or **First Peak**, depending on the **Peak** setting. **Edge (/)**: The data, readings, and parameters displayed correspond to the first crossing point in the gate. The **Peak** setting has no influence.
Signal Polarity	For rectified signals, the **Signal Polarity** is set to **Positive**, and is in read-only. For RF signals, the polarity is set to **Absolute**. When in the **Absolute** mode, all gates measurements consider the absolute value of the signal in the gate, regardless if the signal is positive or negative.

2.7.3.3 Alarm

On the **Alarm** menu, you can set an alarm on all groups, all gates or any individual group or gate. A maximum of three alarms can be set. (See Figure 2-29 on page 69.)

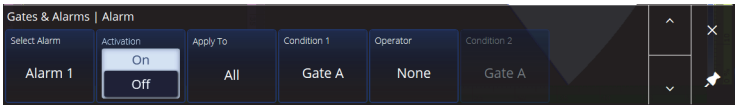


Figure 2-29 Gates & Alarms — Alarm menu

Table 21 Gates & Alarms — Alarm

Option	Description
Select Alarm	Choose the alarm to configure (**Alarm 1** to **Alarm 3**).
Activation	**On** or **Off** activates or deactivates the corresponding alarm indicator light visible on the instrument front panel (see Figure 2-30 on page 71).
Apply To	Select a specific group or apply to **All** groups.
Condition 1	Set the gate condition triggering the alarm. The condition can be set to trigger the alarm when the signal meets a specific gate (e.g., **Gate A**) or when the signal does not meet a specific gate (e.g., **Not Gate A**).
Operator	Select a logical operator linking the two conditions. The **And** operator activates the alarm when both conditions are met. The **Or** operator activates the alarm when either condition is met.
Condition 2	Set a second gate condition triggering the alarm. The condition can be set to trigger the alarm when the signal meets a specific gate (e.g., **Gate B**) or when the signal does not meet a specific gate (e.g., **Not Gate B**).

TIP

The left alarm indicator light is for **Alarm 1**, the middle is for **Alarm 2** and the right for **Alarm 3** (see Figure 2-30 on page 71).

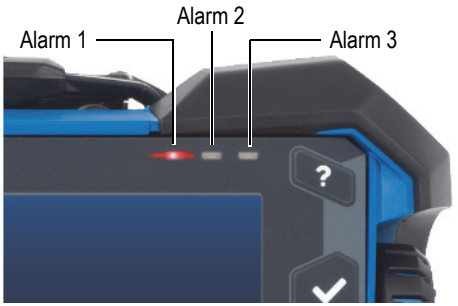


Figure 2-30 Alarm indicator lights

2.7.3.4 Output

The **Output** menu enables you configure an alarm signal and send it to a digital output.

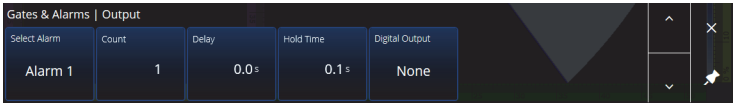


Figure 2-31 Gates & Alarms — Output menu

Table 22 Gates & Alarms — Output

Option	Description
Select Alarm	Choose the alarm signal to configure (**Alarm 1** to **Alarm 3**).
Count	Enter the number of times the alarm condition must be met before triggering the alarm.
Delay	Enter the delay between the occurrence of the alarm condition and the actual alarm trigger.
Hold Time	Enter the duration of the alarm.
Digital output	Use this to send the alarm signal to one of the three **DOUT** digital outputs.

2.7.3.5 Thickness

Using the **Thickness** parameter, you can set the source for thickness measurements, and define the minimum and maximum of the thickness color palette. To access these options, go to **Gate & Alarms > Thickness** (see Figure 2-32 on page 72 and Table 23 on page 72).

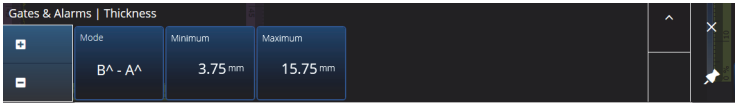


Figure 2-32 Gates & Alarms — Thickness

Table 23 Gates & Alarms — Thickness

Option	Description
Mode	Selects the gate combination used to measure thickness.
Minimum	The minimum thickness of the color scale for the thickness C-scan.
Maximum	The maximum thickness of the color scale for the thickness C-scan.

2.7.3.6 TFM Gates

The A gate is available when using TFM groups. Because the data in TFM is volumetric, a box gate is used that crops the data to a specific targeted zone within the end view.

There are no advanced or thickness controls for the TFM gate, so only the **Gates Main** menu is available (see Figure 2-33 on page 72 and Table 24 on page 73).

Figure 2-33 Gates & Alarms — TFM

Table 24 Gates & Alarms — TFM

Option	Description
Activation	Used to set the gate on the screen **On** or **Off.**
Index Start	Used to set the start position of the selected gate in the index direction. The **Index End** is updated with the **Index Start** to maintain the same gate width.
Index End	Used to set the end position of the selected gate in the index direction. The **Index Start** is not changed with the **Index End.**
Depth Start	The same control as the **Index Start** but in the **Depth** direction.
Depth End	The same control as the **Index End** but in the **Depth** direction.
Threshold	Used to set the height of the gate in the A-scan. This parameter determines the amplitude of a signal in the gate for detection.

When in the raster scan mode, the **Index Start** and **Index End** are read only and locked to the TFM zone **Index Start** and **Index End** values.

2.7.4 Scan

The **Scan** menu enables you to access the **Inspection** and **Area** parameters.

2.7.4.1 Inspection

Using the **Inspection** parameter, you can see and modify the **Type**, **Scan**, and **Encoder** options. To access these options, go to **Scan > Inspection** (see Figure 2-34 on page 73 and Table 25 on page 74).

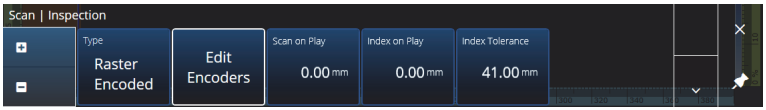
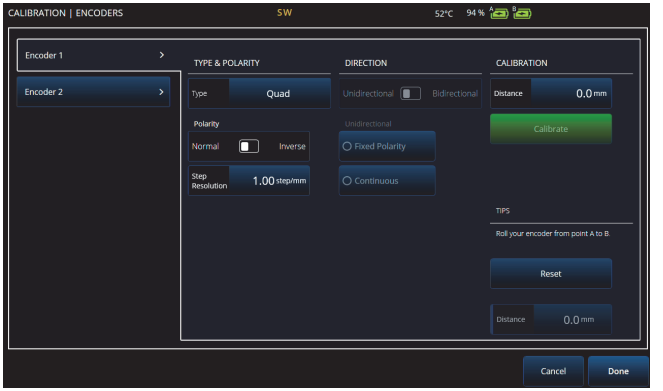


Figure 2-34 Scan — Inspection

Table 25 Scan — Inspection

Option	Description
Type	Used to select the desired type of inspection. The following choices are available: **Time** Data acquisition at precise time intervals. **One-Line Encoded** In one-line scanning, the acquisition is based on an encoder. **Raster Encoded** 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.
Edit Encoder	Used to configure the encoder settings. This option enables you to configure the encoder resolution, polarity, and input. Refer to "Encoder Configuration" on page 75 to know more about encoder options. 
Scan on Play	Defines the value of the Scan position that will be set when the user presses **Play**. The default value is the **Area Scan Start**.
Index on Play	Only available in **Raster Encoded**. Defines the value of the Index position that will be set when the user presses **Play**. The default value is the **Area Index Start**.

2.7.4.2 Encoder Configuration

On the **Edit Encoders** menu, you can select from a list of preset values or manually configure the encoders.

Scanner Presets

If you have an Evident scanner, you can choose it directly from the **Scanner Presets** tab (see Figure 2-35 on page 75). The resolution, input, and polarity will be automatically configured. You can still edit the parameters in the other available tabs (**Scan Axis Encoder** tab and the **Index Axis Encoder** tab).

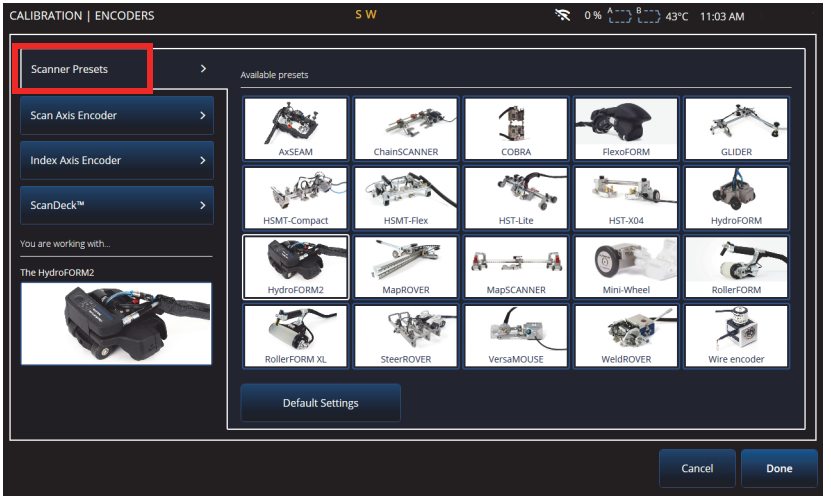


Figure 2-35 Scanner Presets

Scan Axis Encoder and Index Axis Encoder

The **Scan Axis Encoder** tab and the **Index Axis Encoder** tab enable you to select and configure the encoder for each axis. It is also possible to calibrate the encoders on this menu. To access these options, go to **Scan > Inspection** (see Figure 2-34 on page 73 and Table 26 on page 77), and then select **Edit Encoders**.

ScanDeck

When the HydroFORM2 (Next-generation HydroFORM) scanner is selected, an additional menu is displayed in **Scanner Presets**. In this menu you can modify the encoder settings for the HydroFORM2.

You can adjust the **Target Increment** which sets the nominal index distance between each scan line. Also you can set the **Warning Tolerance** to allow a margin before getting a warning that the index distance has been exceeded.

ScanDeck Quick Guide shows the Scandeck button usage for the Next-generation HydroFORM. (See Figure 2-36 on page 76.)

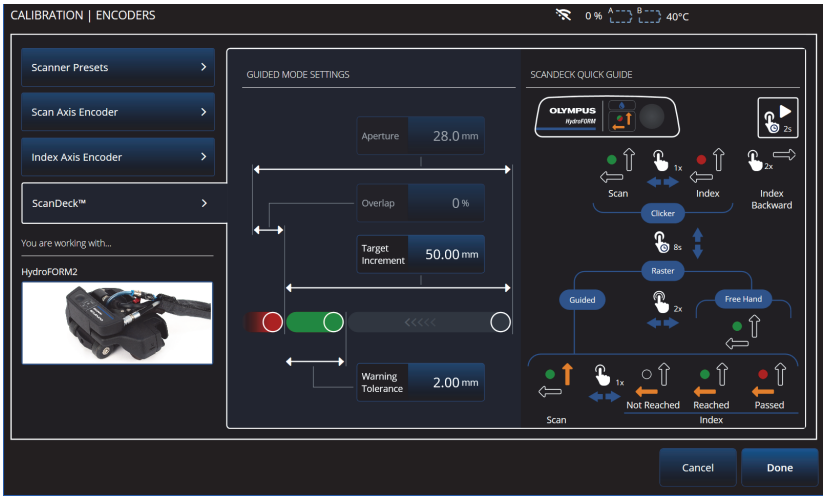


Figure 2-36 HydroFORM 2 ScanDeck

Table 26 Scan — Encoder configuration

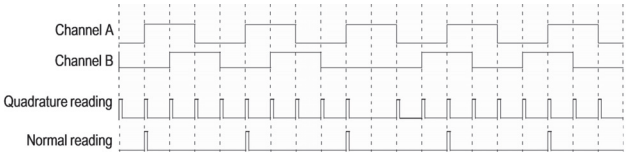
Option	Description
Type	Select the encoder type. Current choices are **Quad** and **Clicker**. Select **Quad** when the attached encoder (5 V TTL output) is a dual-channel output encoder. The channels are generally designated A and B. When the encoder is rotating clockwise (from left to right in the figure below), 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. 
Clicker	Used when you use an Evident indexer device (clicker). Press the push-button of the indexer device to increment the position on the axis. The clicker is frequently used to perform a manual raster scan and is generally assigned to the **Index** axis.
Step Resolution	If the encoder type is **Quad**, then the resolution is the number of encoder counts per unit for the selected encoder. With the **Clicker** type, the resolution is the increment in the axis when the clicker is pressed.
Polarity	Used to reverse the count of the encoder. Select **between Normal** and **Inverse.**
Encoder Input	Select the input source for the selected axis. In a Raster scan, choosing an input for the Scan Axis automatically selects the input for the Index Axis.

Table 26 Scan — Encoder configuration *(continued)*

Option	Description
Preset	When using a clicker, it is possible to turn **ON/OFF** the preset at a fixed value. If the **Preset** is **OFF**, when pressing the clicker, the scan axis value remains the same. When **Preset** is **ON**, pressing the clicker will change the scan axis encoder value to the axis origin. Can be used to simplify the inspection workflow and adjust it to your scanning pattern.
Calibration	To calibrate the encoder resolution, first define the real distance the encoder will travel. Then click **Reset** to restart the encoder count and move the encoder by the specified distance. Then press **Calibrate** to convert the encoder count and the distance to the **Encoder Resolution**. **Distance**: Used to set the distance for the calibration. **Calibrate**: Used to confirm the distance for the calibration. **Reset**: Reinitializes the distance of the encoder to 0. **Distance** (bottom): Displays the actual distance that the encoder traveled.
Index start bound on clicker step	This option is only available with a **0° with overlap** group, and when the Index axis is set to **Clicker**. Turning ON this feature forces the Index Start value to be a multiple of the clicker step, or clicker resolution. The typical use case is the inspection of a pipe with the FlexoFORM. In this case, the zero index is referenced on the top of the pipe, with the index Start and Index End set on each side of the reference (Index Start is negative). With **Index start bound on clicker step** set to ON, this guarantees that using the clicker, the index position will pass exactly by zero, exactly on the reference. It avoids unnecessary calculation to match the Index Start perfectly.

2.7.4.3 Area

Using the **Area** parameter, you can see and modify the **Scan Start**, **Scan End**, and **Scan Res.** options. To access these options, go to **Scan > Area** (see Figure 2-37 on page 79 and Table 27 on page 79).

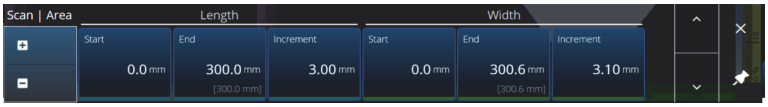


Figure 2-37 Scan — Area

Table 27 Scan — Area

Option	Description
Scan Start	Used to set the starting location of the scan (expressed in mm or in.).
Scan End	Used to set the maximum distance you can scan (expressed in mm or in.).
Scan Res.	Used to set the step (resolution) at which the points will be acquired on the scan (expressed in mm or in.).
Index Start	(Raster scan only) Used to set the starting location of the raster scan in the index axis (expressed in mm or in.).
Index End	(Raster scan only) Used to set the raster end location in the index axis (expressed in mm or in.).
Index Res./Index Step	(Raster scan only) Determines the index resolution. Cannot be modified in the **Linear at 0°** scan.

2.7.4.4 Digital Inputs

The **Digital Inputs** option enables you to configure the digital inputs (DIN). Each of the four **DIN*n*** parameters has an exclusive function. The listed functions can be assigned to any digital input (see Table 28 on page 80).

Use the digital inputs to control the OmniScan X3 flaw detector remotely. Connect your remote controller to the appropriate OmniScan connector. Refer to the *OmniScan X3 User's Manual* for details on the signals and connectors.

When using a preset scanner that has digital input by default, the **Digital Inputs** section will already be filled.

Table 28 Digital input options

Option	Description
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 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 pressing the Save 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 pressing the Play key ( ).
Acquisition step	While holding this DIN active, the acquisition is temporarily frozen. You can only define this option on DIN 3.

2.7.5 Probe & Part

The **Probe & Part** menu enables you to edit parameters related to the positioning and overlay, as well as create your custom probes and wedges in the **Probe & Wedge Manager**.

2.7.5.1 Position

Using the **Position** parameter, you can see and modify the **Skew**, **Scan Offset**, and **Index Offset** options. To access these options, go to **Probe & Part > Position** (see Figure 2-38 on page 81 and Table 29 on page 81).

Figure 2-38 Probe & Part —Position

Table 29 Probe & Part — Position options

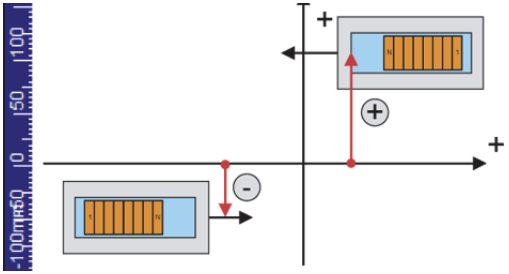
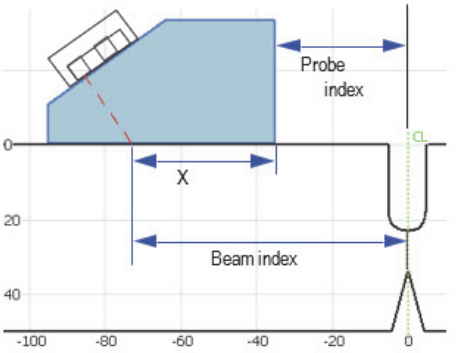
Option	Description
Skew	Orientation of the ultrasonic beam relative to the scan axis. Skew 90 and skew 270 are typically used to define a two-sided, two-probe inspection.
Scan Offset	The **Scan Offset** is the difference between the 0 position marked on the part for inspection and the real start position for the probe center on the scan axis. 

Table 29 Probe & Part — Position options *(continued)*

Option	Description
Index Offset	The beam index offset 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. The **Index Offset** cannot be edited in **TFM**, because it affects the calculation of the focal laws. Use the **Scan Plan** to change the Probe Index Offset in TFM. 

2.7.5.2 Part

Using the **Part** parameter, you can see and modify the **Thickness** option. To access this option go to **Probe & Part > Part** (see Figure 2-39 on page 82 and Table 30 on page 83).

Figure 2-39 Probe & Part — Part

Table 30 Probe & Part — Part

Option	Description
Thickness	Used to set the thickness of the part you will scan. This value is mostly used to fit the overlay and the legs on the signal by adjusting to the real thickness rather than the nominal value. This value cannot be edited in **TFM**, because it affects the calculation of the focal laws. Use the **Scan Plan** to change the **Part Thickness** in TFM.

2.7.5.3 Probe & Wedge Manager

To manage custom probes and wedges, see “Probe & Wedge Manager” on page 197.

2.7.5.4 Weld or Custom Overlay

The title of this submenu varies depending on the choice made in the scan plan. If no overlay is selected, this menu does not appear. If a weld overlay is selected, the **Weld** menu enables you to edit directly (see the description for each parameter in Table 64 on page 138) the following parameters:

- Hot Pass Height
- Hot Pass Angle
- Land Height
- Land Offset
- Root Height
- Root Angle

Parameters that are not relevant or cannot be edited because they depend on other values are read-only.

If the option selected in the scan plan for the overlay is **Custom**, then this section is titled **Custom Overlay** and the following parameters are available to edit:

- Scale
- Rotate
- Horizontal/Vertical Pan
- Horizontal/Vertical Flip

The description for each parameter can be found in “Part & Weld Tab” on page 132.

2.7.6 Focal Laws

The **Focal Laws** menu enables you to access the **Aperture** and **Beam** parameters.

2.7.6.1 Aperture

Using the **Aperture** parameter, you can see and modify the **Element Qty**, **First Element**, and **Last Element** options. To access these options, go to **Focal Laws > Aperture** (see Figure 2-40 on page 84 and Table 31 on page 84).

Figure 2-40 Focal Laws — Aperture

Table 31 Focal Laws — Aperture

Option	Description
Element Qty	Used to set the number of elements of each aperture.
First Element	Used to set the first element of the first aperture.
Last Element	Used to set the last element of the last focal law.
Element Step	Used to see the element step between each focal law when the **Linear** scan type is chosen.

2.7.6.2 Beam

Using the **Beam** parameter, you can edit the focal laws directly, avoiding going back and forth to the scan plan. To access the **Min. Angle**, **Max. Angle, Angle Step**, **Angle**, **Focus**, and **Skew Angle** options, go to **Focal Laws > Beam** (see Figure 2-41 on page 85 and Table 32 on page 85).

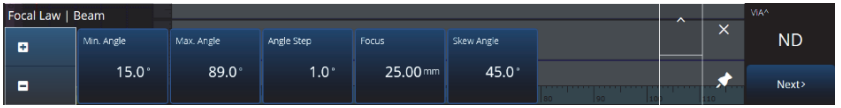


Figure 2-41 Focal Laws —Beam

Table 32 Focal Laws — Beam

Option	Description
Min. Angle	Used to set the minimum angle of the beam (**Sectorial/Compound** law configuration).
Max. Angle	Used to set the maximum angle of the beam (**Sectorial/Compound** law configuration).
Angle Step	Used to set the step value between each angle (**Sectorial/Compound** law configuration).
Angle	Used to set the refracted angle of all the beams (**Linear** law configuration).
Focus	Used to set the focusing depth of the part to be inspected.
Skew Angle	Used to steer the beams at an angle different than the nominal beam skew. This option requires probes with the ability to steer the beam in the passive axis (matrix probes).

2.7.7 Measurements

The **Measurements** menu enables you to access the **Cursors** parameter.

Cursors

Using the **Cursors** parameter, you can edit the cursor position. Cursors can also be moved by tapping directly on the cursor in the layout. To access the **Cursors** tab, go to **Measurements > Cursors** (see Figure 2-42 on page 86 and Table 33 on page 86).

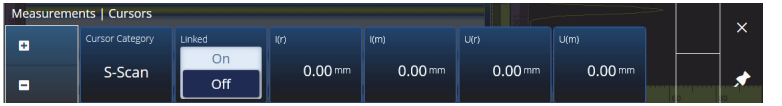


Figure 2-42 Measurements — Cursors

Table 33 Measurements — Cursors

Option	Description
Cursor 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**, **S-scan**, **TFM End View**, **TFM Side View**, **TFM Top View**, or **Data**) depends on the current layout selected. The parameters appearing to the right of **Category** apply to the selected view.
Linked	Used to determine whether the reference and measurement cursors are moved individually (**Off**) or simultaneously (**On**). This parameter affects the parameters in the **Measurements** > **Cursors** submenu and the pop-up button for the cursor parameter.
%(...)	The position on the amplitude axis of the reference cursor (r), (r&m), or the measurement cursor (m).
Delta %(r&m)	The difference on the amplitude axis between the reference cursor and the measurement cursor (only when cursors are linked).
U(...)	The position on the ultrasound axis of the reference cursor (r), (r&m), or the measurement cursor (m).
Delta U (r&m)	The difference on the ultrasound axis between the reference cursor and the measurement (only when cursors are linked).
I (...)	The position on the index axis of the reference cursor (r), (r&m), or the measurement cursor (m).

Table 33 Measurements — Cursors *(continued)*

Option	Description
Delta I (r&m)	The difference on the index axis between the reference cursor and the measurement (only when cursors are linked).
S (...)	The position on the scan axis of the reference cursor (r), (r&m), or the measurement cursor (m).
Delta S (r&m)	The difference on the scan axis between the reference cursor and the measurement (only when cursors are linked).
D (...)	The position on the depth axis in TFM of the reference cursor (r), (r&m), or the measurement cursor (m).
Delta D (r&m)	The difference on the depth axis in TFM between the reference cursor and the measurement (only when cursors are linked).

2.7.8 Display

The **Display** menu enables you to access various display parameters.

2.7.8.1 Compliance

Using the **Compliance** parameter, you can add compliance curves (sizing curves with a dB offset) to the sizing curves. It is available if a TCG or DAC calibration has been performed. If a DGS calibration is applied, use the menu **Scan Plan > Manage DGS** (see Figure 2-43 on page 88 and Table 34 on page 88).

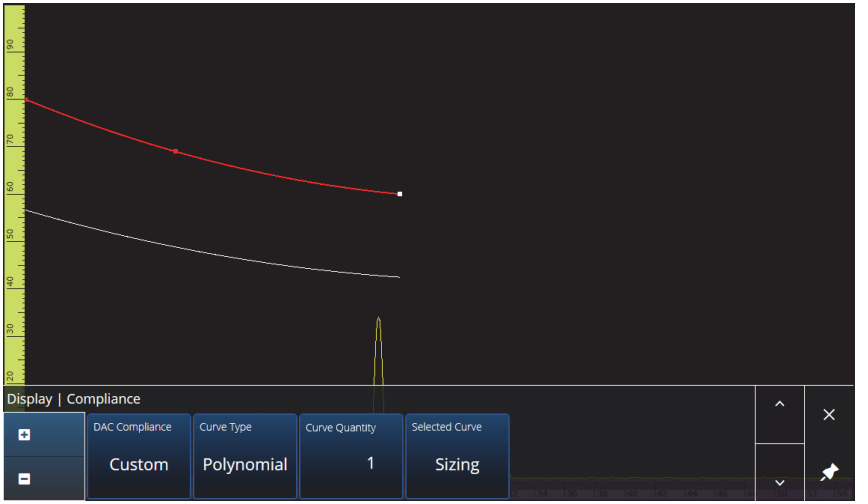


Figure 2-43 Display — Compliance

Table 34 Display — Compliance

Option	Description
DAC Compliance	Used to apply preset compliance curves according to a specific code (JIS or ASME). Choose **Custom** to manually create curves.
Curve Type	Used to select the type of interpolation between the DAC points: **Linear** or **Polynomial**.
Curve Quantity	Used to display the number of compliance curves to manage.
Selected Curve	Used to select the compliance curve to edit. The default selection is the **Sizing** curve, which cannot be edited. Select another curve to edit its **Amplitude Offset**.
Amplitude Offset	The dB difference between the **Sizing** curve and the selected compliance curve.

2.7.8.2 Overlay

The **Overlay** parameter enables you to use multiple legs (**On**) or a single leg (**Off**) in the Overlay. The use of multiple legs will cause the weld or custom overlay to be flipped at each rebound.

2.7.8.3 Data Source

On the Data Source menu, you can modify the **A-Scan**, **Primary C-scan**, **Secondary C-scan**, and **Merged B-Scan** options. To access these options, go to **Display > Data Source** (see Figure 2-44 on page 89 and Table 35 on page 89).

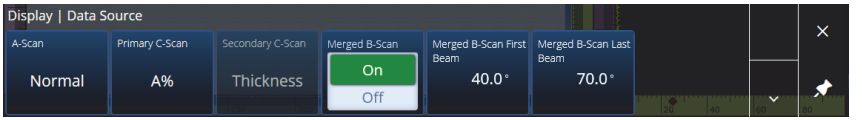


Figure 2-44 Display — Data Source

Table 35 Display — Data Source

Option	Description
A-Scan	Select which A-scan is displayed on the A-scan view. **Normal**: The displayed A-scan is the one that is currently selected; using the data cursor or the Angle/VPA selector in the top bar. **Highest** (%): The data cursor automatically tracks the focal law with the highest amplitude in Gate A. If no signal crosses the threshold, then the selected A-scan defaults to the first one. **Thinnest**: The data cursor automatically tracks the focal law with the thinnest measured thickness. Make sure that the thickness measurement is correctly defined in **Gates & Alarms > Thickness > Mode**. **NOTE** **Highest** and **Thinnest** tracking modes are not available in analysis mode. And, when these tracking modes are active, all layouts containing a B-scan are deactivated.

Table 35 Display — Data Source *(continued)*

Option	Description
Primary C-scan	Select the source of the C-scan for all layouts containing a C-scan. The source can be **A%, B%, I%, I/**, or **Thickness**. Some options may not be available if the related gate is not active. For a Thickness C-scan, choose the **Thickness** mode in **Gates & Alarms > Thickness > Mode**.
Secondary C-scan	Select the source of the second C-scan in an A-C-C layout. The source can be **A%, B%, I%, I/**, or **Thickness**. Some options may not be available if the related gate is not active. For a Thickness C-scan, choose the Thickness mode in **Gates & Alarms > Thickness > Mode**.
Merged B-Scan	Choose to turn **On** or **Off** the **Merged B-Scan** in A-B-S and A-B-C-S layouts.
Merged B-Scan First Beam	Used to change the angle of the first beam. Data below the set angle is not displayed in the merged B-Scan.
Merged B-Scan Last Beam	Used to change the angle of the last beam. Data above the set angle is not displayed in the merged B-Scan.

When in TFM mode, the **Data Source** menu can be edited to select how the data is represented (See Figure 2-45 on page 90 and Table 36 on page 90). Because the gate in TFM is a box shaped, the data source has an influence on both the top and the end views.

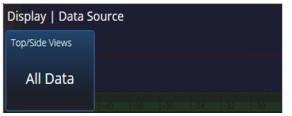


Figure 2-45 Display — Data Source, TFM mode

Table 36 Display — Data Source, TFM mode

Option	Description
All Data	Displays all the data in the **End View.**

Table 36 Display — Data Source, TFM mode

Option	Description
Gate A	Displays only the data within the A gate, in **Top** and **End views.**

2.7.8.4 Grid

Using the **Grid** parameter, you can see and modify the A-scan background grid parameters. To activate the grid, use the menu **View** and turn the **Grid** on. To access these options, go to **Display > Grid** (see Figure 2-46 on page 91 and Table 37 on page 91).

Figure 2-46 Display — Grid

Table 37 Display — Grid

Option	Description
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.
Color	Used to set the color of the grid.

2.7.8.5 Cursors and Axes

Using the **Cursors and Axes** parameter, you can display and modify the **Values** and **C-Scan Axes** options. To access this option go to **Display > Cursors** (see Figure 2-47 on page 92 and Table 38 on page 92).

Figure 2-47 Display — Cursors and Axes

Table 38 Display — Cursors and Axes

Option	Description
Values	Used to display the values (expressed in mm or in.) on the various cursor by tapping the **Cursor Values** button to turn it **ON** or **OFF** (default).
C-Scan Axes	Used to toggle the orientation of the index axis.

2.7.8.6 Default Zoom

Using the **Default Zoom** parameter, you can see and modify the **Default Zoom** options. To access this option go to **Display > Default Zoom** (see Figure 2-48 on page 93 and Table 39 on page 92).

Table 39 Display — Default Zoom

Option	Description
Scan Default Zoom	Used to set the size of the zoom window when the default zoom is applied.

Table 39 Display — Default Zoom *(continued)*

Option	Description
Set to Scan Default Zoom	To use the preset default zoom, these conditions must be met: • The inspection is a one-line scan or raster scan. • The current layout must contain a C-scan and/or a B-scan view. • The C-scan or B-scan must already be in zoom mode. Zoom in a C-scan or B-scan, and click **Set to Scan Default Zoom**. This will change the zoom length on the scan axis to the preset value.

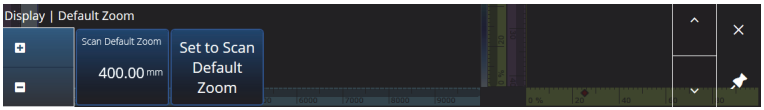


Figure 2-48 Display — Default Zoom

2.7.9 Preferences

The **Preferences** menu enables you to access the **Date & Time**, **Regional**, **Data**, **Connectivity Settings, System**, and **About** parameters.

2.7.9.1 Date & Time

Using the **Date &Time** parameter, you can see and modify the **Time Zone**, **Clock Format**, and **Date Format** options. To access these options, go to **Preferences > Date &Time** (see Figure 2-49 on page 93 and Table 40 on page 94).

Figure 2-49 Preferences — Date & Time

Table 40 Preferences — Date & Time

Option	Description
Time Zone	Used to set the time zone of your instrument. **IMPORTANT** The instrument may be unable to connect to the **CLOUD** if the time zone is incorrectly set.
Clock Format	Used to set the format of the clock. You have the choice between **12h** or **24h**.
Date Format	Used to set the date format. You have the following choices: **YYYY/MM/DD** **YYYY-MM-DD** **MM-DD-YYYY** **MM/DD/YYYY** **DD-MM-YYYY** **DD/MM/YYYY**

2.7.9.2 Regional

Using the **Regional** parameter, you can see and modify the **Units**, **Decimal Separator**, **Thousands Separator**, **Adjust Time**, and **Adjust Date** options. To access these options, go to **Preferences > Regional** (see Figure 2-50 on page 94 and Table 41 on page 95).

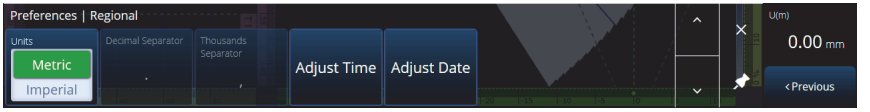


Figure 2-50 Preferences — Regional

Table 41 Preferences — Regional

Option	Description
Units	Used to set the length measurement units to either metric (millimeters) or US customary (inches).
Decimal Separator	Displays the decimal separator.
Thousands Separator	Displays the thousands separator.
Adjust Time	Used to set the time of your instrument.
Adjust Date	Used to set the date of your instrument.

2.7.9.3 Data

Using the **Data** parameter, you can see the **Scan Storage** settings and modify the **Geolocation** options. To access these options, go to **Preferences > Data** (see Figure 2-51 on page 95 and Table 42 on page 95).

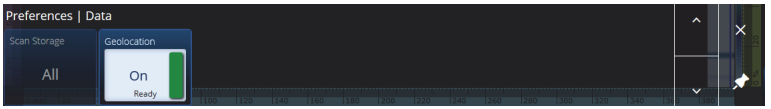


Figure 2-51 Preferences — Data

Table 42 Preferences — Data

Option	Description
Scan Storage	Displays which scans can be saved.

Table 42 Preferences — Data *(continued)*

Option	Description
Geolocation	Used to turn **ON** the geolocation to include the GPS coordinates in the data file. If the instrument is not connected to a wireless LAN network, the geolocation module obtains its position using GPS satellites. Though this makes it slower to acquire its position, it offers high precision in the field, yet low precision indoors. If the instrument is connected to a wireless network, then the instrument can use the network to get its location (offering rapid geolocation and better precision indoors, yet lower precision if the network connection is weak).

2.7.9.4 Connectivity Settings

The **Connectivity Settings** enable you to turn ON or OFF the **Wireless**, **OSC Connect** (required to use the X3 RCS), and **OneDrive** options (see Figure 2-52 on page 96).

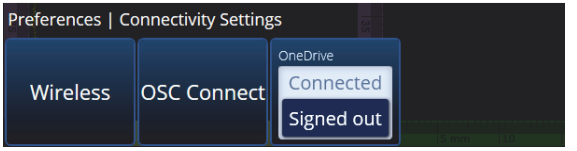


Figure 2-52 Preferences — Connectivity Settings

Wireless

Using the **Wireless** parameter, you can see and modify the **Wireless Enabled**, **Security**, **Password**, **Show Password**, **Advanced Options**, **Add Network**, **Refresh**, **Done**, and **Connect** parameters. To access these options, go to **Preferences > Wireless** (see Figure 2-53 on page 97 and Table 43 on page 97).

In the **Wireless Properties** window, the security level of your selected network is automatically detected.

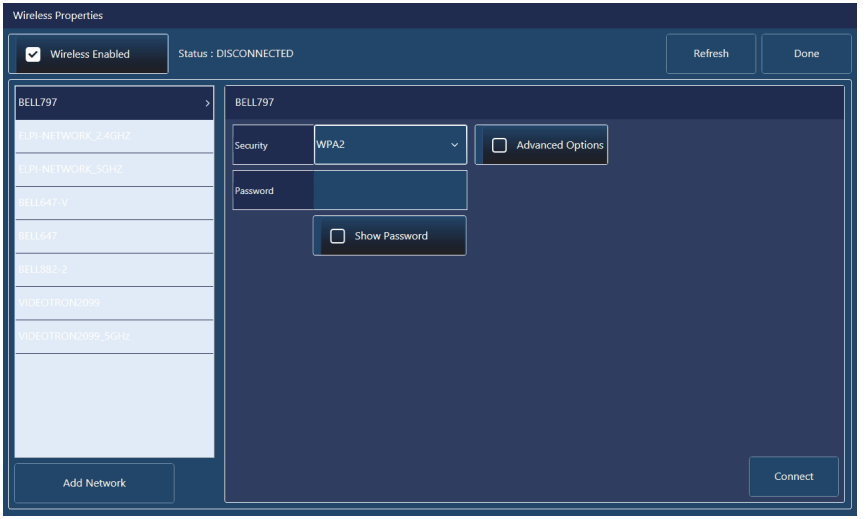


Figure 2-53 Wireless Properties window

Table 43 Preferences — Connectivity Settings — Wireless

Option	Description
Wireless Enabled	The check box used to turn the **Wireless Enabled** on. A check mark means that it is enabled.
Security	Used to indicate the security level of the chosen wireless network, such as **WEP**, **WPA**,**WPA2,** and **EAP**.
Password	Used to enter the password of the chosen network.
Show Password	Used to reveal or hide the password.
Advanced Options	Used to set different options, such as enable the **DHCP**, enter the **IP Address** manually, enter the **Subnet Mask** manually, enter the **Gateway** manually, enter the **DNS** Server 1 manually, and enter the **DNS** Server 2 manually (for the **WPA2**).
Add Network	Used to manually add a wireless network with different options, such as **Security** and **Network Name**.

Table 43 Preferences — Connectivity Settings — Wireless *(continued)*

Option	Description
Refresh	Used to refresh the available wireless networks.
Done	Used to close and confirm.
Connect	Used to connect to the chosen wireless network.

OSC Connect

To use the X3 Remote Collaboration Service (X3 RCS), your OmniScan X3 unit needs a valid connection to the Olympus Scientific Cloud (OSC). See "Olympus Scientific Cloud (OSC) Connection" on page 217.

OneDrive

IMPORTANT

Uploading and downloading files from the OneDrive cloud storage is done in the File Manager. See the "Using the File Manager" on page 192 for more details.

To connect to OneDrive

1. Select the **OneDrive** button to begin the login process. You must repeat this process if you reboot the OmniScan X3 unit because user names and passwords are not saved on the instrument for safety reasons.
2. You must read and accept the **Privacy Statement** to use OneDrive.
3. Enter your login. If you do not have a OneDrive account, you must create one using another device (account creation is locked on the OmniScan X3 unit).
4. Enter your password.
5. If required, enter the PIN to complete the two-factor authentication.

To disconnect from OneDrive

- Select the **OneDrive** button to disconnect. If you reboot the OmniScan X3 unit, the connection with OneDrive is lost.

2.7.9.5 System

Using the **System** parameter, you can turn off the automatic launch of the MXU software if it was activated. To access this, go to **Preferences > System** (see Figure 2-54 on page 99 and Table 44 on page 99).

Figure 2-54 Preferences — System

Table 44 Preferences — System

Option	Description
Boot Launcher	Used to set the start-up of the OmniScan X3 flaw detector to either **Manual** (accesses the launcher) or **Automatic** (continues to the MXU software automatically).

2.7.9.6 About

Using the **About** parameter, you can verify the **System Information**, **Legal Information**, **Licenses**, and **FCC** information. To access these options, go to **Preferences > About** (see Figure 2-55 on page 100 and Table 45 on page 100).

Figure 2-55 Preferences — About window

Table 45 Preferences — About

Option	Description
System Information	Displays the **Model**, **Software Version**, **Manufacturer**, and **Details**. The details may vary from a version to another, but generally includes the list of the new features with respect to the previous version.
Legal Information	Displays the legal information, such as the patent rights protection.
Licenses	Displays the different license agreements from Evident.
FCC	Displays the Federal Communications Commission (FCC) Supplier's Declaration of Conformity.
Done	Used to confirm the terms of this section and exit this window.

2.8 View Menu

The **View** menu provides a range of submenus for inspection configuration (see Figure 2-56 on page 101 and Table 46 on page 101).

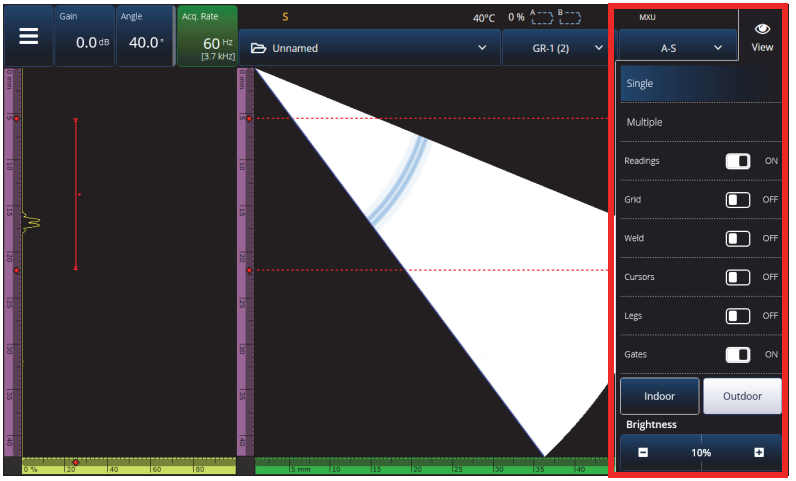


Figure 2-56 View menu window

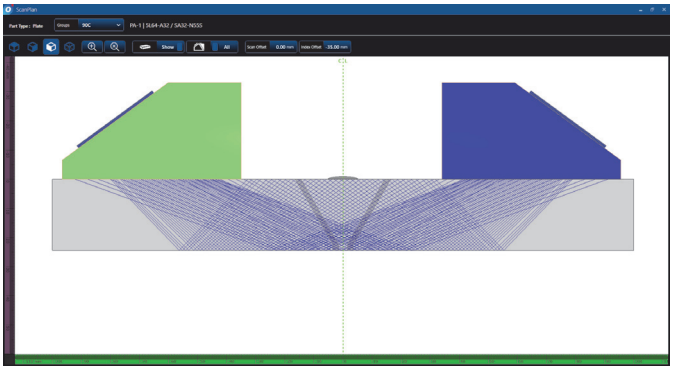
Table 46 View menu options

Option	Description
Single / Multiple	The **View** menu enables you to display the current group (**Single**) or multiple groups (**Multiple**).
Readings	To display the readings on the right of the screen, tap the **Readings** toggle button (**ON/OFF**) to rapidly activate or deactivate the **Readings** display.
Grid	To display the grid on the A-scan, tap the **Grid** toggle button (**ON/OFF**) to rapidly activate or deactivate the **Grid** display.

Table 46 View menu options *(continued)*

Option	Description
Weld/Overlay	To display the overlay of the weld on the S-scan, tap the **Weld** toggle button (**ON/OFF**) to rapidly activate or deactivate the **Weld** display. If a custom overlay is selected, this item is displayed as **Overlay** and can also be toggled **ON/OFF**.
Cursor	To display the cursors on every scan view, tap the **Cursor** toggle button (**ON/OFF**) to rapidly activate or deactivate the **Cursor** display.
Legs	To display the legs on every scan view, tap the **Legs** toggle button (**ON/OFF**) to rapidly activate or deactivate the **Legs** display.
Gates	To display the gates using the **View** menu, tap the **Gates** toggle button to rapidly activate or deactivate the gates display. At least one gate should be activated to enable the display. Make sure that the required inspection gates are activated in **Gates & Alarms > Gates Main**.
Brightness	Tap the minus button to decrease the screen brightness or tap the plus button to increase the screen brightness (expressed in a percentage).
Indoor/Outdoor	Toggle between the **Outdoor** or **Indoor** color scheme. The Indoor color scheme has a dark background with white text, while the Outdoor scheme has a white background with dark text for better contrast.

Table 46 View menu options *(continued)*

Option	Description
Scan Plan	In OmniPC software, there is an additional option on the **View** menu, the **Scan Plan** view. Selecting this Scan Plan option opens a window that contains a schema of the scan plan. The following parameters are available: • **Group** selection • **View** orientation (Top, Side, End, 3D) • **Zoom** • **Show** part (ON/OFF) • **Show** all groups/current group • **Scan Offset** of the current group • **Index Offset** of the current group 

NOTE

The 👁 **View** menu enables or disables the display of the gates, but the gates can still be used for your setup. However, if the **Activation** parameter is set to **OFF** (under **Gates & Alarms > Gates Main**), the gates are disabled and cannot be used for your setup.

If your inspected part includes an overlay (weld or custom), you can toggle the visibility of the overlay. The overlay is a drawing of the weld geometry or your selected drawing superimposed on the S-scan view. This feature can help you

visualize where indications are located relative to the part or weld geometry (see Figure 2-57 on page 104). Overlay reflection (to take into account the second leg, third leg, etc.) can be turned ON/OFF in the **Menu > Display > Overlay > Multiple Legs**.

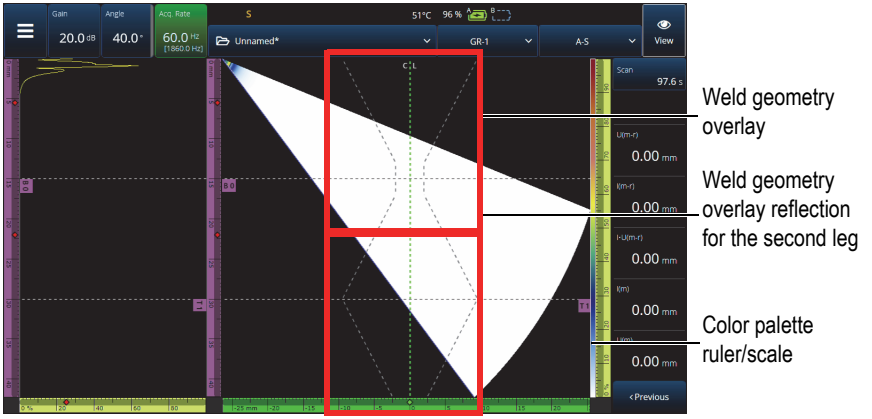


Figure 2-57 Example of a V-Offset weld geometry overlay

2.9 Scan and Index Indicators and Parameters

The **Scan** and **Index** parameters (see Figure 2-58 on page 105) have two purposes. The values in the Scan and Index fields indicate the current position of the data cursors, but they can also be used to change the position of the data cursors.

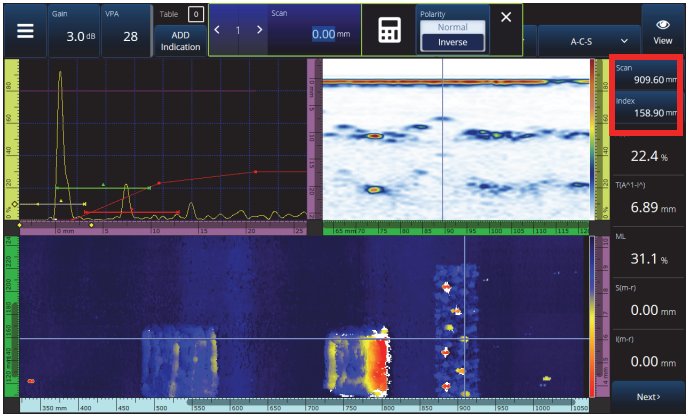


Figure 2-58 Scan and Index indicators and parameters

Table 48 on page 109 presents the **Scan** and **Index** functions depending on the configuration and acquisition mode.

Table 47 Scan and Index functions

	During acquisition		During analysis (paused)	
Inspection type	**Scan**	**Index**	**Scan**	**Index**
Time	Displays elapsed time since the acquisition began [Play key ( )].	N/A	Navigates the data by scrolling along the scan axis or jumping to a specific location.	N/A
One-Line Encoded	Reads the current position along the scan axis. Jumps to a specific scan position in order to set the encoder value on the fly.	N/A	Navigates the data by scrolling along the scan axis or jumping to a specific location.	N/A

Table 47 Scan and Index functions *(continued)*

	During acquisition		During analysis (paused)	
Raster Encoded	Reads the current position along the scan axis. Jumps to a specific scan position in order to set the encoder value on the fly.	Reads the current position along the index axis. Jumps to a specific index position in order to set the encoder value on the fly.	Navigates the data by scrolling along the scan axis or jumping to a specific location.	Navigates the data by scrolling along the index axis or jumping to a specific location.

Typical Use Cases

1. Setting or correcting the positions of the scan and index encoders during acquisition to compensate for obstacles.

   When scanning components with complex shapes and obstacles (such as a pressure vessel), you may need to correct the encoders' position read by the OmniScan X3 unit to reflect the actual probe position. The **Scan** and **Index** parameters enable you to edit the current encoder positions to a specific value, and "force" them into a specific position.

   To change the scan or index encoder position during data acquisition, follow these steps (order is important).

   a) Make sure that the scanner or probe is in the correct position and stationary.

   b) As needed, clear the data [Play key ( )]. This may not be desirable if data has already been acquired.

   c) Press either the **Scan** or **Index** control. A menu opens that enables you to type a new value using the numerical keypad or to invert the encoder polarity (see Figure 2-59 on page 107).

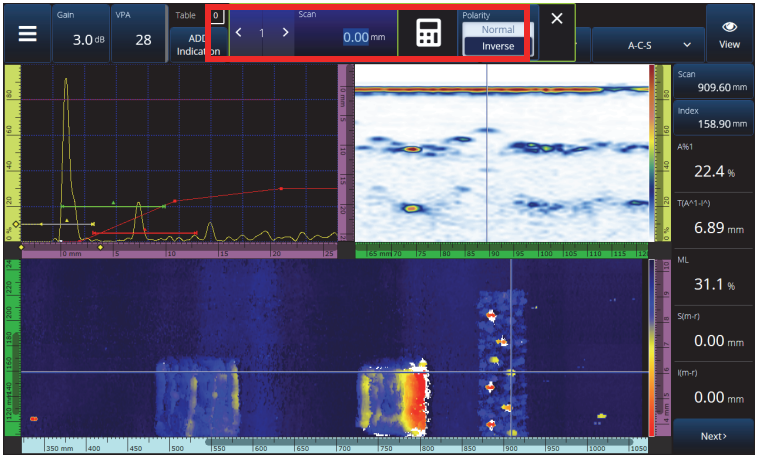


Figure 2-59 Jumping to a specific location by typing a number with the numerical keypad

If you frequently need to reset the encoder positions, consider configuring the **Scan on Play** and **Index on Play** parameters to reset the encoders at the start of each acquisition, eliminating the need to edit them every time. See Table 23 on page 72 for more details.

2. Performing data analysis.

   Navigate the data by tapping the **Scan** or **Index** parameter, and then turning the OmniScan X3 knob to scroll the data cursor.

2.10 Changing the Color Palettes

You can change the color palettes for the amplitude (B-scan, C-scan, or S-scan for PA/UT or End, Side, or Top view for TFM) or thickness C-scan.

To change a color palette

- Tap and hold the color palette ruler/scale (shown on the right side of Figure 2-57 on page 104), and then select **Load**. Review the available color palettes, and tap **Open** to change the palette (see Figure 2-60 on page 108).

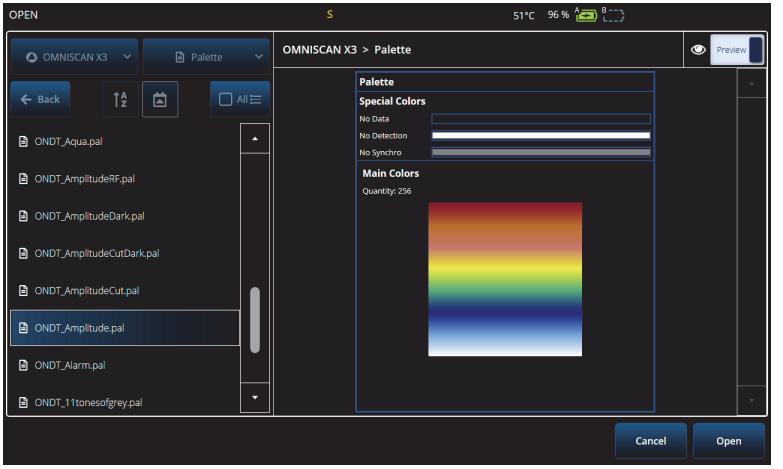


Figure 2-60 Color palette selector

To change the color palette bounds

- By tapping once on the amplitude/thickness ruler, it is possible to effectively zoom on the color palette. Tapping to the bottom of the ruler opens a pop-up indicating **Start**, which allows to change the start of the color palette. Everything below the Start value is the same color. Tapping the top of the color palette ruler opens a pop-up indicating **Range**, which allows to change the range of the color palette.

To restore the default palette

- Long press the palette ruler/scale (shown on the right side of Figure 2-57 on page 104) and select **Restore Default Palette** (see Figure 2-61 on page 108).

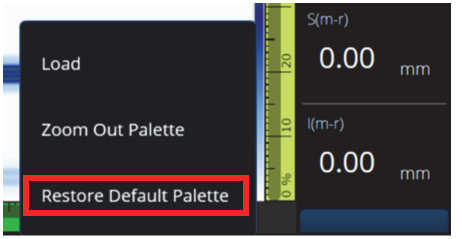


Figure 2-61 Restore Default Palette

2.11 Files

Tap the 🗁 **File menu** to load a setup file (inspection mode) or data file (analysis mode), to preview a report, or to manage other options (see Figure 2-62 on page 109 and Table 48 on page 109).

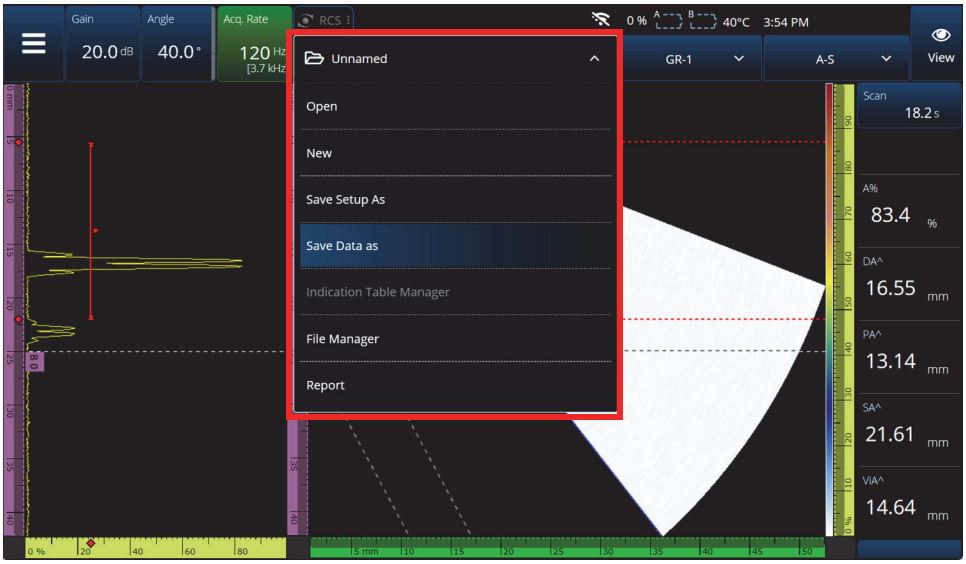


Figure 2-62 File menu

Table 48 File menu options

Option	Description
Open	Tap to open a setup file for acquisition or a data file for analysis.
New	Used to create a new file by loading the default setup.
Save Setup As	Used to save the current setup with a different name.

Table 48 File menu options *(continued)*

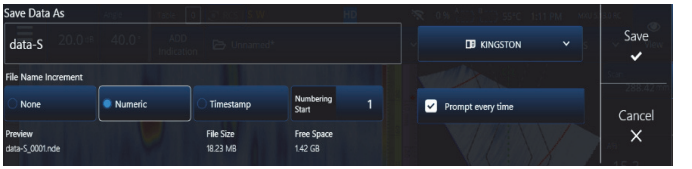
Option	Description
Save Data As	Opens the data saving prompt.  **File Name**: Enter the base file name of the data file. Select either OmniScan X3 or an external drive, USB or SD card as the save location. If **File Name Increment** is set to **None**, this will be the final file name. If **File Name Increment** is selected, the base file name is a prefix, and the final name will have a suffix depending on your increment selection. **Numeric**: Add a number in the format _#### at the end of the base file name. **Timestamp**: Add the current time in the format yyyy_mm_dd ##h##m##s at the end of the base file name. **Numbering Start**: Choose the first number used for file name increment. **Prompt every time**: If this box is checked (default), every time you press the Save key ( ), the **Save Data As** prompt is displayed. If a **File Name Increment** (numeric or time) is selected, you can uncheck this box to auto-increment the base file name every time the Save key ( ) is pressed. This prompt will not appear until you select **Save Data As** again. **File Size** and **Free Space** are displayed for the selected drive.
Indication Table Manager	Used to set up the **Indication Table** during your live analysis.
File Manager	Used to manage files by deleting, renaming, or transferring them.

Table 48 File menu options *(continued)*

Option	Description
Report	Used to create a report from the **Indication Table Manager**.

2.12 Readings

All ten readings displayed on the right-hand side of the screen are included in a generated report and saved in a setup file. You can easily switch which UT parameters are displayed in the readings, either individually or as a list. A description for each parameter is provided in the readings **Select** menu when it is highlighted (see Figure 2-63 on page 111).

To select the reading list to be displayed

1. Tap and hold any of the readings to open a contextual menu.
2. Choose between **Select Reading List** (changes all the displayed parameters from a predefined list) or **Select Reading** (edits one reading at a time):
   a) With **Select Reading List**, you can choose from a list of preconfigured readings (see Figure 2-63 on page 111).

Figure 2-63 Selecting the Reading List

NOTE

The **Select Reading List** option sets all ten readings at once so they are optimized for applications such as **PA+TOFD**, **TOFD**, **Manual Weld** and **Automated Weld**.

b) With **Select Reading**, you can replace one specific reading with any of the available readings (see Figure 2-64 on page 112).

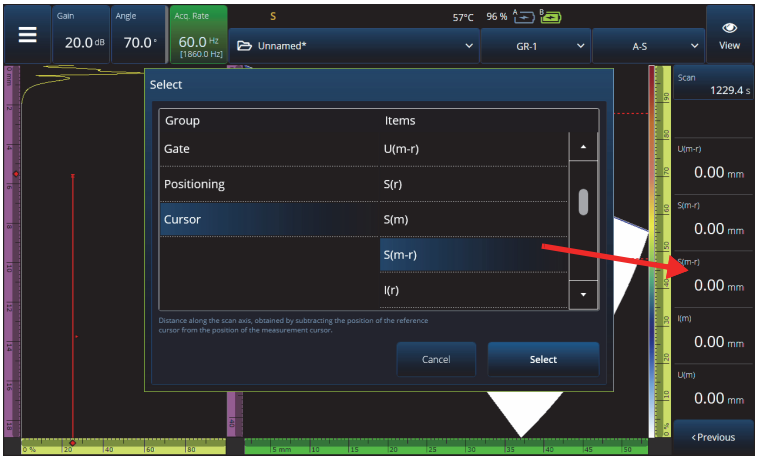


Figure 2-64 Reading selection example

NOTE

The **Select Reading** option is used to change the reading you have selected. You can change it for different groups, such as **Gate**, **Positioning,** and **Cursor**.

2.12.1 Gate Category Reading

Table 49 on page 113 shows the **Gate** category reading codes and their descriptions.

Table 49 Gate reading code descriptions

Category	Description
A%	Peak amplitude of the signal detected in gate **A**. The measured peak depends of the **Peak** settings: **Max Peak** or **First Peak**.
B%	Peak amplitude of the signal detected in gate **B**. The measured peak depends of the **Peak** settings: **Max Peak** or **First Peak**.
I%	Peak amplitude of the signal detected in gate **I**. The measured peak depends of the **Peak** settings: **Max Peak** or **First Peak**.
A^ or **(A/)**	Position of the signal peak value in gate **A** (or crossing point in gate **A**). The measurement taken depends on the gate mode that is selected.
B^ or **(B/)**	Position of the signal peak value in gate **B** (or crossing point in gate **B**). The measurement taken depends on the gate mode that is selected.
I^ or **(I/)**	Position of the signal peak value in gate **I** (or crossing point in gate **I**). The measurement taken depends on the gate mode that is selected.
AdBr	Difference between the current amplitude in gate **A** and the reference amplitude (in dB).
A%r	Difference between the current amplitude in gate **A** and the reference amplitude (in %).
AdBA	Difference between the current amplitude in gate **A** and the current threshold of gate **A** (in dB).

2.12.2 Positioning Category Reading

Table 50 on page 114 shows the **Positioning** category reading codes and their descriptions. If the gate **Measure** option is set to **Edge (/)**, then the description refers to the crossing point in the gate rather than the peak.

Table 50 Positioning reading code descriptions

Category	Description
PA^	Distance on the part surface between the wedge (or probe) front face and the indication detected in gate **A**.
PB^	Distance on the part surface between the wedge (or probe) front face and the indication detected in gate **B** (see the **PA^** definition).
DA^	Depth in the part of the reflector producing the indication detected in gate **A**.
DB^	Depth in the part of the reflector producing the indication detected in gate **B**.
SA^	Sound path from the part entry point to the indication detected in gate **A**.
SB^	Sound path from the part entry point to the indication detected in gate **B**.
VsA^	Volumetric position of the indication detected in gate **A** in relation to the scan axis.
VsB^	Volumetric position of the indication detected in gate **B** in relation to the scan axis.
ViA^	Volumetric position of the indication detected in gate **A** on the index axis.
ViB^	Volumetric position of the indication detected in gate **B** on the index axis.

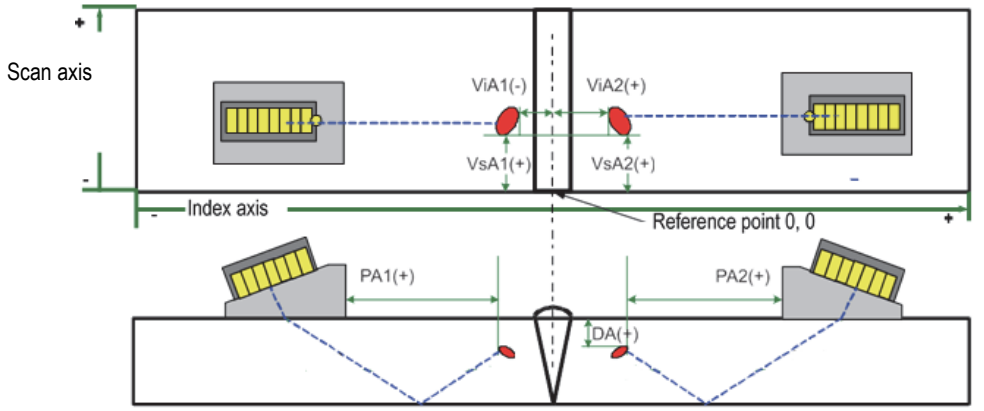


Figure 2-65 Diagram of the PA, DA, ViA, and VsA readings

2.12.3 Cursor Category Readings

Table 51 on page 115 shows the **Cursor** category reading codes and their descriptions.

Table 51 Cursor reading code descriptions

Category	Description
%(r)	Amplitude value at the reference-cursor position.
%(m)	Amplitude value at the measurement-cursor position.
%(m-r)	Amplitude value obtained by subtracting the amplitude of the reference cursor from the amplitude of the measurement cursor.
U(r)	Position of the reference cursor on the ultrasound axis.
U(m)	Position of the measurement cursor on the ultrasound axis.
U(m-r)	Distance along the ultrasound axis, 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.

Table 51 Cursor reading code descriptions *(continued)*

Category	Description
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.
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.
I•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.
%(U(r))	Amplitude of the signal at the position of the reference cursor on the ultrasound axis. For TOFD group only.

Table 51 Cursor reading code descriptions *(continued)*

Category	Description
%(U(m))	Amplitude of the signal at the position of the measurement cursor on the ultrasound axis. For TOFD group only.

2.12.4 Corrosion

Table 52 on page 117 shows the **Corrosion** category reading codes and their descriptions.

Table 52 Corrosion reading code descriptions

Category	Description
T(x)	T is a dynamic reading used to measure thickness. Thickness can be measured using one gate or subtracting two gate values, so x will change according to the **Thickness Mode** selected.
ML	Material loss, expressed in percentage (%), is the part thickness minus the value in the T reading divided by the part thickness.
Tmin	The thinnest reading recorded during the current acquisition.
S(TminZ)	The scan axis position of the Tmin reading.
I(Tmin)	The index axis position of the Tmin reading.
Angle(Tmin)	The relative focal law or virtual probe aperture (VPA) of the Tmin reading.
TminZ	The thinnest reading recorded within the zone created by the reference and measurement cursors of the thickness C-scan display.
S(Tmin)	The scan axis position of the TminZ reading.
I(TminZ)	The index axis position of the TminZ reading.
Angle(TminZ)	The relative focal law or virtual probe aperture (VPA) of the TminZ reading.

2.12.5 Immersion

Table 53 on page 118 shows the **Immersion** category reading codes and their descriptions.

Table 53 Immersion reading code descriptions

Category	Description
I/	Position of the signal when it crosses the gate **I**. The measurement taken depends on the gate mode.
I(w)/	Position of the signal when it crosses the gate **I**, using the velocity of water.

2.12.6 Sizing

Table 54 on page 118 shows the **Sizing** category reading codes and their descriptions.

Table 54 Sizing reading code descriptions

Category	Description
A%Curve	Difference, in percentage, between the peak amplitude in gate A and the corresponding amplitude of the selected sizing curve.
AdbCurve	Difference, in dB, between the peak amplitude in gate A and the corresponding amplitude of the selected sizing curve.
B%Curve	Difference, in percentage, between the peak amplitude in gate B and the corresponding amplitude of the selected sizing curve.
BdbCurve	Difference, in dB, between the peak amplitude in gate B and the corresponding amplitude of the selected sizing curve.
ERS	Equivalent reflector size used with the DGS.
Hardness Depth	Measured within the area of the gate in the active End View. It indicates the depth at which the difference between the upper and lower sections in the gate is maximal. Available only in PCI and OmniScan X3 64.

Table 54 Sizing reading code descriptions *(continued)*

Category	Description
AdBCurveG	Difference, in dB, between the peak amplitude of the signal detected in gate A, and the corresponding amplitude of the selected sizing curve. The reading will react to changes in Gain Offset.
A%CurveG	Difference, in percentage, between the peak amplitude of the signal detected in gate A, and the corresponding amplitude of the selected sizing curve. The reading will react to changes in Gain Offset.
BdBCurveG	Difference, in dB, between the peak amplitude of the signal detected in gate B, and the corresponding amplitude of the selected sizing curve. The reading will react to changes in Gain Offset.
B%CurveG	Difference, in percentage, between the peak amplitude of the signal detected in gate B, and the corresponding amplitude of the selected sizing curve. The reading will react to changes in Gain Offset

2.12.7 Generic Reading Codes

Table 55 on page 119 shows the **Generic** reading codes that appear when abnormal conditions occur and no value can be displayed.

Table 55 Generic reading code descriptions

Category	Description
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.

2.13 Rulers/Scales

Rulers/scales on the vertical or horizontal sides of the data views are associated with various axes. Figure 2-66 on page 120 provides an example of multiple views with rulers/scales.

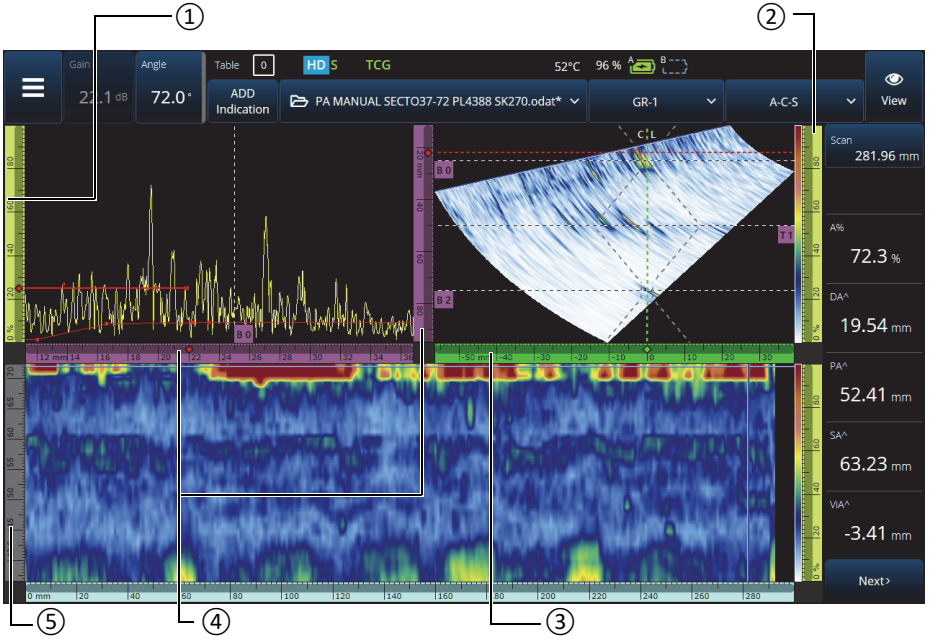


Figure 2-66 Example of multiple views with various rulers/scales

Table 56 Rulers/scales multiple views

Item number	Description
1	Amplitude axis
2	Color palette ruler/scale
3	Index axis

Table 56 Rulers/scales multiple views *(continued)*

Item number	Description
4	Ultrasound axis
5	Angle axis

Each ruler/scale is filled with a dedicated color to help you identify the axis on the various views. Figure 2-67 on page 121 provides examples of rulers/scales with 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.

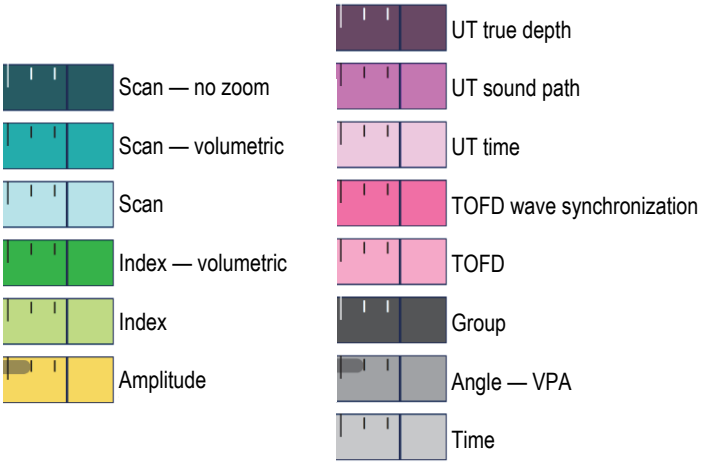


Figure 2-67 Examples of rulers/scales

2.14 Operation Modes

The OmniScan X3 flaw detector features two modes: inspection mode and analysis mode. Figure 2-68 on page 122 illustrates the basic operations of each mode and switching between modes.

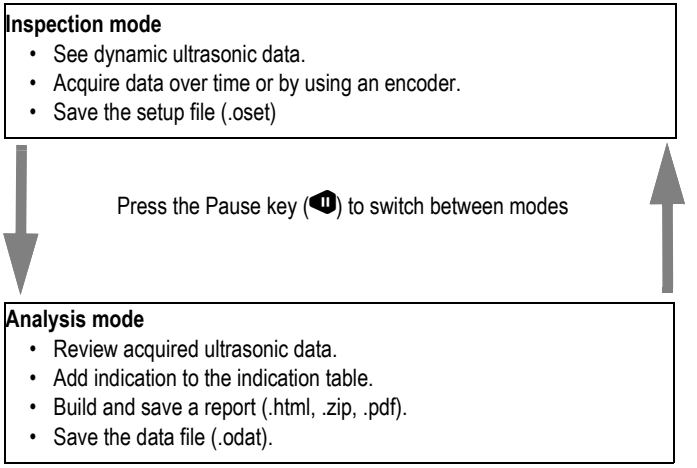


Figure 2-68 Inspection and analysis mode functions

2.14.1 Inspection Mode

Inspection mode is the default acquisition mode when the OmniScan X3 flaw detector is turned on. Inspection mode has the following characteristics:

- The instrument continuously produces ultrasonic beams and dynamically displays ultrasonic data.
- Pressing the Play key ( ) starts data recording for a scanned area (using an encoder) or for a predetermined period of time.
- Pressing the Pause key ( ) pauses the data acquisition and activates analysis mode.

2.14.2 Analysis Mode

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

- The instrument stops acquiring data, and the recorded data is available for analysis.
- The acquisition indicator illuminates solid orange.

2.15 Outline Colors on Parameter Buttons

In certain submenus, some or all of the parameter buttons are outlined in colors that indicate the interface element to which the parameter applies.

There are three colors, each referring to specific gate:

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

2.16 Compression (TOFD Only)

The Compression feature is available (see Figure 2-69 on page 124) to support corrosion mapping and composite inspection applications.

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” icon (

) is displayed on the status indicator.

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.

Figure 2-69 Compression example

2.17 High Definition (PA-UT Only)

The High Definition “HD” icon (HD) (see Figure 2-70 on page 125) feature indicates that each point of data is represented by at least one pixel. A larger scan area may contain too many data points to be represented by one pixel, so a compression will be applied (keeping the maximum amplitude) and the HD icon will not be displayed.

This icon may appear by zooming on a section. If the HD icon appears, it means that all the data points are represented in the view and are not compressed.

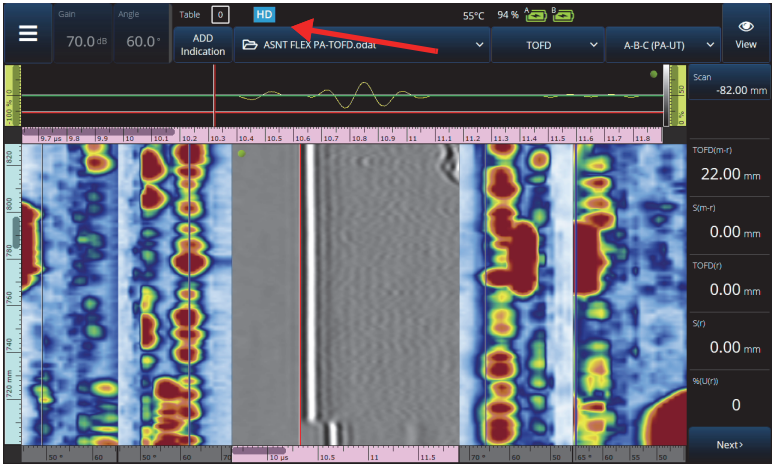


Figure 2-70 High definition example (PA-UT only)

2.18 Shortcuts

Some operations that are done frequently have a shortcut available directly in the views. To access the shortcut, tap and hold (right-click) on the screen to view the list of shortcuts.

Table 57 Shortcuts

View	Shortcut name	Description
All	**Set Reference Cursor**	Positions the cursor where you tapped. This is a shortcut for **Measurements** > **Cursors**.
	Set Measurement Cursor	Positions the cursor where you tapped. This is a shortcut for **Measurements** > **Cursors**.

Table 57 Shortcuts *(continued)*

View	Shortcut name	Description
A-scan	**Enable/Disable Envelope**	Turns ON or OFF the A-scan envelope, which keeps track of the maximum amplitude recorded at each position in the A-scan.
	Clear Envelope	Only available when the envelope is ON. This resets the envelope.
	Enable/Disable A-scan Synchro	Available when the group type is **0° with overlap**. Turns ON or OFF the A-scan synchronization on gate **I**.
S-scan	**Index Offset**	Modifies the **Index Offset** directly without going to **Probe & Part > Position**.
	Skew Left (90°)	Flips the probe orientation.
	Skew Right (270°)	Flips the probe orientation.
	Set Data Cursor	Selects the focal law where you tapped.
C-scan	**A%, B%, I%, I/**	Depending on which gates are active, these shortcuts may or may not be available. Changes the data source of the C-scan.
	Scan Offset	Modifies the **Scan Offset** directly without going to **Probe & Part > Position**.
	Set Data Cursor	Selects the focal law where you tapped. This is a shortcut for **Measurements > Cursors**.

Table 57 Shortcuts *(continued)*

View	Shortcut name	Description
B-scan	**Activate Merged B-Scan**	Changes the B-scan display to a merged B-scan.
	Scan Offset	Modifies the **Scan Offset** directly without going to **Probe & Part > Position.**
	Set Data Cursor	Selects the focal law where you tapped. This is a shortcut for **Measurements > Cursors**.
Top or side view	**Scan Offset**	Modifies the **Scan Offset** directly without going to **Probe & Part > Position**.
Any ruler	**Zoom Out**	Resets the zoom.

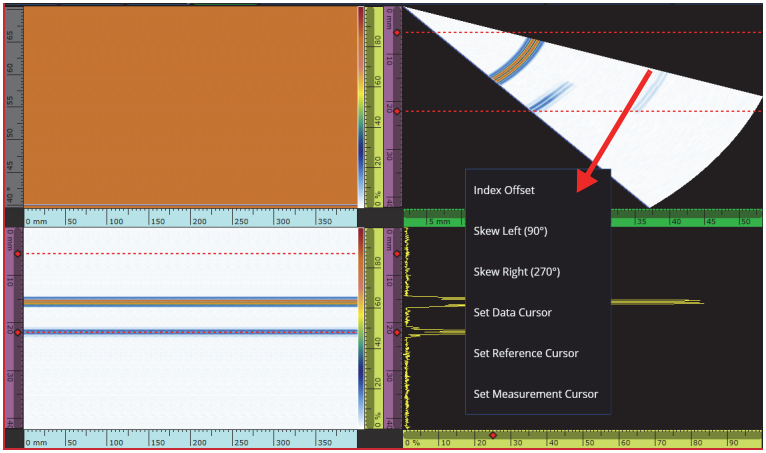


Figure 2-71 Shortcuts menu example

2.19 Export — OmniPC Software

In OmniPC software, there is an additional shortcut available. Right-clicking in a C-scan or a B-scan displays the option **Export C-scan** (when in a C-scan) or **Export All A-scans** (when in a B-scan) option. When you click **Export**, a .txt file is created in C:\Users\%USERNAME%\Documents\OlympusNDT\OmniPC\Export.

The exported file has the same name as the data file used for the export, with the current timestamp added. The data is structured as indicated in Table 58 on page 128.

Table 58 Exported file data structure

Data File = Data file name
Inspection Date = Date when the file was saved
Group = Name of the exported group
Focal Law = Law configuration (sectorial, linear, etc.)
Type = For an exported B-scan, this line says A-scan. For an exported C-scan, the line says C-scan and the type of C-scan (A Amplitude, B Amplitude, Thickness, etc.)
ScanStart = First position in the scan axis
Scan Qty = Number of scan positions
Scan Resol. = Distance between each scan line
IndexStart = First Angle/VPA
Index Qty. = Number of index positions
Index Resol. = Distance between each index position
USound Start = UT Start
USound Qty. = Point Quantity
USound Resol. = Distance between each A-scan point
Ampl. Min. (%) = 0
Ampl. Max. (%) = 800 or 200
Ampl. Resol (%) = Data resolution in the amplitude axis

Table 58 Exported file data structure *(continued)*

Gate Start (mm) = For a C-scan, beginning of the gate
Gate Length (mm) = For a C-scan, width of the gate
Gate Level (%) = Gate Threshold
Bit Depth = 16
Thickness Resol. (mm) = Resolution for a thickness C-scan
Min Thickness (mm) = Minimum thickness of the scale
Max Thickness (mm) = Maximum thickness of the scale
Data Table

For an exported B-scan, the data is structured (see Table 59 on page 129) as follows:

Table 59 B-scan Export

Position (not shown in file)	Data in the file			
Scan 0, Index 0	A-scan point 1	A-scan point 2	...	A-scan point last
Scan 1, Index 0	A-scan point 1	A-scan point 2	...	A-scan point last
Scan..., Index 0	A-scan point 1	A-scan point 2	...	A-scan point last
Scan last, Index 0	A-scan point 1	A-scan point 2	...	A-scan point last
Scan 0, Index 1	A-scan point 1	A-scan point 2	...	A-scan point last
Scan 1, Index 1	A-scan point 1	A-scan point 2	...	A-scan point last
Scan..., Index 1	A-scan point 1	A-scan point 2	...	A-scan point last
Scan Last, Index 1	A-scan point 1	A-scan point 2	...	A-scan point last
Scan 0, Index 2	A-scan point 1	A-scan point 2	...	A-scan point last

For an exported C-scan, the data is structured (see Table 60 on page 130) as follows:

Table 60 C-scan Export

Units	Scan 0	Scan 1	Scan 2	... Scan End
Index End	C-scan data	C-scan data	C-scan data	C-scan data
...	C-scan data	C-scan data	C-scan data	C-scan data
Index 2	C-scan data	C-scan data	C-scan data	C-scan data
Index 1	C-scan data	C-scan data	C-scan data	C-scan data
Index 0	C-scan data	C-scan data	C-scan data	C-scan data

3. Scan Plan

The **Scan Plan** wizard is used to create or modify the required parameters for inspecting a part.

Select ☰ **Main menu**, > **Plan & Calibrate > Scan Plan** to create a complete setup for your application (see Figure 3-1 on page 132). The **Scan Plan** contains these main tabs:

- **1 PART & WELD**
- **2 PROBES & WEDGES**
- **3 GROUPS**
- **4 SCANNING**

After setting parameters on the first tab, and going through the numbered substeps, tap the second tab to continue the **Scan Plan** wizard (see Figure 3-1 on page 132).

TIP

You can exit the **Scan Plan** wizard menu at any time by tapping **Done** at the top right of the screen.

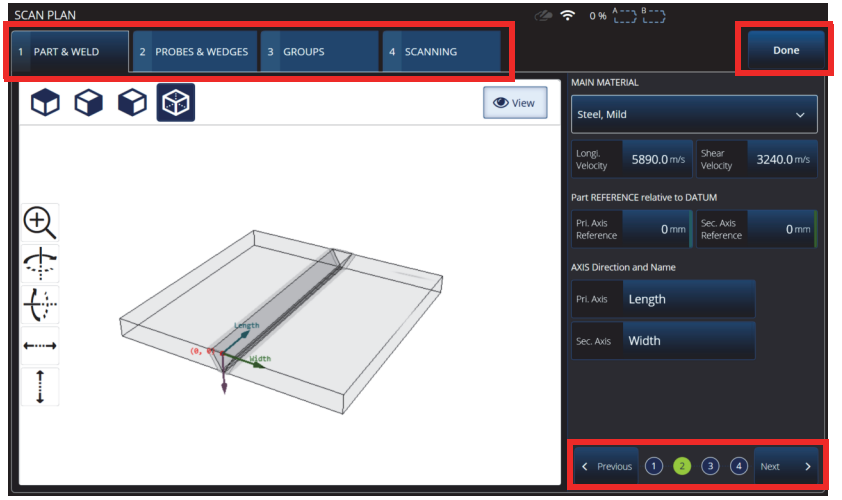


Figure 3-1 Scan Plan tabs, numbered substeps

3.1 Part & Weld Tab

Use the **PART & WELD** tab to define the material, geometry, and weld for the part. Depending on the selected **Part Category**, up to four substeps are displayed to refine the part definition.

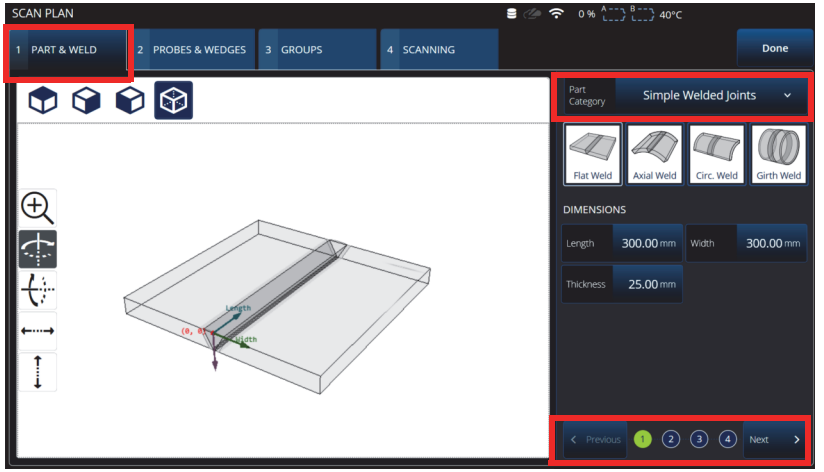


Figure 3-2 Scan Plan > Part & Weld > Substep 1

3.1.1 Part and Weld Substep 1

In substep 1, select the **Part Category** (see Figure 3-2 on page 133).

- **Simple Geometry** (No Weld)
- **Simple Welded Joints**
- **Custom Part**

Table 61 Part & Weld substep 1

Option	Description
Part Category	**Simple Geometry (No Weld):** Choose between **Flat Plate**, **Pipe / Tube**, and **Curved.** **Simple Welded Joints**: Choose between **Flat Weld**, **Axial Weld**, **Circ. Weld**, and **Girth Weld.** **Custom Part**: **Flat Plate**
Dimensions	Sets dimensions according to the **Type** of part selected.

3.1.2 Part and Weld Substep 2

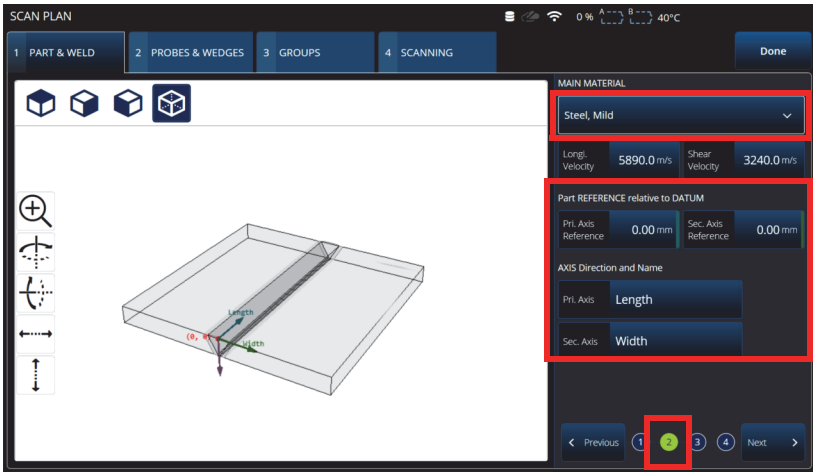


Figure 3-3 Scan Plan > Part & Weld > Substep 2

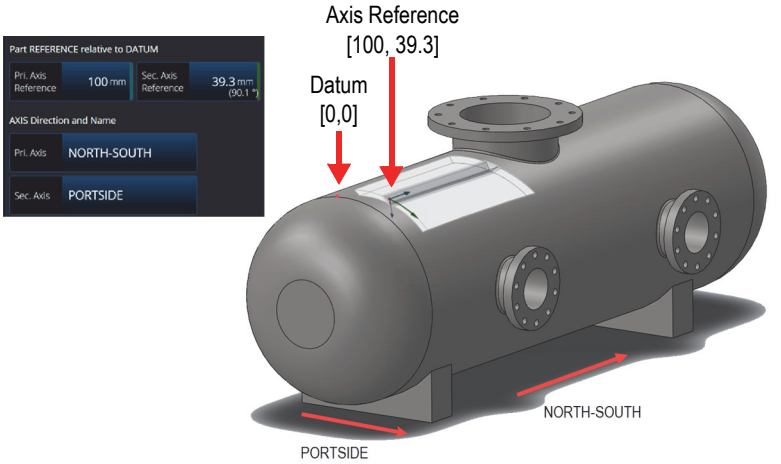


Figure 3-4 Part reference example

In substep 2, select the **MAIN MATERIAL**, define the **Part REFERENCE relative to DATUM**, and define the **AXIS Direction and Name** (see Figure 3-3 on page 134 and Figure 3-4 on page 134).

Table 62 Part & Weld substep 2

Option	Description
Material	**MAIN MATERIAL**: Choose the material of the inspected part from the list (it is set to **Steel, Mild** by default). **Longi. Velocity**: Velocity of the longitudinal wave in the material. This value is automatically set when you choose the **Material**. This value can be edited manually. **Shear Velocity**: Velocity of shear waves in the material. This value is automatically set when you choose the **Material**. This value can be edited manually. **Part REFERENCE Relative to DATUM:** Set the Primary and Secondary **Axis Reference** distance. **AXIS Direction and Name:** Assign different names to the Primary and Secondary axes.

3.1.3 Part and Weld Substep 3

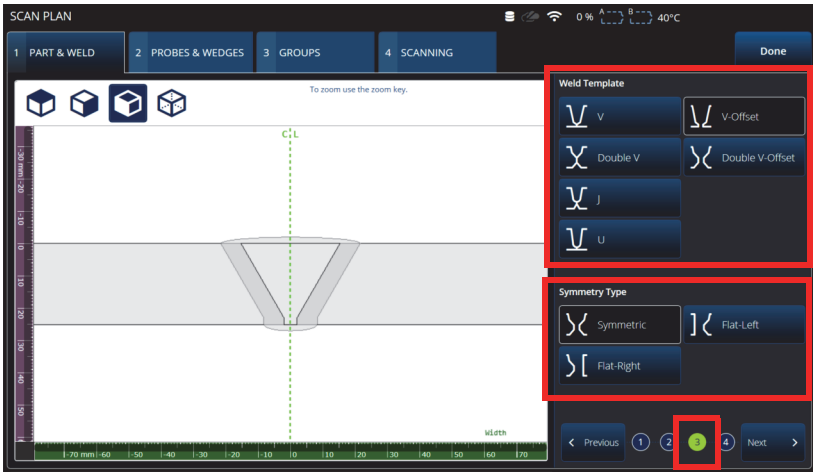


Figure 3-5 Scan Plan > Part & Weld > subtep 3

In substep 3, specify the joint type using the **Weld Template** and **Symmetry Type parameters** (see Figure 3-5 on page 136)**.**

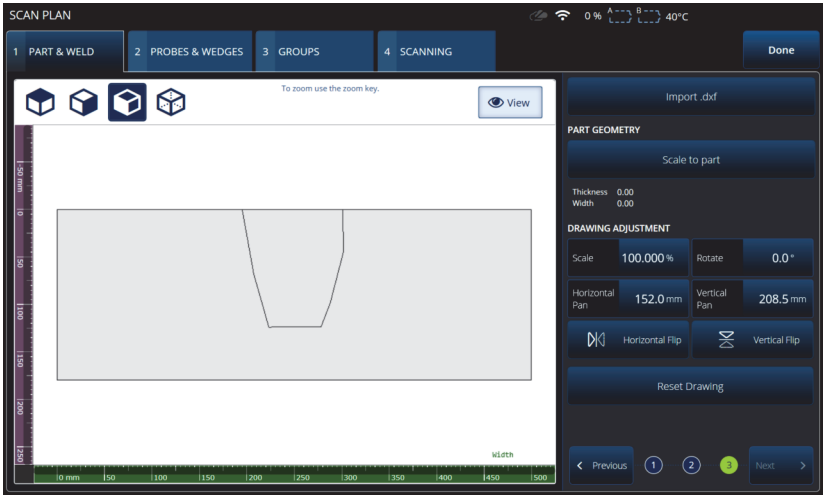


Figure 3-6 Custom part substep 3

Substep 3 for **Custom part** provides various options to adjust a custom overlay on the part (see Figure 3-6 on page 136 and Table 64 on page 138).

Table 63 Part & Weld substep 3

Option	Description
Simple Welded Joints	Select a **Weld Template**: **V**, **V-Offset**, **Double V**, **Double V-Offset**, **J**, or **U**. Select the **Symmetry Type**: **Symmetric**, **Flat-Left**, or **Flat-Right**.
Custom Part	**Import .dxf:** Used to load a .dxf file containing the custom overlay. The file must have already been transferred using the **File Manager**. **PART GEOMETRY:** Use **Scale to part** to constrain to the set part dimensions. **DRAWING ADJUSTMENT:** Used to modify the scale, rotation and position of the drawing. You may also flip the drawing and reset the original dimensions and position.

3.1.4 Part and Weld Substep 4

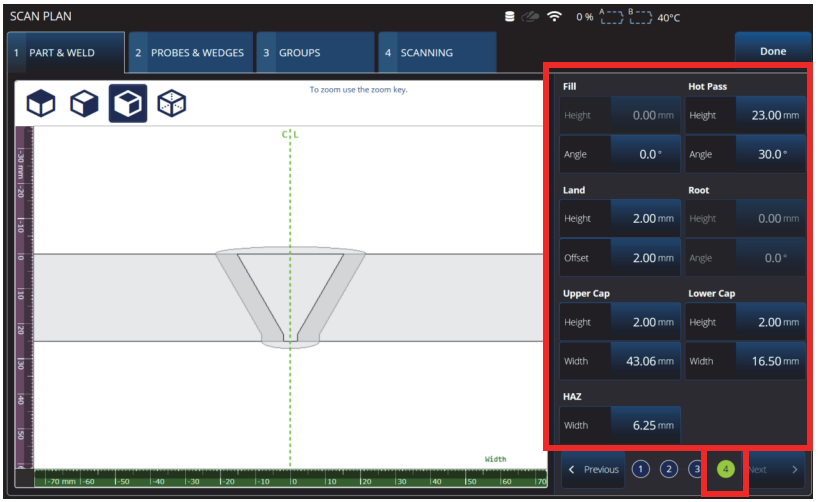


Figure 3-7 Scan Plan > Part & Weld > substep 4

In substep 4, specify additional weld properties (see Figure 3-7 on page 138).

Table 64 Part & Weld substep 4

Option	Description
Weld properties	Set the weld properties: • **Fill** • **Hot Pass** • **Land** • **Root** • **Upper and Lower Cap** • **HAZ**

3.2 Probes & Wedges Tab

Use the **PROBES & WEDGES** tab to define the probes and wedges used for the inspection (see Figure 3-8 on page 139). At the top, set the different physical connections (up to eight). On the right, set the probe and wedge configuration related to the selected group.

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

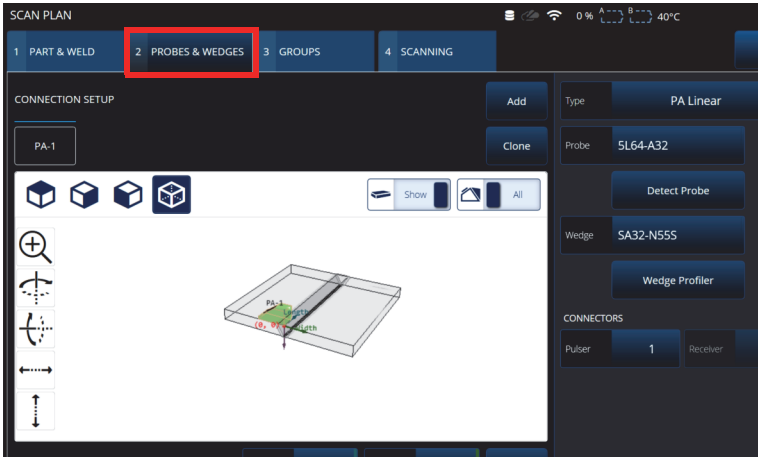


Figure 3-8 Scan Plan > Probes & Wedges

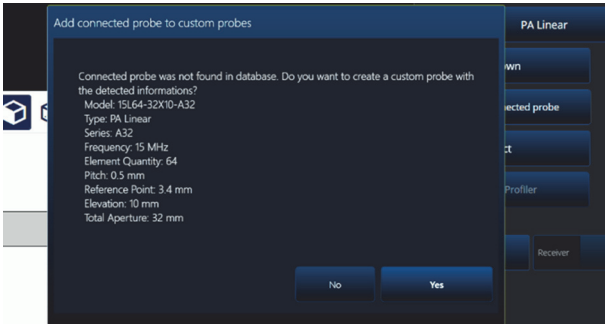


Figure 3-9 Add connected probe dialog

Table 65 Probes & Wedges options

Option	Description
Connection Setup	**Add:** Add a new probe and assign a connector from the following list: • PA: Phased array probe assigned on the PA connector. • UT on PA: UT probe assigned on the PA connector (typically with the use of a splitter). • UT on P1R1 / UT on P2R2: UT probe assigned on the UT connectors. The pairs are independent. **Clone:** Used to create a copy of an existing probe and wedge configuration.
Type	PA: PA Linear, PA Linear Pitch-Catch, PA Dual, Dual Linear 0° UT: TOFD, Pulse-Echo, Dual UT, Pitch-Catch.
Probe	Probe selection based on the **Probe & Wedge** library.
Detect probe	Use this button to detect the probe connected to the instrument. If the probe is not part of the database, it can be added as a custom probe. See Figure 3-9 on page 139.
Wedge	Wedge selection based on the **Probe & Wedge** library.
Wedge Profiler	Use this button to launch the **Wedge Profiler** (see “Wedge Profiler” on page 143).
Pulser	**PA**: Used to choose the first pulser of the probe. For a single probe on the PA connector, the **Pulser** value should be 1. The **Pulser** value should be higher when configuring the second probe on a splitter (according to the splitter wiring). **UT**: Displays the UT connector if selected, or enables you to edit the pulser value if a splitter is used in a **UT on PA** configuration.

Table 65 Probes & Wedges options *(continued)*

Option	Description
Receiver	Displays the receiver that is set according to the probe configuration and the **Pulser** value. The **Pulser** value can be edited only in a **UT on PA** with Dual UT probe configuration.
	Used to set the scan offset of the selected probe.
	Used to set the index offset.
	Used to set the distance between the probes in a Pitch-Catch configuration.
	Used to flip the skew to either 90 or 270.
	Tap to display the top view of the 3D viewer.
	Tap to display the front view of the 3D viewer.
	Tap to display the side view of the 3D viewer.
	Tap to display the perspective view of the 3D viewer.
	Tap to display the wedges or only the one selected.
	Deletes the current probe.

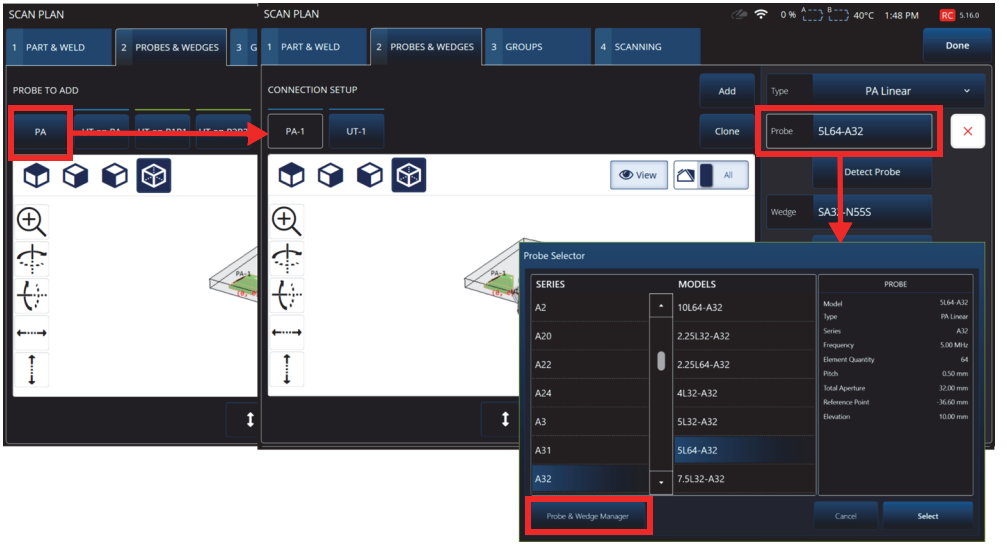


Figure 3-10 Scan Plan > Probes & Wedges > Add — Probe selection example

---

TIP

If your probe or wedge is not available in the predefined list, you can define a new one by clicking the **Probe & Wedge Manager** button (see “Probe & Wedge Manager” on page 197).

For PA probes, use the **Probe & Wedge Manager** to add custom probes or wedges, which is located outside the Scan Plan. For UT probes, you can also use the **Probe & Wedge Manager**, but a quick probe and wedge editor is available if the **Unknown** probe or wedge is selected. A combination of a custom (unknown) probe with a wedge from the list is also possible. When selected from the list, the probe or wedge parameters are fixed, but choosing the **Unknown** option enables you to edit the parameters (see Table 66 on page 143).

Table 66 New probe & wedge options

Option	Description
Frequency	The probe frequency used to calculate the default pulse width and visualization of the dead zone in TOFD.
Diameter	If the selected probe is **Unknown**, the **Diameter** parameter is editable and used primarily for visualization. The probe is assumed to be circular (for a custom-square probe, use **Probe & Wedge Manager**).
Refracted Angle	The refracted angle in the material. Snell's law is used to draw the angle of the wedge.
Wedge Travel	The distance between the probe surface and the beam exit point.
Velocity	Velocity of the wedge material.
Reference Point	See Figure 6-7 on page 200 (UT wedge reference point).

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.

3.2.1 Wedge Profiler

The **Wedge Profiler** is used to empirically validate and adjust the parameters of a wedge. The resulting new parameters are automatically applied to the focal law calculation.

The **Wedge Profiler** becomes available once a valid probe and wedge combination has been selected (see Figure 3-11 on page 144). The wedge profiler can be used for all PA linear probes (FLAT, AOD, and COD) for all available part types.

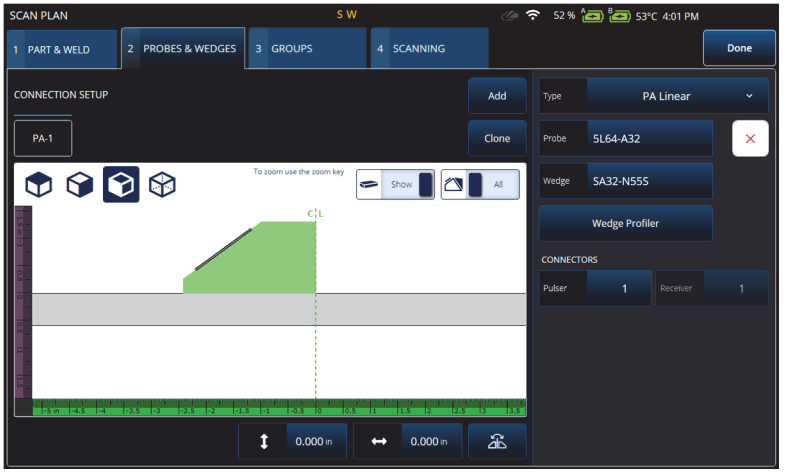


Figure 3-11 Probe and Wedge selection

The **Wedge Profiler** opens a calibration screen that displays the S-scan of a linear group with an element quantity of 1. The resulting display shows the detected wedge interface.

The A gate is active and is used to select the interface signal.

The **Gain** can be changed as needed to adjust the amplitude of the interface response.

The **Measure** button initiates the wedge profile, which recalculates the wedge angle and first element height (See Figure 3-12 on page 145 and Table 67 on page 145).

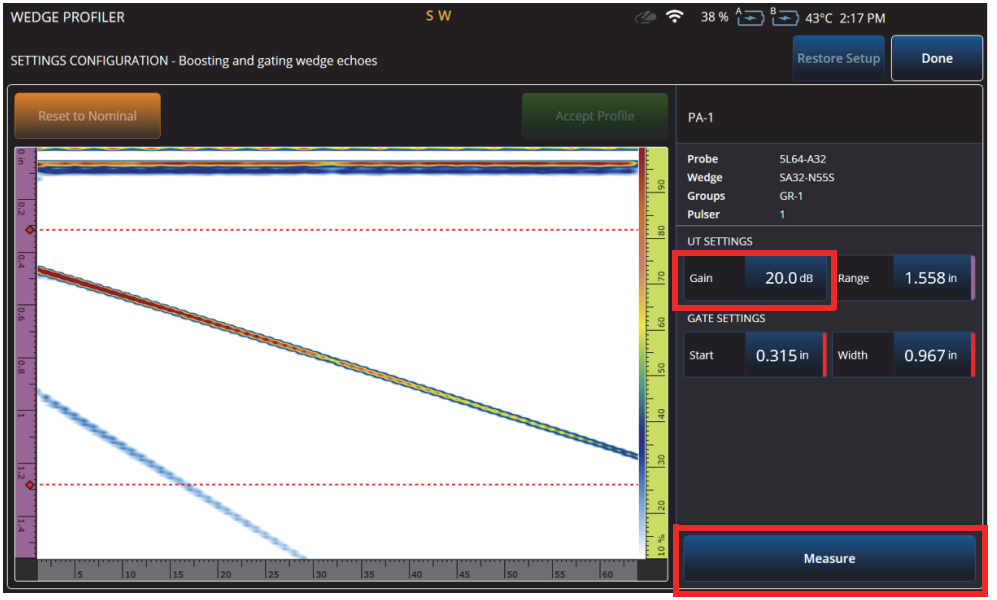


Figure 3-12 Wedge Profiler calibration

Table 67 Wedge Profiler options

Option	Description
Gain	Modify the gain of the signal.
Range	Modify the A-scan range.
Start	Modify the A-gate start.
Width	Modify the A-gate width.
Measure	Measure the wedge dimensions from the signal in the A-gate.

After the wedge parameters have been measured, the signal is redisplayed, but with the beam delays so that the wedge interface is horizontal in the S-scan.

The expected position of the interface is displayed with a dotted green line for visual comparison. You can make manual adjustments to the first element height and wedge angle to correct any remaining deviation.

The new values can be accepted and applied to the setup by pressing **Accept Profile,** or the nominal values can be reset with the **Reset to Nominal** button (see Figure 3-13 on page 146).

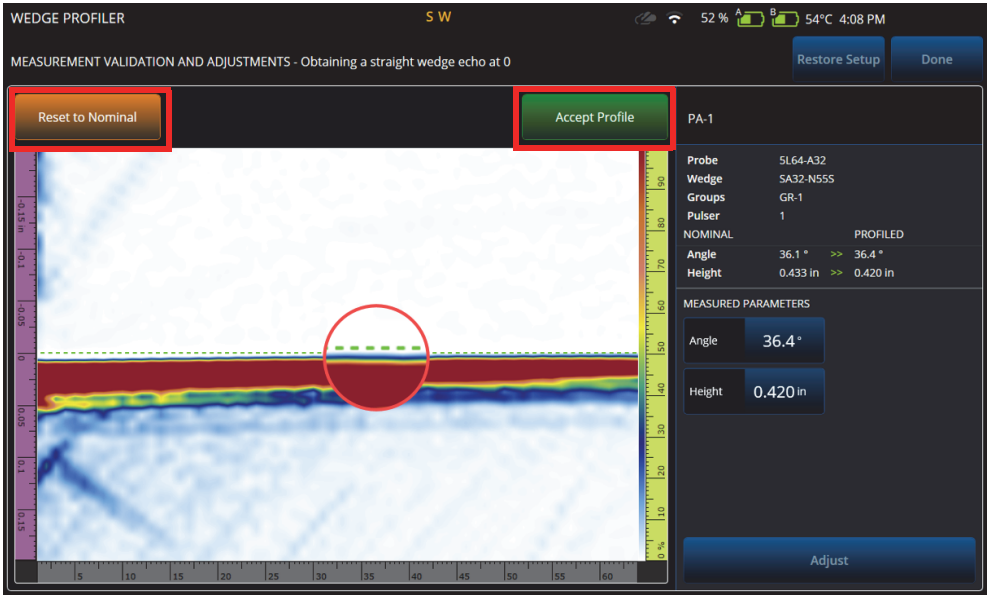


Figure 3-13 Measurement adjustment

Table 68 Wedge Profiler validation options

Option	Description
Reset to Nominal	The original wedge measurements can be restored with this button.
Accept Profile	Accept and replace the nominal values with those that have been measured using the Wedge Profiler.

Table 68 Wedge Profiler validation options *(continued)*

Option	Description
Angle	Displays the measured wedge angle and enables you to manually edit the angle.
Height	Displays the measured first element height and enables you to manually edit the height.
Adjust	Applies the manually edited settings to calculate new delays.
Restore Setup	Reapplies the wedge configuration that was saved in the setup after a measurement has been completed, even if they are different than the nominal values.
Done	Confirms the wedge values and exits the Wedge Profiler.
Diameter (COD only)	Enables you to manually edit the wedge diameter.

NOTE

In the case of a COD wedge, you can also adjust the diameter of the wedge interface manually after the angle and elements' heights have been detected. The same green dotted line can be used to align the wedge interface.

The gap between the nominal and user defined interface cannot be edited directly, but will update when the diameter is changed manually (see Figure 3-14 on page 148).

Figure 3-14 Measurement validation

NOTE

If calibrations have been done beforehand in a new or previously saved setup, the wedge profile can still be confirmed by resetting the nominal values and measuring the wedge.

After the values have been remeasured, you can accept the new values or restore the previous setup.

If the new values are accepted, any previous calibrations will be reset. The reset applies to calibrations done with nominal values or previously saved values.

3.3 Groups Tab

On the **GROUPS** tab, the groups are defined based on the probe configuration defined previously. One group per probe is created by default, which you can edit using the menu on the right side. To create more than one group on a probe, use the **Add** or **Clone** button. A group is a set of beams, or focal laws, defined by the **Law Config**.

Figure 3-15 Scan Plan > Groups

The parameters for each group can have multiple pages, which you can scroll through using the **Previous** and **Next** buttons at the bottom right of the screen (see Figure 3-16 on page 150 and Table 69 on page 150).

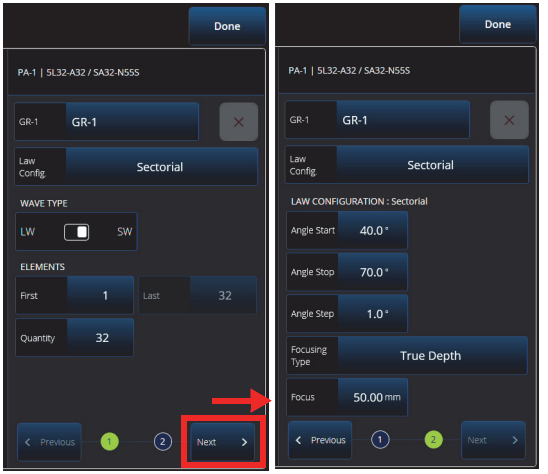


Figure 3-16 Scan Plan > Groups > Law Config. parameters

NOTE

It is not possible to have fewer than one or more than eight groups. The maximum number of focal laws is 1024. Each defined probe must have at least one group assigned. Some configurations, such as **0° with overlap**, allow only one group. Therefore if more than one probe or group is defined, these configurations will not be available.

Table 69 Groups — New Set — Configuration options

Option	Description
GR-1	Used to set the name of the group.

Table 69 Groups — New Set — Configuration options *(continued)*

Option	Description
Law Config. PAUT	**Sectorial:** Provides a multiple-angle scan using the same elements for every angle of the scan. **Linear:** Provides a linear scan at a configurable angle. You can use this mode at a zero-degree angle if you do not need overlapping scans. **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. Use an element quantity smaller than the total number of elements in the probe to obtain the benefits offered by this scan type compared with the **Sectorial** scan. **Coupling Check**: Provides a single beam firing at 0° in the material to validate the coupling. This group has a built-in configuration that sends a signal to the I/O connector if the amplitude in gate **A** drops below the threshold. **Law file**: Loads a custom .law file that configures the PA connector. Supported .law files include versions 5.0, 5.2, and 5.3. **0° with overlap**: Provides a linear scan at 0°. Used primarily for raster scanning with some overlap between each scan line. This group can only be used alone. See Figure 3-17 on page 154.
Law Config. FMC	**TFM**: Provides a TFM scan over the selected area based on a reconstruction of the FMC acquired data. The TFM scan uses all elements of the probe. **PCI**: Uses a similar algorithm to the standard TFM, but instead of summing amplitude elementary A-scans, it sums the phase of those elementary A-scans. See Figure 3-17 on page 154.

Table 69 Groups — New Set — Configuration options *(continued)*

Option	Description
Law Config. PWI	See “Plane Wave Imaging (PWI)” on page 211. **TFM**: Provides a TFM scan over the selected area based on a reconstruction of the PWI acquired data. The TFM scan uses all elements of the probe. **PCI**: Uses a similar algorithm to the standard TFM, but instead of summing amplitude elementary A-scans, it sums the phase of those elementary A-scans. See Figure 3-17 on page 154.
Wave Type	Used to set the toggle between **LW** and **SW**. **LW**: Longitudinal wave **SW**: Shear wave
Elements	**First**: Displays the first element on the probe. **Last**: Displays the last element on the probe. **Quantity**: Used to set the number of elements used in the focal law (the size of the aperture). For a M × N matrix array probe, element quantity can only be a multiple of M, the number of elements in the primary axis. **Step**: Used to set the spacing between the consecutive focal laws (for linear scans and zero-degree law configuration).

Table 69 Groups — New Set — Configuration options *(continued)*

Option	Description
Law Configuration: (Sectorial)	**Angle Start:** Used to set the angle of the first beam in the material. **Angle Stop:** Used to set the angle of the last beam in the material from the wedge. **Angle Step:** Used to set the angular step between each focal law. **Skew Angle:** Used to steer the beam angle (matrix probes only). **Focusing Type:** • **True Depth:** The focus is at the same depth for all beams. • **Half Path:** The focus is at the same sound path for all beams. • **Projection:** The focus is at a set distance from the edge of the probe at the defined angle. • **Unfocused:** The beam is not focused at any point. **Focus depth:** Used to set the focusing depth.
Law Configuration: (TFM)	**Wave Set**: Switch between **Pulse Echo** or **Self Tandem** to display the different wave set options for each mode. Choose the wave set that best fits the inspection. Appropriate wave set selection is crucial for a good TFM inspection. Use the AIM to facilitate wave set selection. See “Groups — View Menu” on page 154 and Figure 3-18 on page 155. **Min/Max Index:** Used to set the limits of the TFM zone in the index axis. **Min/Max Depth**: Used to set the limits of the TFM zone in the depth axis. The **Maximum Depth** is currently limited to the depth of the sample.

Table 69 Groups — New Set — Configuration options *(continued)*

Option	Description
Focusing (TOFD)	**PCS**: Used to set the probe center separation (**PCS**). This is the distance between the exit points of the two probes. **Focus (%)**: Used to set the focus in depth of the beam in percentage (%) of the thickness. **Focus (mm/inch)**: Used to set the depth of the focus of the beam. Focus can be input in percentage or distance; changing one recalculates the other.

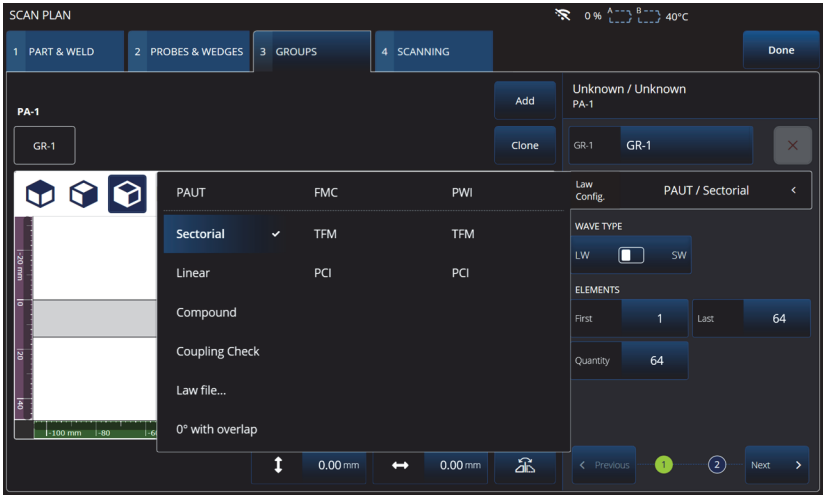


Figure 3-17 Groups—Law Config.

3.3.1 Groups — View Menu

Use these settings to modify the visual representation of the scan plan. The View menu items change depending on the display type.

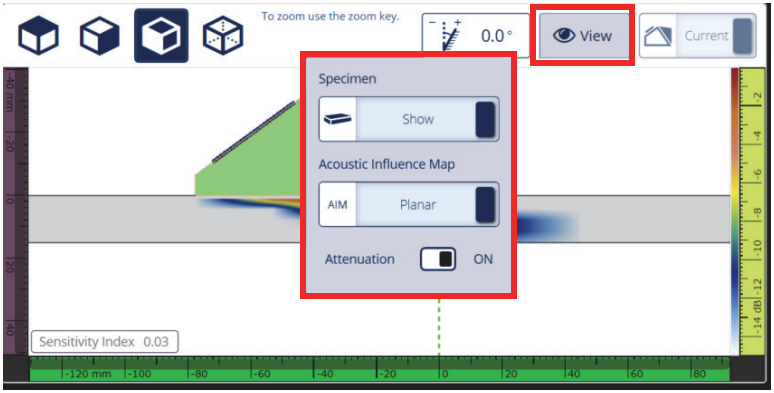


Figure 3-18 Groups —View menu in FMC and PWI displays

Table 70 Groups —View menu in FMC and PWI displays

Option	Description
View — Specimen	You can toggle between **Show** and **Hide**.
View — Acoustic Influence Map	Without impacting the acoustic configuration, you can use this option to select the defect type in the AIM tool (**Spherical** or **Planar**). Selecting the appropriate defect type in the AIM model helps ensure you select the proper **Wave Set**.
View — Attenuation	AIM attenuation can be turned **ON** or **OFF.**

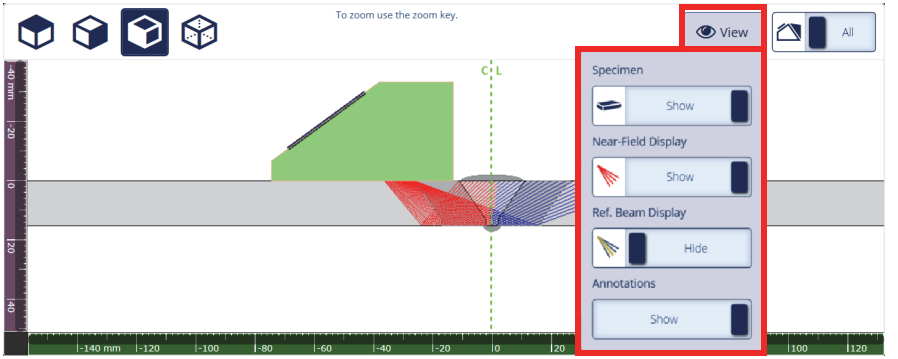


Figure 3-19 Groups —View menu in sectorial display

Table 71 Groups —View menu in sectorial display

Option	Description
View — Specimen	You can toggle between **Show** and **Hide**.
View — Near Field Display	Show or Hide the near field display, which appears in red in the visual representation. See “Near field calculation” on page 156.
View — Reference Beam	Show or Hide the reference beam, which appears as a yellow line.
Annotations	Display or hide the annotations. ◆ Axis Direction and Name ◆ Part Reference relative to Datum ◆ Group Name

3.3.2 Near field calculation

The near field value is calculated using formula (1) on page 157.

Variables are defined in Table 72 on page 157.

To calculate the near field (Nf) value,

$$Nf = h \times A^2 \times f / (4 \times c2) \quad (1)$$

To calculate near field in specimen (Np) value

If Nf - rv ≥ 0, use

$$Np = Nf - rv$$

If Nf - rv < 0, use

$$Np = - (A^2 \times f) / (4 \times c2) \quad (2)$$

NOTE

If the near field value Np is negative, the near field is located in the wedge and is assigned a negative value. In this case, we use formula (2) on page 157.

Table 72 Near field formula variables

Variable	Description	Units
f	Probe frequency	Hz
N	Number of probe elements	-
wedgeAngle	Wedge Angle	rad
θr	Refraction angle	rad
θi	Incidence angle	rad
L	Probe length	m
W	Probe width	m
A	Transducer aperture dimension	m
E	Elevation	m
p	Probe pitch	m
h	Correction coefficient	-
r_w	Wedge sound path length	m

Table 72 Near field formula variables *(continued)*

Variable	Description	Units
r_v	Adjusted wedge sound path length	m
c1	Wedge sound speed	m/s
c2	Probe sound speed	m/s
Nf	Near field value	m
Np	Near field value in specimen	m

The value of the different variables are calculated with the provided equations.

Transducer aperture (A):

$$L = 0.95p \times N$$

$$W = 0.95 \times E$$

Where 0.95 is the apodization value.

If $L \times \cos(\text{wedgeAngle} - \theta i) \geq W$, use

$$A = L \times \cos(\text{wedgeAngle} - \theta i) \times \cos(\theta r) / \cos(\theta i)$$

Else,

$$A = W$$

Correction coefficient (h)

$$h = 0.6546 \times \text{ratio}^3 - 0.3112 \times \text{ratio}^2 + 0.0411 \times \text{ratio} + 0.9987$$

Where:

If A = W

$$\text{ratio} = W/A$$

If A < W

$$\text{ratio} = A/W$$

Wedge sound path length (r_w)

Variable r_w is obtained by measuring the distance between the entry point of the ray into the specimen and the centroid of the central element of the active aperture.

For an even number of elements in the active aperture, calculate the distance between the entry point of the ray in the specimen and the mid point between the centroid of the two central elements of the active aperture.

Adjusted wedge sound path length (r_v)

If the refraction angle $\theta r \neq 0$ rad,

$$r_v = r_w \times \tan(\theta i) / \tan(\theta r)$$

If $\theta r = 0$ rad

$$r_v = r_w \times c1 / c2$$

3.4 Scanning Tab

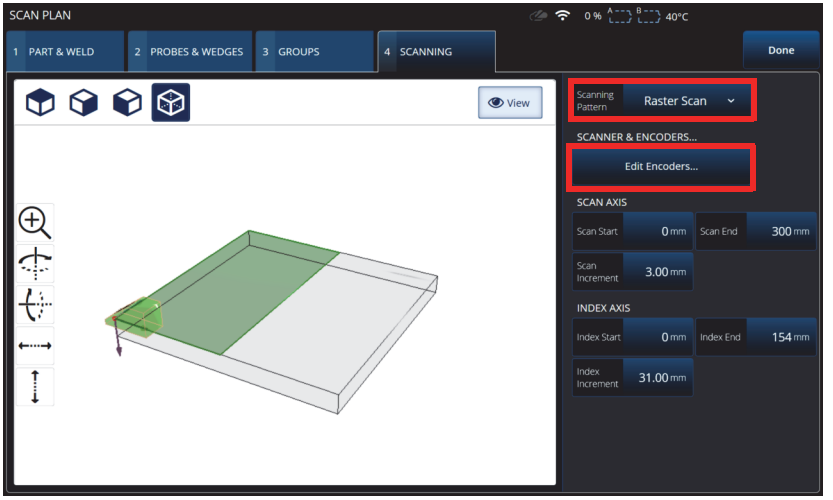


Figure 3-20 Scan Plan > Scanning

On the **SCANNING** tab, you can define the **Scanning Pattern** and Scan Area parameters by changing the **Scan Axis** and **Index Axis** values. You can also select and edit encoder parameters (see Figure 3-20 on page 159).

See Table 73 on page 160 for a description of the options.

Table 73 Scan — Area

Option	Description
Scan Start	Used to set the starting location of the scan (expressed in mm or in.).
Scan End	Used to set the maximum distance you can scan (expressed in mm or in.).
Scan Res.	Used to set the step (resolution) at which the points will be acquired on the scan (expressed in mm or in.).
Index Start	(Raster scan only.) Used to set the starting location of the raster scan in the index axis (expressed in mm or in.).
Index End	(Raster scan only.) Used to set the raster end location in the index axis (expressed in mm or in.).
Index Res./Index Step	(Raster scan only.) Determines the index resolution. Cannot be modified in the **Linear at 0°** scan.

4. Calibration

Depending on your requirements, before starting an inspection, you can perform several calibration procedures using a probe, wedge, and calibration block composed of the same material as the part to be inspected.

To perform a calibration

1. Select ☰ > **Plan & Calibrate > Calibration Tools** to access the **PA/UT/TFM Calibration** wizard (see Figure 4-1 on page 162). See "TOFD Calibration" on page 180 to perform a TOFD calibration. Like the Scan Plan wizard, the Calibration wizard's workflow is separated into several tabs or sections (for different types of calibration).
2. On the **Group** tab (see Figure 4-1 on page 162), select the group you wish to calibrate. For a UT group, also choose the sizing method to calibrate: **TCG**, **DAC**, or **DGS**.
3. Navigate through the other tabs to calibrate the group. On each tab after the **Group** tab, the calibration parameters are on the right and the views are on the left.
4. Set your parameters, then move your probe to adjust the signal according to the calibration type.
5. Subsequently tap either **Get Position** or **Calibrate**. When you are satisfied with your adjustment, tap **Accept Calibration**.
6. You can then either continue to another tab in the **Calibration** wizard, or exit by tapping **Done**.

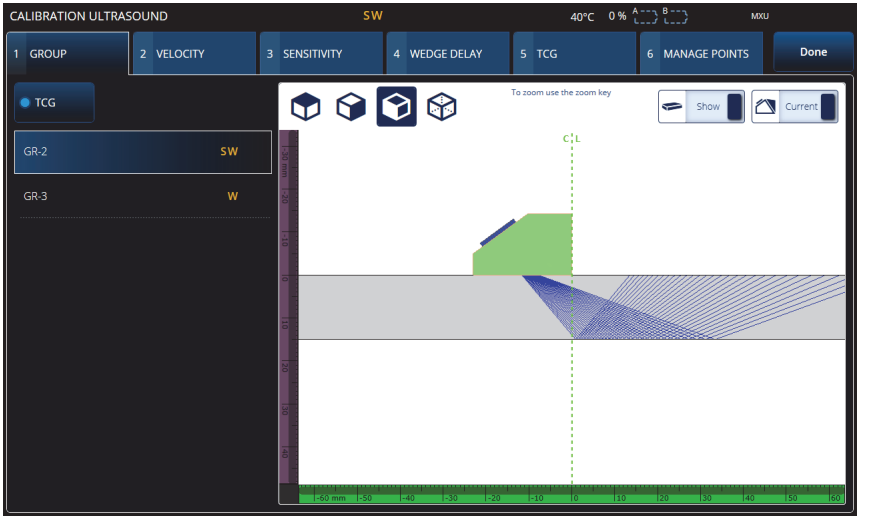


Figure 4-1 Calibration > Group

IMPORTANT

If an ultrasonic velocity calibration and a wedge delay calibration are required, perform the ultrasonic velocity calibration before the wedge delay calibration. The OmniScan X3 flaw detector 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 when you perform the ultrasonic velocity calibration.

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 it was before the calibration).

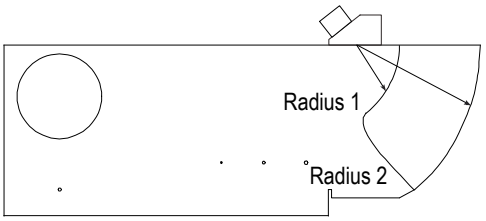
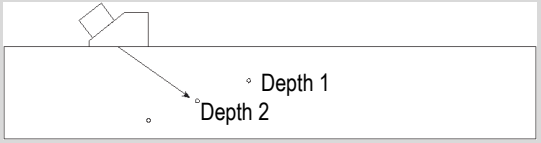
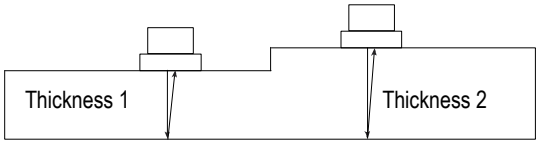
NOTE

The Calibration indicators (icons) turn green after calibration (see Table 5 on page 31).

4.1 Reflector Types

Calibration procedures are performed using calibration blocks with different types of known reflectors. Table 74 on page 163 illustrates the types of probes, wedges, and calibration blocks used for each type of reflector.

Table 74 Reflector, probe, and calibration block types

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

4.2 Ultrasonic Calibration

You can calibrate several ultrasonic aspects using the Calibration wizard.

Velocity

Used to calibrate the velocity of the sound propagation in the material of the inspected part (see Figure 4-2 on page 164). The calibration block must have two known reflectors and be made of the same material as the part to be inspected. The **Velocity** is calibrated along with the wedge delay in one process for UT channels. In a UT group, **Velocity** calibration is performed simultaneously with wedge delay calibration.

To calibrate Velocity:

1. Define the two targets. The maximum distance a target can be set depends on the range. Increase the range if needed to reach a target further.
2. Find your target by manually moving the probe over the calibration block.
3. Maximize the signal in gate A, making sure that the most direct path hits the target.
4. Hold the position and press **Get Position** below gate A.
5. Repeat step 3 and 4 for gate B.
6. If the calibration procedure was successful and the velocity seems correct, then press **Accept**. Otherwise, reset the calibration and repeat steps 1 to 6.

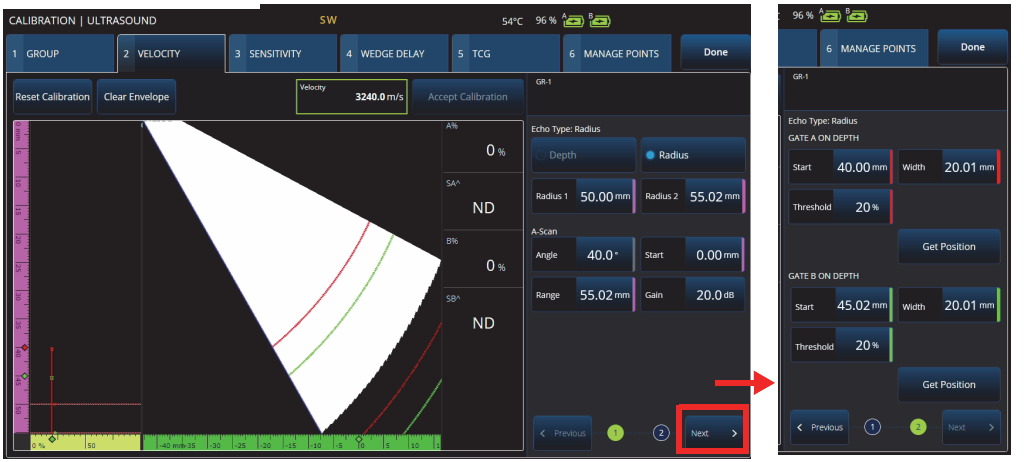


Figure 4-2 Calibration > Velocity

Sensitivity (PA group only)

Used to calibrate the detection sensitivity for a reference reflector (see Figure 4-3 on page 165 and Table 75 on page 165). 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.

To calibrate, simply adjust the calibration parameters (display and gates), then scan a reference reflector. When all focal laws have scanned the reflector, press **Calibrate**. The envelope signal is used to calculate the amount of gain required for each focal law to bring them at the reference amplitude (typically 80 %).

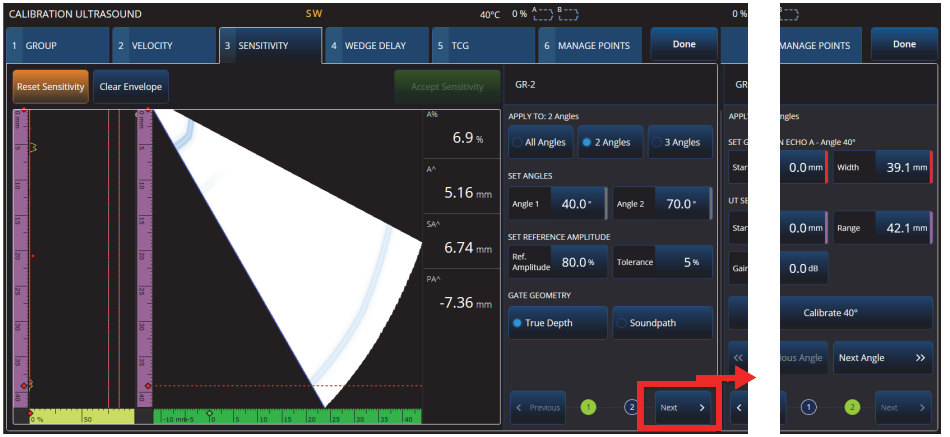


Figure 4-3 Calibration > Sensitivity

Table 75 Sensitivity tab options

Option	Description
Reset Calibration	Resets the sensitivity calibration. The “S” at the top of the screen disappears.
Clear Envelope	Clears the envelope on the bottom view. The green line disappears.

Table 75 Sensitivity tab options *(continued)*

Option	Description
Calibrate	Applies a beam gain for each focal law so that the amplitude over the reference defect is compensated.
Accept Calibration	Accepts and saves the sensitivity calibration. The “S” at the top of the screen turns green.
Apply to	**All Angles/VPA**: The calibration applies to all focal laws of the group. **2 Angles:** The calibration applies to two angles of the sectorial scan. The gain for the other angles will be interpolated from the calibrated values. **3 Angles:** The calibration applies to three angles of the sectorial scan. The gain for the other angles will be interpolated from the calibrated values.
Set Reference Amplitude	**Ref. Amplitude:** The calibration target (80 % by default). **Tolerance:** Displays horizontal white and red dotted lines at **Ref.Amplitude** ± **Tolerance**. Used to check if the calibration is within tolerance.
Gate Geometry	**True Depth:** Sets the gate according to the depth in the material for the sensitivity calibration. **Sound Path:** Sets the gate according to the distance traveled in the material for the sensitivity calibration.
Gate A	**Start:** Used to set where the gate starts in reference to the origin (can be expressed in mm or in.). The origin is either the zero of the ultrasound axis or the crossing point of gate I if the current signal is synchronized on **I/**. **Width:** Used to set the width (length) of the gate.
UT Settings	**Gain:** Used to set the signal gain value for the sensitivity calibration. **Start:** Used to set the start of the displayed A-scans. **Range:** Used to set the range of the displayed A-scans.

Table 75 Sensitivity tab options *(continued)*

Option	Description
Previous **Next**	Used to alternate between the parameters on the first and second pages.

NOTE

You can validate your **Sensitivity** calibration by clearing the envelope, redoing the manipulation, and checking that the amplitude from all focal laws are within tolerance.

PA Wedge Delay

Used to calibrate the delay corresponding to the sound propagation within the wedge (see Figure 4-4 on page 168 and Table 76 on page 169). A wedge delay calibration enables you to identify the face of the wedge contacting the part. This establishes a zero position for the entry surface of the part. The calibration procedure requires a calibration block with one known reflector.

To calibrate the wedge delay

1. Adjust the UT range and gain to see two (2) reflectors.
2. Set the nominal position of the reflector (in **Radius** or **Depth**).
3. If needed, fine-tune the position of the gates to get the signal inside the gate.
4. Move your probe to maximize the signal in gate **A**. The chart at the bottom displays the position of the maximum amplitude peak in the gate for each focal law.
5. After all focal laws have scanned the reflector, press **Calibrate**.
6. Press **Accept** if the results are satisfactory.

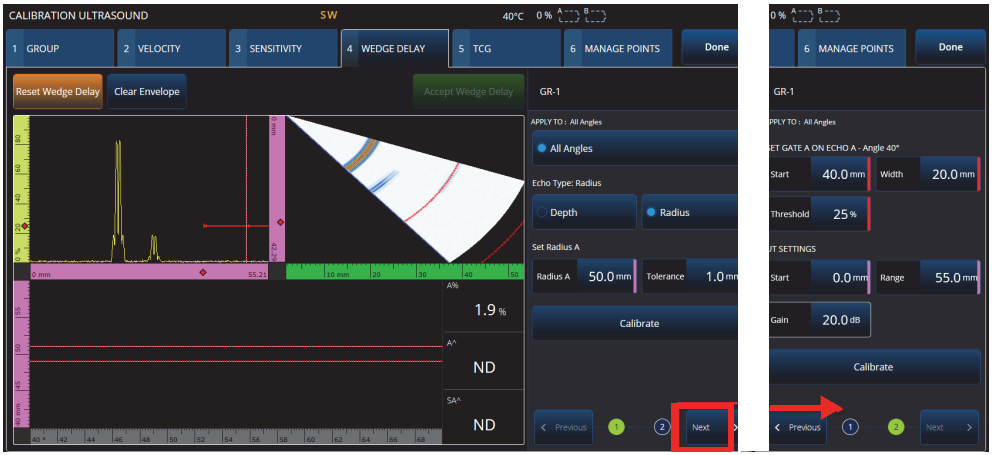


Figure 4-4 Calibration > Wedge Delay

Velocity & WD (**UT** group only)

Used to calibrate (with a single wizard) both the sound propagation in the material of the inspected part and the delay corresponding to the sound propagation within the wedge. It is possible to calibrate a wedge delay only, or a wedge delay and velocity at the same time.

To calibrate the Wedge Delay and Velocity

1. Adjust the UT range and gain to see two (2) reflectors.
2. Set the nominal position of the 2 reflectors (in **Radius** or **Depth**). Only one reflector is required to calibrate the **Wedge Delay** only.
3. Press **Next**.
4. If needed, fine-tune the position of the gates to get both signals inside the gates.
5. Move your probe to maximize the signal in gate **A**.
6. Press **Get Position**. The software records the peak location. Note that the peak is taken on the real signal, not on the envelope.
7. Repeat step 6. on page 168 for the reflector in gate **B**. Ignore this step to calibrate the wedge delay only.
8. Press **Accept** if the results are satisfactory.

Table 76 Wedge tab options

Option	Description
Reset Calibration	Resets the wedge delay calibration. The “W” at the top of the screen disappears.
Clear Envelope	Clears the envelope on the bottom view. The green line disappears.
Calibrate	Calibrates the wedge delay by automatically applying beam delays to each focal law so that the reference is seen at the same distance for all beams.
Accept Calibration	Accepts and saves the wedge delay calibration. The “W” at the top of the screen turns green.
Echo Type	**Depth:** Used to set to depth or thickness reflector types, herein referred to as the reflector. **Radius:** Used to set to radius reflector types, herein referred to as the reflector.
Set	**Depth/Radius A:** Used to set the nominal depth of the reflector. **Tolerance:** Used to set the tolerance. **Depth/Radius 1**: In UT, used to set the nominal distance of the reflector. **Depth/Radius 2**: In UT, used to set the nominal distance of the second reflector to get the **Velocity** and **Wedge Delay** at the same time. Reflector 2 cannot be at the same depth as reflector 1.
Gate A	**Start:** Used to set where the gate can start in reference to the origin (can be expressed in mm or in.). **Width:** Used to set the width of the gate (the bottom points red line of the S-scan and the largest of the continuous red line on the A-scan). **Threshold**: Used to set the height of the gate.

Table 76 Wedge tab options *(continued)*

Option	Description
A-scan	**Gain:** Used to adjust the signal gain value to get a good signal in the gate. **Start:** Used to set the start of the displayed A-scans. **Range:** Used to set the range of the displayed A-scans.
Previous **Next**	Used to alternate between the parameters on the first and second pages.
Done	Select **Done** to apply and close the wedge delay calibration settings.

4.3 TCG/DAC Calibration

The OmniScan X3 flaw detector offers TCG (time-corrected gain). Sizing functions enable you evaluate the size of a reflector anywhere in the part by measuring or compensating for the signal attenuation.

For UT and PA channels, it is possible to create a DAC or TCG. The menus for the DAC (distance-amplitude correction) calibration are very similar to the menus for the TCG calibration.

To build a DAC instead of a TCG in UT and PA, select the DAC option in the Group tab of the calibration wizard (see Figure 4-5 on page 171 and Table 77 on page 171).

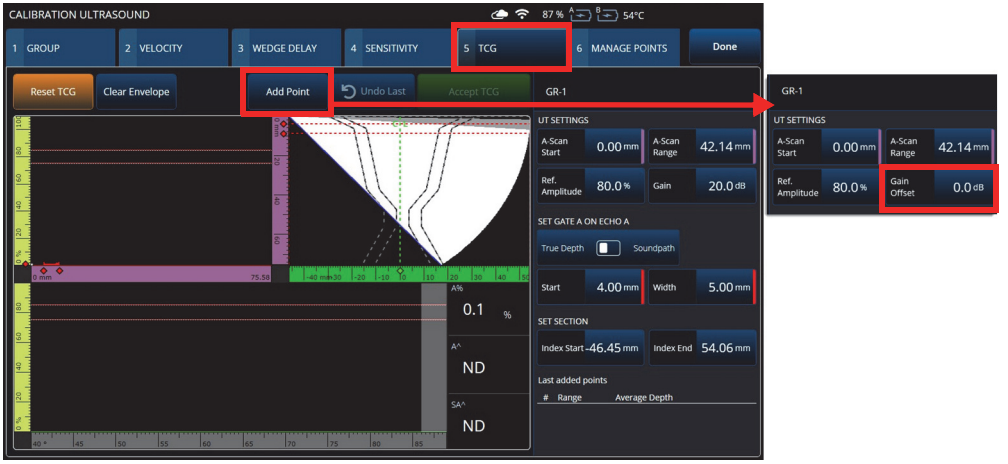


Figure 4-5 Calibration > TCG

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 DAC (distance-amplitude correction).

Table 77 TCG tab options

Option	Description
Reset TCG	Resets the TCG calibration. The TCG at the top of the screen disappears.
Clear Envelope	Clears the envelope on the bottom view. The green line disappears.
Accept Calibration	Accepts and saves the TCG calibration. The **TCG** at the top of the screen turns green.

Table 77 TCG tab options *(continued)*

Option	Description
Set Section	PA TCG calibration can be done in sections. For example, some calibration blocks, because of their design, can cause unwanted echoes at higher angles, from some corner traps or other features. By selectively ignoring some angles from a TCG calibration, it is possible to build the TCG in two separate sequences. Another practical application of the **Set Section** could be to calibrate at high depth only the lower angles, as the high angles are used only for one leg of inspection. **First Angle**: By default, it is the first angle of the group. Restricting this angle will gray out the corresponding angles in the amplitude chart. **Last Angle**: By default, it is the last angle of the group. Restricting this angle will gray out the corresponding angles in the amplitude chart. **Index Start**: Setting this value will gray out and exclude the corresponding zone in the amplitude chart and S-scan. **Index End**: Setting this value will gray out and exclude the corresponding zone in the amplitude chart and S-scan.
UT Settings	**A-Scan Start**: The start of the digitized range for calibration. **A-Scan Range:** The length of the digitized range for calibration. **Ref. Amplitude:** The target amplitude for calibration. When adding a point, a TCG point will be automatically applied so that the amplitude of the reference defect is equal to the **Ref. Amplitude**. **Gain:** You can change the gain to bring the amplitude higher or lower to facilitate the calibration process.

Table 77 TCG tab options *(continued)*

Option	Description
Set Gate A on Echo A	**Start:** Used to set where the gate starts in reference to the origin (can be expressed in mm or in.). The origin is either the zero of the ultrasound axis or the crossing point of gate I if the current signal is synchronized on **I/**. **Width:** Used to set the width (length) of the gate. **Threshold**: Used to set the height of the gate**.**
Add Point	After manually scanning a reference target over all focal laws, selecting Add Point will add a TCG point for each focal law. The point will be created at the position of the maximum echo in the gate. The Gain for each point will be set so that the amplitude for each focal law is equal to Ref. Amplitude (see Figure 4-5 on page 171).
Undo Last	Removes only the last TCG point created. To correct an invalid TCG point, remove it before using **Add Point** on the same reflector.
Last Added Points	Displays a table of the last added TCG points. There are three columns in the table: # (identifier), **Range** (PA-only, the First-Last angle used), and **Average Depth** (average of the TCG point position from all focal laws). This table is a live table; if you leave the TCG tab and come back, the table will be cleared.
Previous **Next**	Used to alternate between the parameters on the first and second pages.
Done	Select **Done** to apply and close the TCG calibration settings.

DAC

The distance-amplitude correction (DAC) curve is used to plot the variation in amplitude between reflectors of the same size at varying distances from the probe. A DAC does not change the gain, but instead sets a reference curve that

varies with distance (for the TCG, the TCG gain applied is set for a constant reference level).

It is possible to change from a DAC curve to a TCG curve (and vice versa) by selecting either option in the **Group** tab of the calibration wizard (see Table 78 on page 174).

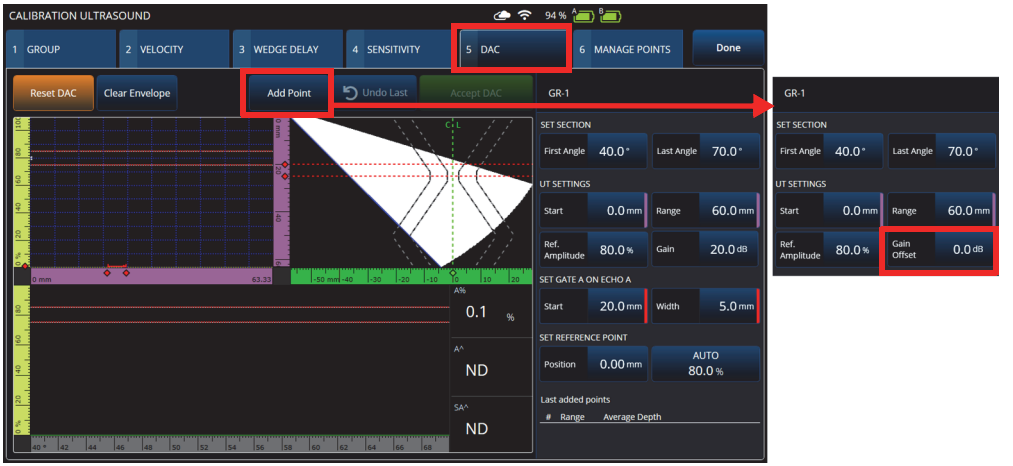


Figure 4-6 Calibration > DAC

Table 78 DAC tab options

Option	Description
Reset DAC	Resets the DAC curve. The DAC indicator at the top of the screen disappears.
Clear Envelope	Clears the A-scan envelope.
Add Point	After manually scanning a reference target over all focal laws, selecting Add Point will add a DAC point for each focal law. The point will be created at the position of the maximum echo in the gate (see Figure 4-6 on page 174).
Undo Last	Removes the last added DAC point.

Table 78 DAC tab options *(continued)*

Option	Description
Accept DAC	Accepts and saves the DAC calibration. The **DAC** indicator at the top of the screen turns green.
UT Settings	**A-Scan Start**: The start of the digitized range for calibration. **A-Scan Range:** The length of the digitized range for calibration. **Ref. Amplitude**: The reference level. The Reference Point level will be at this amplitude, and the first point of the DAC is set to this amplitude, using the Auto XX % button. **Gain**: This can be either adjusted manually or set using the Auto XX % button.
Gate A	The signal must be in the gate to use the **Add Points** function. **Start**: The start of the gate in reference to the origin. **Width**: The width of the gate.
Reference Point Position	The position of the origin of the DAC curve. You can use the **Reference Point Position** to adjust the initial slope of the DAC. A DAC point position cannot be before the reference position. By default, the reference is set at 0.
Last Added Points	The list of added DAC points. This list is live, so the list is cleared if you leave this tab and come back. There are 2 columns: # (identifier) and Depth (the DAC point depth).

TFM TCG

Users can set a TCG in TFM Delay-And-Sum (TCG is not relevant for Phase-Coherence Imaging).

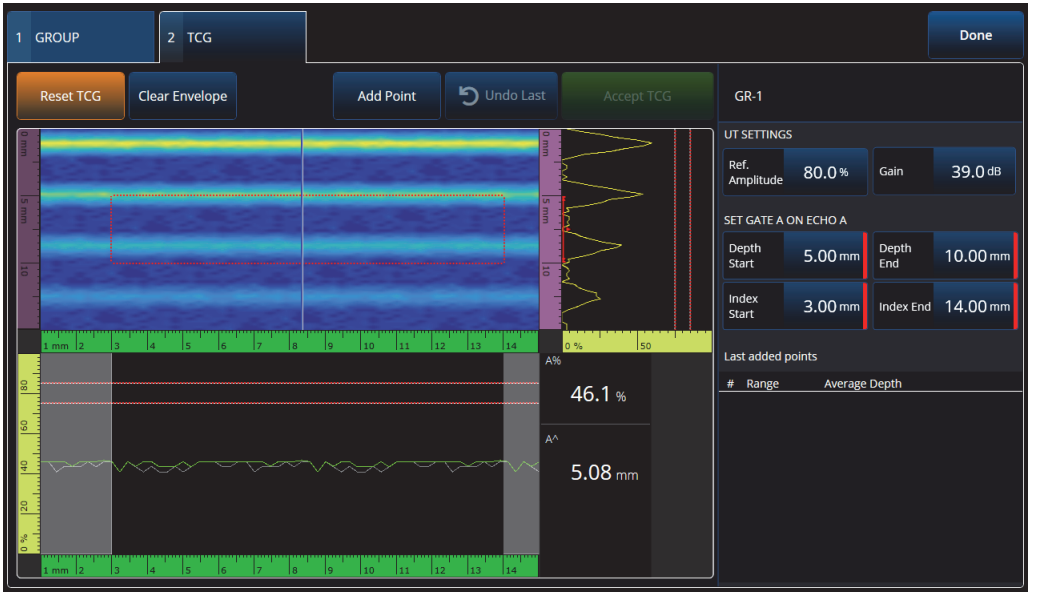


Figure 4-7 TFM TCG interface

Table 79 TFM TCG options

Option	Description
Ref Amplitude	Define the amplitude level of calibration.
Gain	Sets the initial gain before starting the calibration.
Depth Start/Index Start/Depth End/Index End	Used to position the gate. A reference reflector must pass through the gate to capture the maximum amplitude at each position.
Reset TCG	Reset the TCG. The TCG indicator at the top of the screen is cleared.
Clear Envelope	Clears the A-scan envelope.
Add Point	Adds a TCG point on the maximum envelope signal within the box gate.

Table 79 TFM TCG options *(continued)*

Option	Description
Undo Last	Removes the last added TCG point.
Accept TCG	Accepts and saves the TCG calibration. The TCG indicator at the top of the screen turns green.

4.4 Manage Points

The **Manage Points** tab (see Figure 4-8 on page 177 and Table 80 on page 177) is used to verify the TCG (or DAC) points value or to manually create or edit TCG (or DAC) points, bypassing the calibration wizard.

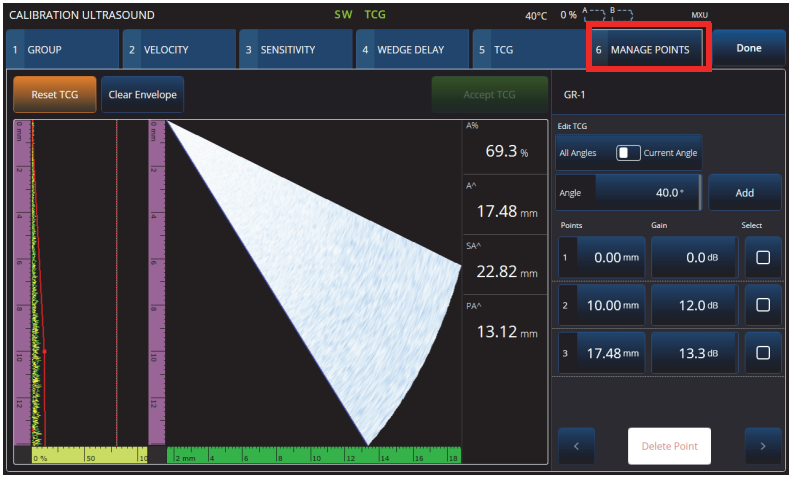


Figure 4-8 Calibration > Manage Points

Table 80 Manage Points tab options

Option	Description
Reset Calibration	Resets the TCG calibration. The **TCG** (or **DAC**) calibration indicator disappears.

Table 80 Manage Points tab options *(continued)*

Option	Description
Clear Envelope	Clears the envelope on the bottom view. The green line disappears.
Accept Calibration	Accepts and saves the **TCG** (or DAC) calibration. The **TCG** (or DAC) turns green.
Edit Points	To create or edit TCG points only for the selected **Angle** (or VPA), use the **Current Angle** option. To apply TCG points for all focal laws at the same time, use the **All Angles/VPA** option.
Angle (PA)	When using the **Current** option, select on which angle (VPA) the TCG point will be modified. This option also determines which A-scan is seen in the layout.
Add	Used to add a **TCG** (or **DAC**) point.
Points	Used to set the position on the ultrasound axis.
Gain (TCG)	Used to set the gain on the point.
Amplitude	Used to set the amplitude of the DAC curve at that position.
Select	Used to select a point. You can then delete it by tapping **Delete Point**.
A %	The peak amplitude of the signal detected in gate A.
A^	The depth in the part of the reflector producing the indication detected in gate **A**.
PA^	The distance on the part surface between the wedge (or probe) front face and the indication detected in gate **A**.
SA^	The sound path from the part entry point to the indication detected in gate **A**.
Done	Select **Accept Calibration** to save the Manage Points settings, and then select **Done**.

4.5 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 probe and wedge information. You can use the DGS calibration wizard to quickly set up and easily evaluate flaw size.

To perform a DGS calibration

1. Go to **Menu > Plan & Calibrate > Calibration Tools**.
2. On the **Group** tab, select the desired group, and then click the **DGS** button.
3. Select the **DGS** tab.
4. Under **Select Reflector**, select the type of reference reflector used to build the DGS curve: **SDH**, **FBH**, **K1 IIW**, or **K2 DSC**. (If **SDH** or **FBH** is selected, the hole diameter must be specified.)
5. Under **Set Curves Level**, 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 input part.
   c) Select **Warning Curves**, 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.
6. Under **Set Attenuations**, proceed as follows:
   a) Select **Cal. Block Att.** to specify the attenuation (dB/mm) for the material of the calibration block.
   b) Select **Specimen Att.** to specify the attenuation (dB/mm) for the material of the inspected part.
7. This step may be skipped if the sensitivity has already been adjusted. Under **Set Gate A on Echo A**, position the gate on a reference reflector, and then select **Auto XX%**.

8. Scanning over the reference, build an envelope in the A-scan, and then select **Calculate DGS**.

4.6 TOFD Calibration

This section explains how to calibrate a TOFD group.

4.6.1 WD & PCS

On the **TOFD Calibration** tab, you can calibrate the TOFD group so that the cursor readings are converted to depth instead of time. This is typically done during Analysis, but it can be done before. The TOFD calibration has a simplified process and is done outside of a calibration wizard. To access the **TOFD Calibration** options, go to **Plan & Calibrate > TOFD Calibration** (see Figure 4-9 on page 180 and Table 81 on page 181).

With the **TOFD Calibration**, you can calibrate either:

- The **Wedge Delay** and **PCS** (assuming a fixed velocity).
- The **Wedge Delay** (assuming the **PCS** and **Velocity** are correct).
- The **Wedge Delay** and **Velocity** (calibrates the wedge delay but validates the velocity. This calibration does not apply the velocity).

To perform a TOFD calibration

1. Choose the calibration type (see Figure 4-9 on page 180 and Table 81 on page 181).
2. Define the targets. For **Vel & WD** and **WD & PCS**, typically the two targets are Target 1= 0 (0 depth as it is the lateral wave) and Target 2 is the material thickness. For **WD** only, use any known reference.
3. Position the **Reference Cursor** on the first target (lateral wave or other) and the **Measurement Cursor** on the second (back wall echo or other).
4. Select **Calibrate**.

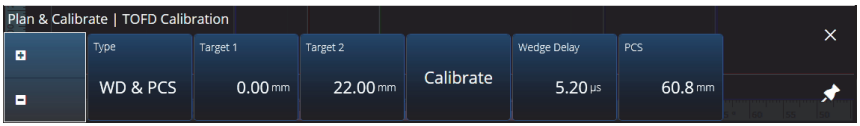


Figure 4-9 TOFD Calibration — WD & PCS

Table 81 TOFD Calibration — WD & PCS type options

Option	Description
Type: WD & PCS	Wedge delay and probe center separation: Used to calibrate, with one wizard, both the delay of the sound propagation within the wedge and the distance between the exit points of the two probes. Use the correct velocity to get an accurate calibration.
Target 1	Used to set the nominal depth of the first target (a value of 0 can be used to target the lateral wave at the surface).
Target 2	Used to set the nominal depth of the second target.
Calibrate	Before selecting **Calibrate**, make sure that both cursors are positioned on the echoes corresponding to the targets. When both cursors are correctly positioned, the **Calibrate** function will adjust the **Wedge Delay** and **PCS** values.
Wedge Delay	Used to set the delay corresponding to the sound propagation within the wedge. This value is automatically modified when **Calibrate** is selected.
PCS	Used to set the probe center separation (PCS). This is the distance between the exit points of the two probes (only available for the TOFD group). This value is automatically modified when **Calibrate** is selected.

4.6.2 Wedge Delay

Select the **Wedge Delay** calibration type for TOFD to calibrate **Wedge Delay** only. To modify the **Type (Wedge Delay)**, **Target 1**, **Calibrate**, and **Wedge Delay** options, go to **Plan & Calibrate > TOFD Calibration** (see Figure 4-10 on page 182 and Table 82 on page 182).

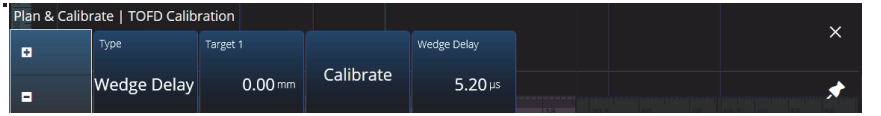


Figure 4-10 TOFD Calibration — Wedge Delay

Table 82 TOFD Calibration — Wedge Delay type options

Option	Description
Type: Wedge Delay	Used to calibrate the delay of the sound propagation within the wedge. The PCS and velocity must be correct for the calibration to be accurate.
Target 1	Used to set the nominal depth of the first target (a value of 0 can be used to target the lateral wave at the surface).
Calibrate	Before selecting **Calibrate**, make sure that the reference cursor is positioned on the echo corresponding to the target. When the cursor is correctly positioned, the **Calibrate** function will adjust the **Wedge Delay**.
Wedge Delay	Used to set the delay corresponding to the sound propagation within the wedge. This value is automatically modified when **Calibrate** is selected.

4.6.3 Encoder Calibration

For the encoder calibration, see “Inspection” on page 187

4.6.4 Velocity and Wedge Delay

To modify the **Type (Vel. & WD)**, **Target 1**, **Target 2**, **Calibrate**, **Wedge Delay**, and **Velocity** options, go to **Plan & Calibrate > TOFD Calibration** (see Figure 4-11 on page 183 and Table 83 on page 183).

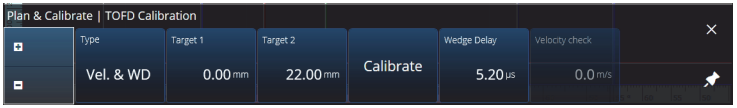


Figure 4-11 TOFD Calibration — Velocity and Wedge

Table 83 Plan & Calibrate — Velocity and Wedge options

Option	Description
Type: Vel. & WD	Velocity and wedge delay: Used to calibrate the delay of the sound propagation within the wedge.
Target 1	Used to set the nominal depth of the first target (a value of 0 can be used to target the lateral wave at the surface).
Target 2	Used to set the distance (expressed in mm or in.) of the second target for the calibration.
Calibrate	Used to set **Target 1** and accept the calibration.
Wedge Delay	Used to calibrate the delay corresponding to the sound propagation within the wedge. This value is automatically set when **Calibrate** is selected.
Velocity Check	Displays the velocity in the material of the inspected part after the calibration has been confirmed.

4.6.5 Lateral Wave Processing

Available only in analysis mode (MXU software and OmniPC software), the Lateral Wave Processing option enables you to synchronize sections of the lateral wave and remove the lateral wave over a specified interval. To access this option, go to **Plan & Calibrate > TOFD Calibration** (see Figure 4-12 on page 184 and Table 84 on page 184).

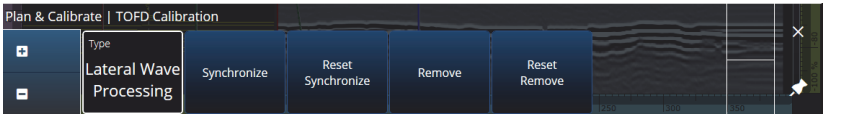


Figure 4-12 TOFD Calibration —Lateral Wave Processing

Table 84 Plan & Calibrate — Lateral Wave Processing options

Option	Description
Lateral Wave Processing	Select this type to use the lateral wave synchronization and lateral wave removal.
Synchronize	Synchronizes a TOFD B-scan, realigning a selected area to improve the readability. The zone to synchronize is delimited by the cursors in the scan axis, and by the gate A in the ultrasound axis. Before selecting **Synchronize**, do these steps: 1-Using the reference and measurement cursors on the scan axis in the B-scan, define the width of the section to synchronize. 2- Select a reference A-scan using the data cursor. This reference is typically a clean A-scan. This A-scan must be within the zone delimited by the reference and measurement cursors. 3- Make sure that gate A is active. 4- Position gate A around the lateral wave. The gate should be tight enough around the signal, but it must capture the lateral wave of all A-scans inside the zone. 5- Select **Synchronize**. Multiple zones can be independently synchronized. Repeat steps 1 to 5 using another zone.
Reset Synchronize	Remove the synchronization of the A-scans within the reference and measurement cursors on the scan axis. To remove all synchronization, place these cursors at the start and end of the full B-scan.

Table 84 Plan & Calibrate — Lateral Wave Processing options *(continued)*

Option	Description
Remove	Removes the lateral wave from the signal to help the detection of defects close to the surface. The removal is applied to a zone defined by the reference and measurement cursors on the scan axis. Multiple sections of removed lateral wave can be defined. To use the **Remove** option: 1- Follow the steps to synchronize the lateral wave. **Lateral Wave Removal** can only be performed on data that was previously synchronized. 2- Using the reference and measurement cursors on the scan axis in the B-scan, define the width of the section to remove. 3- Select a reference A-scan using the data cursor. This reference is typically a clean A-scan. This A-scan must be within the zone delimited by the reference and measurement cursors. 4- Select **Remove**.
Reset Remove	Restores the signal of the A-scans within the reference and measurement cursors on the scan axis.

NOTE

You must activate gate A to use **Lateral Wave Processing**. The gate can be active and hidden if it is not convenient to keep it displayed. Turn OFF the gates option in **View** to turn off the display of the gate while keeping it active for synchronization.

5. Inspection

The OmniScan MXU software interface is designed to be intuitive: you can familiarize yourself with its operation by navigating through the interface and testing various functions and buttons. For more details, see "OmniScan Interface" on page 27.

The basic inspection parameters are available in the ⌖ **UT Settings > General** submenu (see "UT Settings" on page 45).

5.1 Setting the Reference Gain

Auto (80%) reference gain

The suggested, default reference gain value can be selected by tapping the **Gain** area of the screen, and then selecting **Auto (80%)**. This setting adjusts the gain so that the reflector signal inside gate A reaches the reference level of 80% of the full-screen height. Position gate **A** accordingly before using **Auto (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.

To set the reference gain

- Select **UT Settings > Advanced > Reference dB > ON** to activate the reference gain.

5.2 Setting Up for an Inspection Using an Encoder

IMPORTANT

Before setting up for an inspection that uses encoders, an X or XY encoder must be properly connected to the I/O connector.

To set up for an inspection using an encoder

1. In the **Scan > Inspection > Type** list, choose the type of scan to be used to scan the part.
2. Select **Scan > Inspection > Encoders** to access the encoder setup screen, and set the encoder parameters according to your specifications (see Figure 5-1 on page 188). You can use a preset scanner, or edit the parameters for the axis from this menu.

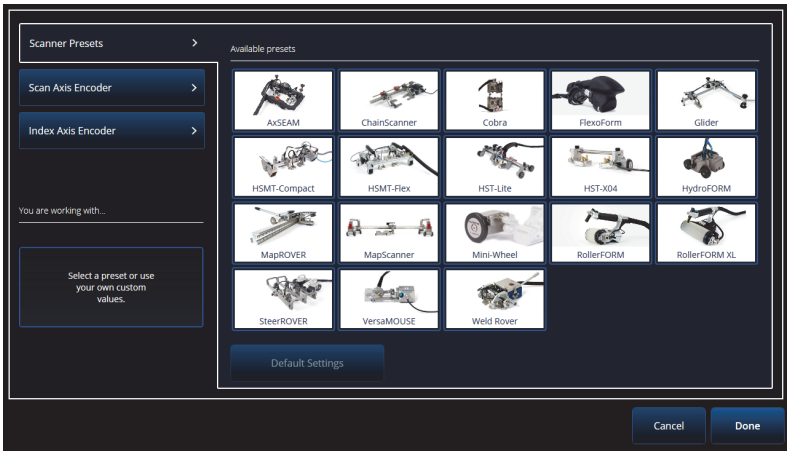


Figure 5-1 Scanner Presets selection list

3. If necessary, calibrate the encoder using the **Encoder Calibration** tool available on the right side of the encoder setup screen.
4. Define the area to inspect and the resolution on the **Scan > Area** menu.

5. When you are ready to scan, press the Play key ( ).

5.3 Configuring the Indication Table

The indication table displays detailed information on reflectors identified and recorded during an inspection. This information is used to create an inspection report.

To configure the indication table

1. Add an indication to the table by setting your layout and cursors on an indication (in Analysis mode) and then tapping **Add Indication** (see Figure 5-2 on page 189 on the left). Repeat for every indication you want to add.

Figure 5-2 Indication Table Manager window

2. Tap 🗁 **File > Indication Table Manager** to access the indication table (see Figure 5-2 on page 189 on the right and Table 85 on page 190).
3. Move through the list to review indications, add reference numbers and comments, and delete indications as required.

Table 85 Indication Table Manager options

Item number	Description
1	Reference number
2	Comments
3	Delete
4	Scrolling controls

6. Managing Files, Probes, Wedges, and Reports

Inspection setups and data are kept and organized using files, and presented in reports. You can use the 🗁 **File** menu to access multiple file parameters, the **Report** tool, and the **File Manager** (see Figure 6-1 on page 191).

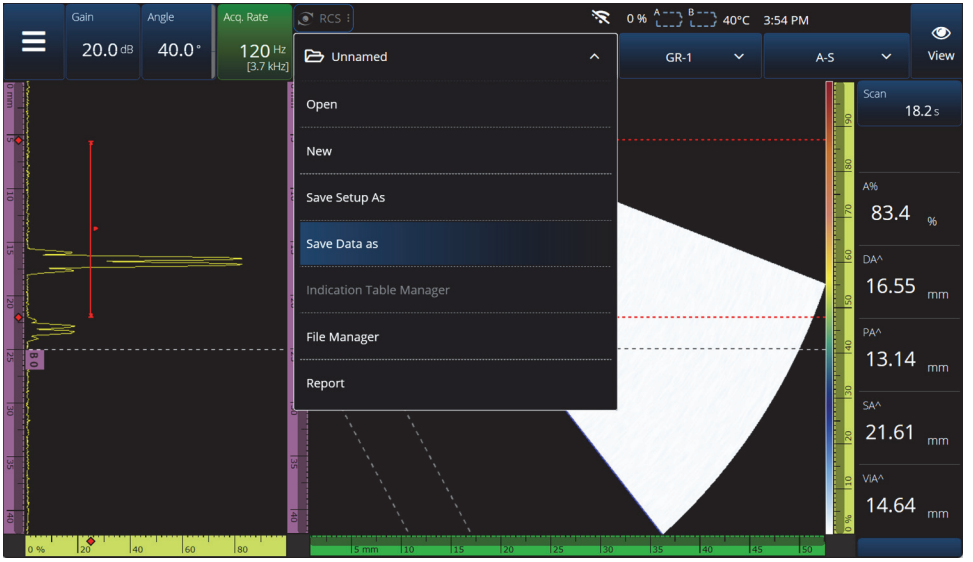


Figure 6-1 File menu

6.1 Saving, Naming, and Opening Files

It is recommended that you regularly save your setup and data files to prevent accidental data loss.

- To save your setup file, select **Save Setup As** on the 🗁 **File** menu (see Figure 6-1 on page 191).
- To name your data file, select **Save Data As** on the 🗁 **File** menu. You can subsequently save the file by pressing the Save key (●) located just above the Power key (⏻). Either option will open the file saving prompt. Enter the base name in the field. To save it as is, select the **File Increment=None**. If you want to add a number or a timestamp after the base file name, choose either option. The final file name can be seen in the **Preview**. Finally, if you chose a File increment other than None, you can optionally uncheck the **Prompt every time** to auto-increment the file after each save without going through this prompt every time you press the Save key (●).
- To open a file, select **Open** on the 🗁 **File** menu (see Figure 6-1 on page 191), and then choose the directory in which the file is located. You can open the setup file or data file by selecting the file type. You can also order the files alphabetically or by date using the **Filter** icon, and preview the selected file (see Figure 6-2 on page 192).

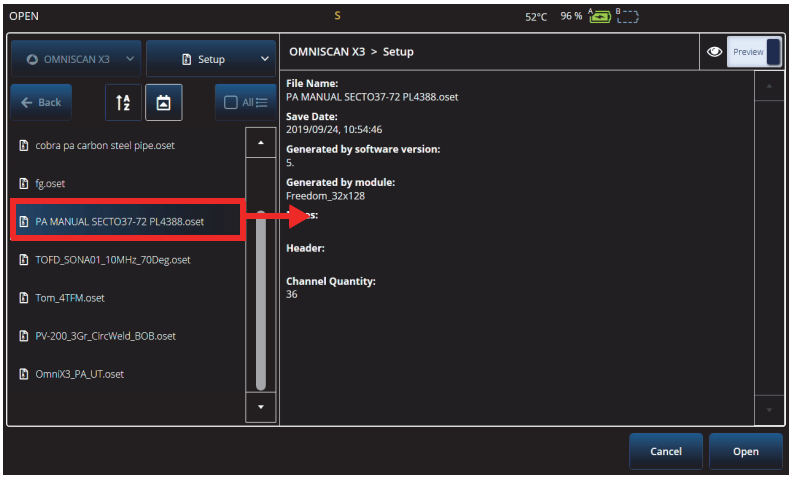


Figure 6-2 Open menu

6.2 Using the File Manager

To access various options for managing your files, select **File Manager** on the 🗁 **File** menu (see Figure 6-3 on page 193 and Table 86 on page 193).

Figure 6-3 File Manager window options

Table 86 File Manager options

Item number	Description
1	Select the source drive: OmniScan X3 hard drive, USB, SD card.
2	Select (filter) the desired file type: setup, data, image, report, etc.
3	Navigate to folders on the selected drive.
4	File information. In multiselection mode, only the names, the file count, and their total size are displayed.
5	Transfer the selected file(s) to a destination of your choice.
6	Rename the selected file. Not available in multiselection mode.
7	Upload to OneDrive or download from OneDrive. See “Using OneDrive With the File Manager” on page 195.

Table 86 File Manager options *(continued)*

Item number	Description
8	Delete one or multiple files.
9	Remaining space on the selected drive.
10	To transfer or delete multiple files at once, turn **Multiselection** ON.

To transfer files from an external drive (USB) to the OmniScan X3 unit

1. Insert the USB key (or SD card) in the unit.
2. Select the USB as the source drive.
3. Select the file type to transfer: setup, data, palette, overlay, etc. You can transfer multiple files at once if they are the same file type.
4. Navigate the folders and subfolders to find your files. Tap once on a folder name to move to that folder. Use the **Back** button to go back.
5. Tap once on the file you want to transfer, or turn on multiselection, and tap every file you want to transfer (this will select the box next to the files).
6. Tap the **Transfer** button.
7. Make sure you select the OmniScan X3 hard drive as the destination and tap **Copy to**.
8. The files are now available on the unit. Tap **Done** to exit the File Manager.

To transfer files from the OmniScan X3 unit to an external drive

1. Insert the USB key (or SD card) in the unit.
2. Select the OmniScan X3 drive as the source drive.
3. Select the file type to transfer: setup, data, palette, overlay, etc. You can transfer multiple files at once if they are the same file type.
4. Tap once on the file to transfer, or turn on multiselection, and tap every file to transfer (this will select the box next to the files)
5. Tap the **Transfer** button.
6. Select the destination drive (if more than one is available, make sure that the correct one is selected).
7. Tap **Copy to** to transfer the files.

8. The files are now available on the drive. They are located in the folder *olympus_x3* and sorted in subfolders according to file type.
9. Tap **Done** to exit the **File Manager**.

Using OneDrive With the File Manager

To use OneDrive to transfer files to and from the cloud, you must first be connected to Internet, then you must be logged in to a OneDrive account. See "Connectivity Settings" on page 96. In the File Manager, the OneDrive button is now active (see Figure 6-4 on page 196)

Sending Files to OneDrive

Select the file(s) to send to OneDrive and tap **Upload to OneDrive**. The files are sent to the OmniScan X3 Series folder on OneDrive. A green checkmark appears on the file, indicating the file is located on the OmniScan X3 hard drive and on the cloud.

Deleting Files Synched With OneDrive

When deleting a file that is only local on the hard drive, the file is permanently destroyed. If the file is uploaded to OneDrive (it has a green checkmark), deleting the file only destroy the local copy, and keep the copy on the cloud. A cloud icon appears next to the file to indicate that the file only exists in the cloud for now.

Deleting files that are not on the OmniScan X3 (only in OneDrive) is not possible. Use a computer to manage the files in OneDrive.

Downloading Files From OneDrive

Any file in the appropriate folder in OneDrive (data files in OmniScan X3 Series/Data, setup files inOmniScan X3 Series, etc.) will also appear in the File Manager. If no copy of the file exists locally on the OmniScan X3, the file appears with a cloud icon next to it.

To get a copy of this file from OneDrive to the OmniScan X3, simply tap **Download from OneDrive**. The icon next to the file change from the cloud icon to a green checkmark, which means a copy of this file is present on OneDrive and on the OmniScan X3.

If multiple instruments are connected to the same OneDrive account, all of them have access to the same files. This can be used to remotely sharing files between instruments. One instrument sends a file to OneDrive, and the others can now see it in their own **File Manager** and can download a local copy.

Synchronizing With OneDrive

If a file is saved OneDrive and locally, but there is a more recent version on one of them (if a setup saved in OneDrive is modified in the MXU software for example), the option Synchronize OneDrive allows to copy only the most recent copy on both OneDrive and the OmniScan X3. A file that is more recent on the OmniScan X3 has an arrow up icon next to it, while a more recent file on OneDrive has an arrow down icon.

If a file is corrupted, or no longer exists, an orange triangle icon appears on the file. Read the file preview for help on the error.

Figure 6-4 Possible statuses of files in the File Manager

1. A file is on both the OmniScan X3 and OneDrive, but one copy is more recent. Synchronize with OneDrive to keep the most recent in both places.
2. A file in the OneDrive directory, but not on the OmniScan X3. Download it to get a local copy.
3. A file not yet synchronized with OneDrive. Upload to OneDrive to send a copy to the cloud.
4. A file both in the OmniScan X3 and OneDrive, with the same name and saved date.

6.3 Probe & Wedge Manager

Use the **Probe & Wedge Manager** if you want to create custom probe and wedge configurations that are not in the default list provided by Evident (see Figure 6-5 on page 197 and Table 87 on page 197).

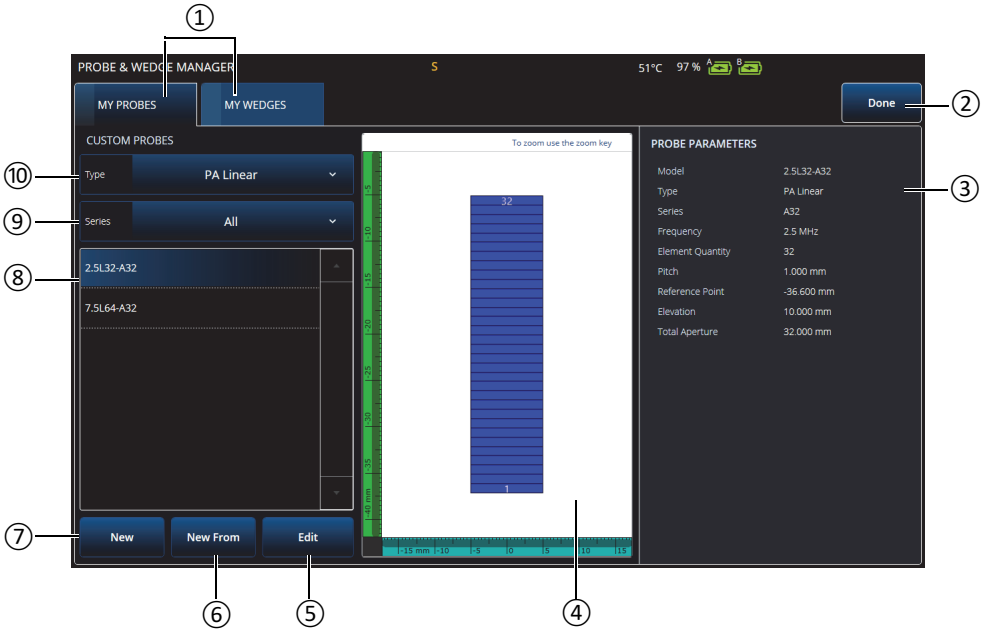


Figure 6-5 Probe & Wedge Manager window

Table 87 Probe & Wedge Manager window options

Item number	Description
1	**My Probes** and **My Wedges** tabs.
2	Use the **Done** button to exit the **Probe & Wedge Manager** window.
3	Section to see a preview of all parameters for the selected probe or wedge.

Table 87 Probe & Wedge Manager window options *(continued)*

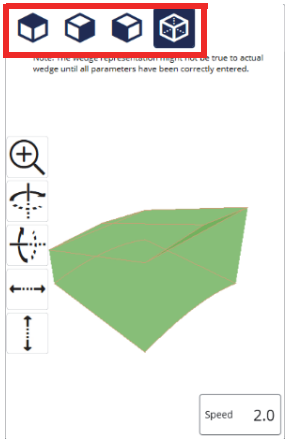
Item number	Description
4	Section to visualize the probe or wedge. A 2-D reconstruction of the selected probe or wedge is displayed according to the parameters defining this probe or wedge. Wedges can also be displayed in 3D. It is possible to change the 3D view by selecting the top cube icons on the visualization pane. The cube icon on the far right enables you to freely rotate and pan the 3D view and offers more options. To adjust the 3D view, select any of the icons in the view, and follow the on-screen instructions. You can use the OmniScan X3 knob to adjust each selected view parameter. 
5	Use the **Edit** button to edit the selected probe or wedge. Note: To delete a probe or wedge, first click on **Edit**, and the red “X”, and then the **Delete** button. 
6	Use the **New From** button to create a probe or wedge from an existing or standard model. This is a convenient way to create a new probe/wedge and save time when entering the parameters.
7	Use the **New** button to create a probe or wedge from scratch.

Table 87 Probe & Wedge Manager window options *(continued)*

Item number	Description
8	Lists all available probes and wedges on the local instrument. Select any probe or wedge from the list to visualize its parameters or edit it.
9	Use **Series** to quickly jump to a specific probe series. You can create probe series yourself; this can be useful to include probes from other manufacturers or custom-made models ordered from Evident.
10	Use **Type** to filter the different probes or wedges. Only types selected are shown and considered.

6.3.1 Nomenclature Information on Probes and Wedges

By default, the OmniScan MXU software establishes the phased array (PA) probe's **Reference Point** at the position of the first element. To set the **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. To avoid issues with a custom probe, make sure that the value of the **Reference Point** is negative, and that its absolute value is equal to Reference point = -1 × (number of elements) × Probe Pitch.

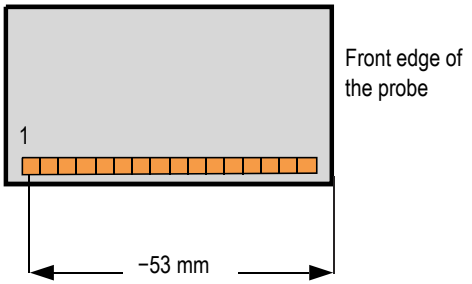


Figure 6-6 PA probe reference point measurement

The wedge's **Reference Point** is for the UT group only. It is 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 6-7 on page 200). The beam exit point is usually marked by a line on the wedge.

This value is negative because the OmniScan MXU software establishes the wedge **Reference Point** by default at the beam exit point. To place the **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).

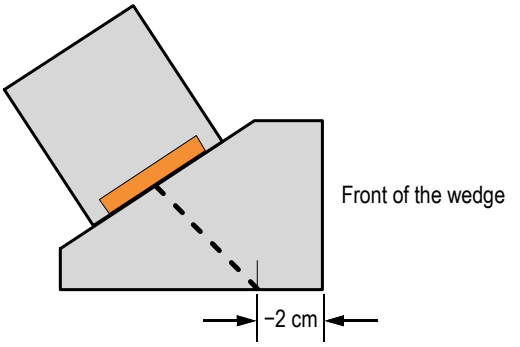


Figure 6-7 UT wedge reference point measurement

By default, the OmniScan MXU software establishes the phased array wedge's **Primary Offset** point at the position of the first element. To set this reference point at the front edge of the wedge, in **Primary Offset**, enter the distance between the front edge of the wedge and the position of the first element. The value must be negative (see Figure 6-8 on page 200).

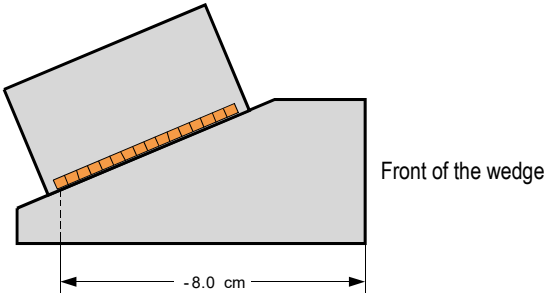


Figure 6-8 Primary offset measurement

A **Secondary offset** of **0** indicates 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 6-9 on page 201).

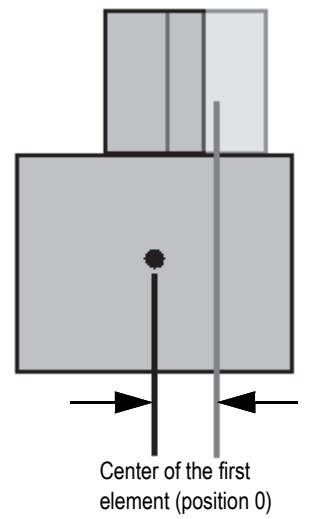


Figure 6-9 Secondary offset

6.3.2 Adding a Probe or Wedge

To add a probe or wedge

- Select the **New** or **New From** button. This activates the edit mode. Input all the parameters. You need to enter all the parameters correctly to generate an accurate visual representation.

6.3.3 Editing a Probe or Wedge

To edit a probe or wedge

- Select the **Edit** button. This activates the edit mode. You need to enter all the parameters correctly to generate an accurate visual representation. This is especially important for the wedge representation.

Editing a probe or wedge is facilitated by live indicators displayed on the left. The indicators appear for certain parameters when selected. Only variables showing a physical dimension can be shown (see Figure 6-10 on page 202 and Figure 6-11 on page 203).

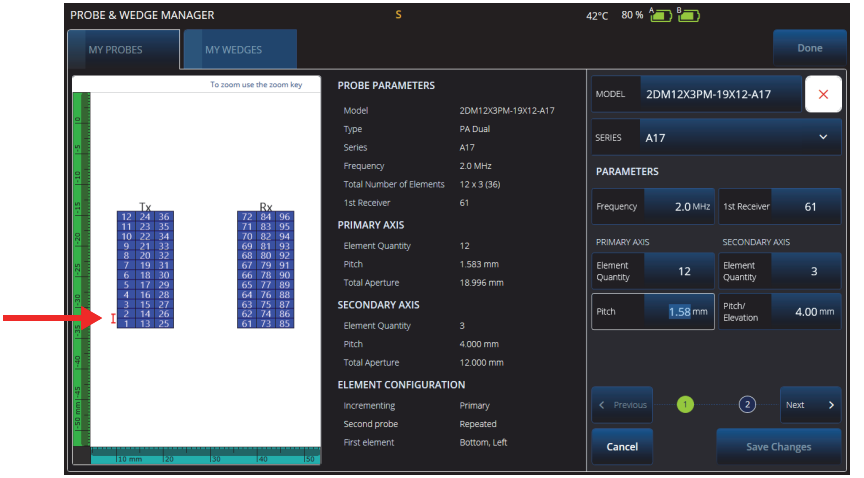


Figure 6-10 Editing a PA Dual probe — Red indicator emphasizing the selected parameter

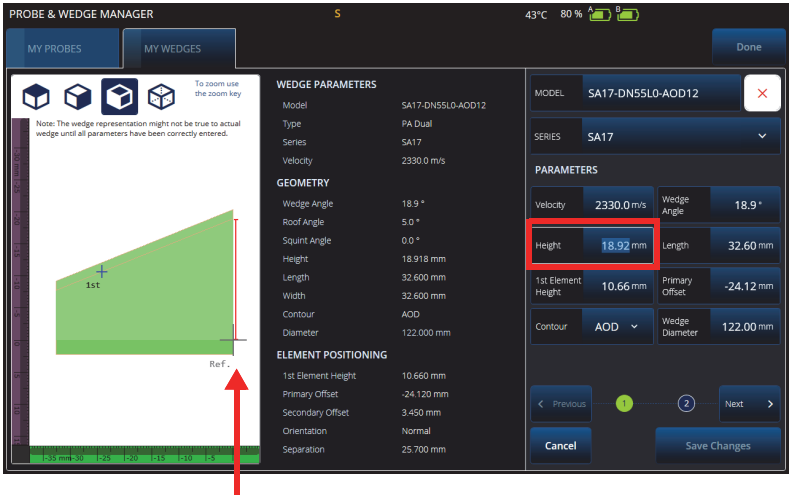


Figure 6-11 Editing a PA Dual wedge — Red indicator emphasizing the dimension

For PA probes, the element number is displayed. This can ease editing the parameters of complex dual PA probes, which can have a variety of wiring and element patterns, depending on the probe manufacturers.

The **Element Configuration** parameters are used to adjust and confirm the element configuration, which is eased by the dynamic live 2D representation (see Figure 6-12 on page 204).

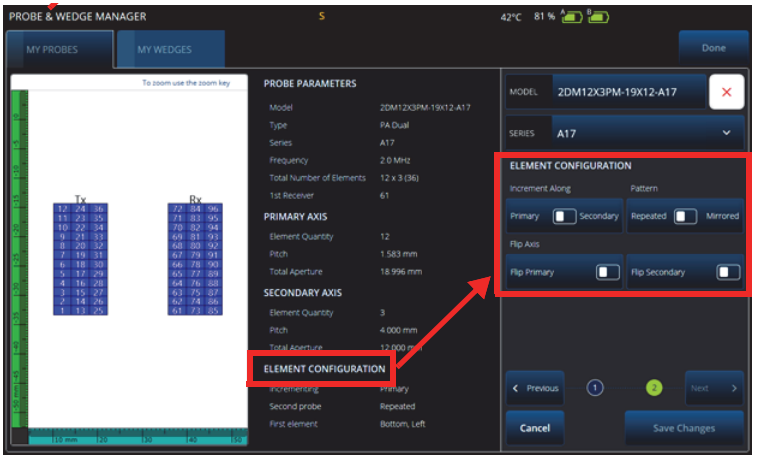


Figure 6-12 PA Dual probe advanced parameters — Element Configuration parameters

6.3.4 Deleting a Probe or a Wedge

To delete a probe or a wedge

- Select a probe or wedge, click **Edit,** the red "**X**", and then the **Delete** button.

6.4 Reports

On the 🗁 **File** menu, select **Generate Report** to configure and print reports using the **Report Manager** (see Figure 6-13 on page 205 and Table 88 on page 205).

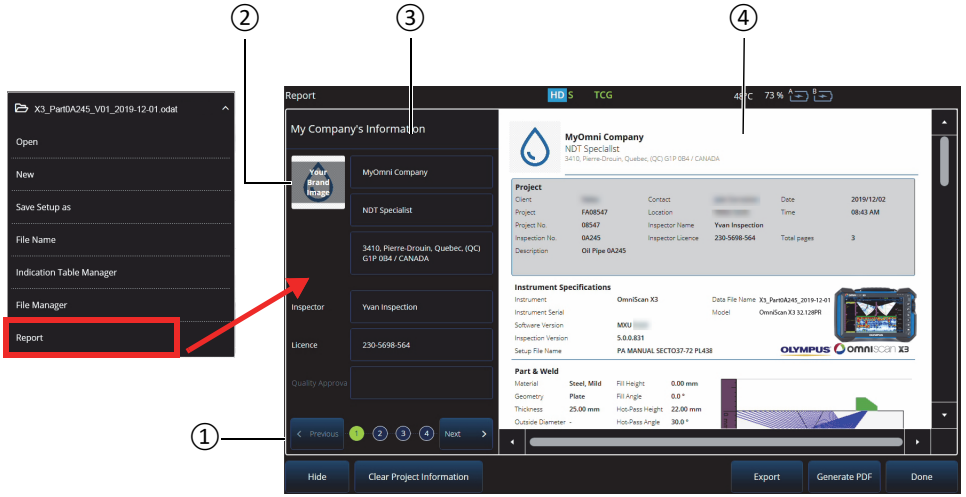


Figure 6-13 Report Manager window

Table 88 Report Manager window options

Item number	Description
1	Scroll multiple configuration pages.
2	Add your company logo or other images.
3	Edit the fields in this section to configure your report.
4	Report preview section

7. Total Focusing Method (TFM)

You can set up and use a TFM group on the OmniScan X3 flaw detector.

7.1 TFM Law Configuration

To set up a TFM law configuration

1. Select ☰ Main menu > **Wizard (Plan & Calibrate) > Scan Plan**.
2. Set up the **Part & Weld**, and the **Probes & Wedges**.
3. On the **Groups** tab, select **Law Config. > TFM**, and complete your desired TFM law configuration (see Figure 7-1 on page 208).

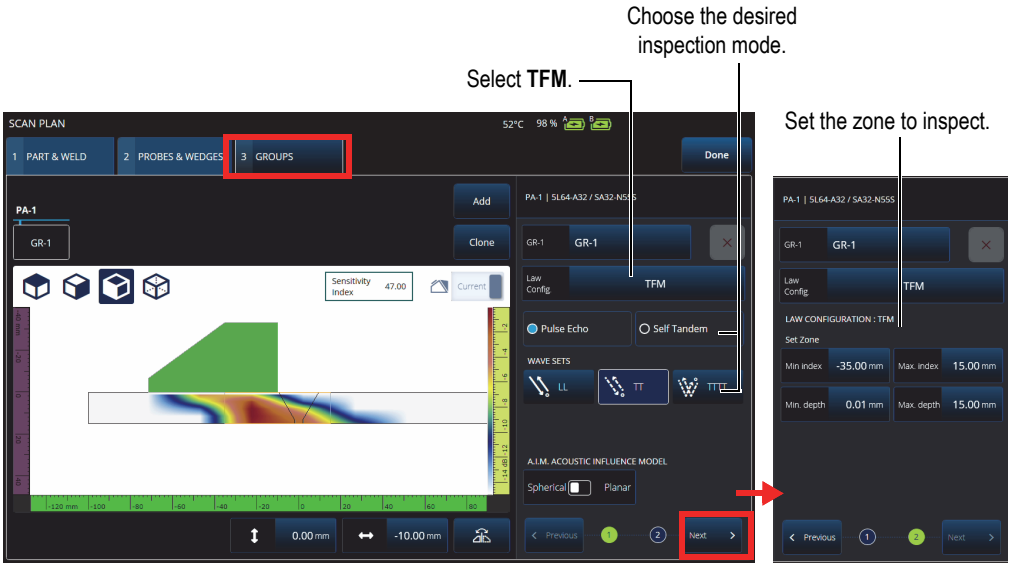


Figure 7-1 TFM on the Groups tab

7.2 Acoustic Influence Map (AIM)

The Acoustic Influence Map (AIM) modeling tool can help you select the correct propagation mode (or wave set) for a given flaw. On the OmniScan X3 flaw detector, you can use this tool to create a model that represents an amplitude map in the material. Each pixel of the AIM represents the theoretical amplitude that could be obtained if a reflector was at this position. The map is coded using different colors that each represent a specific 3 dB range.

For example, the red color indicates that the ultrasonic response is very good and varies between 0 and −3 dB with respect to the maximum amplitude. Orange indicates a range from −3 dB to −6 dB, yellow from −6 dB to −9 dB, etc. The maximum amplitude for each map is specified with the **Sensitivity Index**. This value represents the maximum sound pressure of the currently selected AIM (the highest amplitude pixel). This can help to decide between two propagation modes typically the mode with the highest sensitivity index should have a better SNR on the specified reference defect. When configuring the AIM model, you can choose either a **Spherical** (volumetric) type flaw, such as porosity, or a **Planar** type flaw, such as a crack (see Figure 7-2 on page 209).

When you adjust the defect type, the AIM model is automatically updated to show the predicted amplitude response of the selected wave set (propagation mode) for that flaw. This will help you select the wave set that is best suited for your inspection.

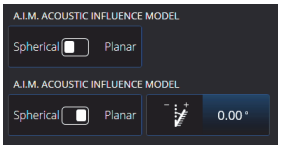


Figure 7-2 AIM Acoustic Influence Map tool

7.3 TFM Settings

After you exit the **Scan Plan** wizard menu (step 1. on page 207), the TFM settings have replaced the **UT Settings** under the ☰ Main menu > ▦ **TFM Settings**. Figure 7-3 on page 209 shows the **Advanced** parameters under ▦ **TFM Settings**.

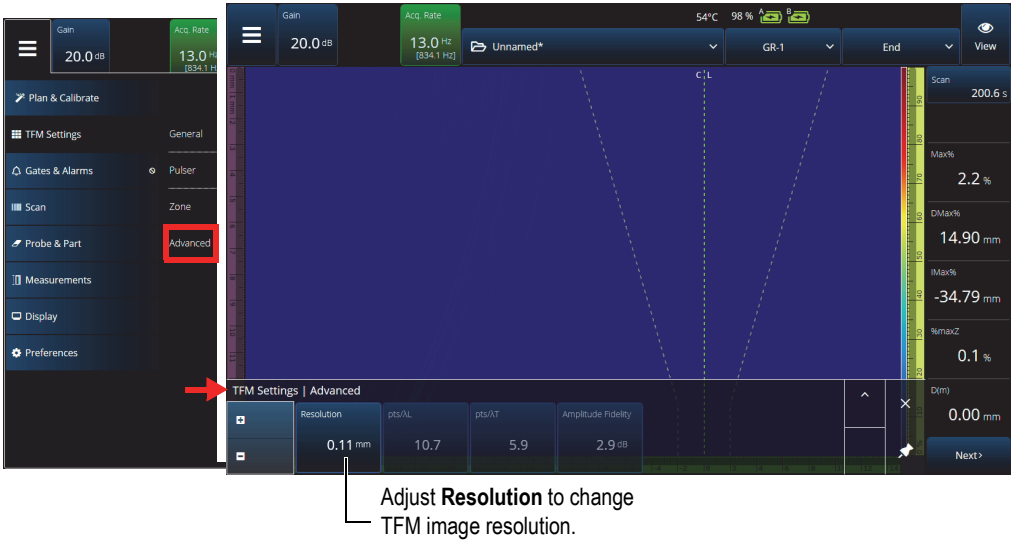


Figure 7-3 TFM Settings — Advanced parameters

See “TFM Settings” on page 56 for more information on TFM settings.

7.4 Phase Coherence Imaging (PCI)

This method is only available on the OmniScan X3 64 model. PCI uses a similar algorithm to the standard TFM, but instead of summing amplitude elementary A-scans, it sums the phase of those elementary A-scans, with the corresponding delays at each pixel. Instead of obtaining an amplitude at every pixel, we are acquiring a coherence value. Typically, base material without flaws, and long planar reflectors, have a low coherence. It is also typical for corners, tips, small reflectors to have a high coherence factor.

To toggle between **Phase Coherence** Imaging and regular **Delay-and-Sum**, tap on **Menu > TFM Settings > General > TFM Method**.

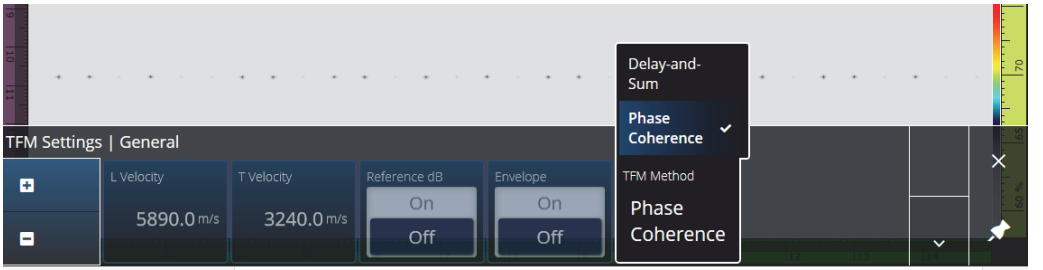


Figure 7-4 TFM Settings — PCI

Like the Envelope feature, PCI is applied to all TFM groups at the same time.

In a PCI inspection, almost all settings from TFM can be used: Filters, Voltage, Sparse, and Resolution. However, Table 89 on page 211 lists the features that are modified.

Table 89 Modified features

Feature	Description
Gain	The Gain is grayed out, because PCI is an amplitude-free technique, so adding gain is not relevant. Instead, adjusting the color map to interpret what is considered relevant is accomplished by adjusting the color palette and the zoom-level on the palette. It is also impossible to saturate a PCI signal by nature, so subtracting gain to remove saturation is pointless.
TCG Calibration	TCG is not required to be adjusted on phase. TCG is a concept that sets the reference amplitude, but phase coherence does not need to be equalized.
Readings	Some readings are modified to indicate "Coherence" rather than "%" or "Amplitude". This is to remind the user that TFM in PCI mode is expressed in terms of phase coherence rather than amplitude. Hardness Depth reading is only displayed on Omniscan X3 64.

7.5 Plane Wave Imaging (PWI)

Plane Wave Imaging (PWI) is a data collection method, similar to the Full Matrix Capture (FMC).

It is based on plane wave emission at different angles and reception of the associated elementary A-Scans.

Delays for summation are calculated using the plane wave propagation at each angle and the reception path for each element.

PWI is available on the OmniScan X3 64 with the following settings:

- Linear probe
- Plate or AOD
- TT or LL wave sets
- One group

8. Analysis with the OmniPC Software

In the OmniPC software, you can analyze OmniScan X3 data files. The majority of the menus found in the MXU software are present. Although many fields are read-only, the interface remains similar to the MXU software.

Start the analysis by using the **Open** button to select a file to analyze. The OmniPC software is organized in tabs:

OmniPC

Includes the same control as the main interface of the MXU software: Gain, VPA selector, layout selection, view options. The Zoom key (🔍) on the OmniScan X3 unit is replaced by a zoom button in this tab. It is also possible to use keyboard shortcuts instead of the zoom button.

UT Settings

Same as the ☰ Main menu **>** ⌖ **UT Settings** in the MXU software, except that all fields are read-only.

Gates

Same as the ☰ Main menu **> Gates** in the MXU software. On this tab, it is possible to turn ON/OFF gates and manipulate them. It is not possible to change the gate or the A-scan synchronization in post-processing.

Scan

Contains information from the scan. Read-only.

Probe & Part

Same as the ☰ Main menu **> Probe & Part** in the MXU software, but it does not include the **Probe & Wedge Manager**. This tab enables you to edit the probe position and the overlay.

Focal Law

Read-only section containing information on the current group focal law configuration.

Measurements

Turns ON/OFF the linked cursors.

Display

Enables you to edit the same parameters as on the ☰ Main menu **> Display** in the MXU software. You can also select the **Thickness** mode from the **Display** menu rather than from the **Gate** menu in the MXU software.

Preferences

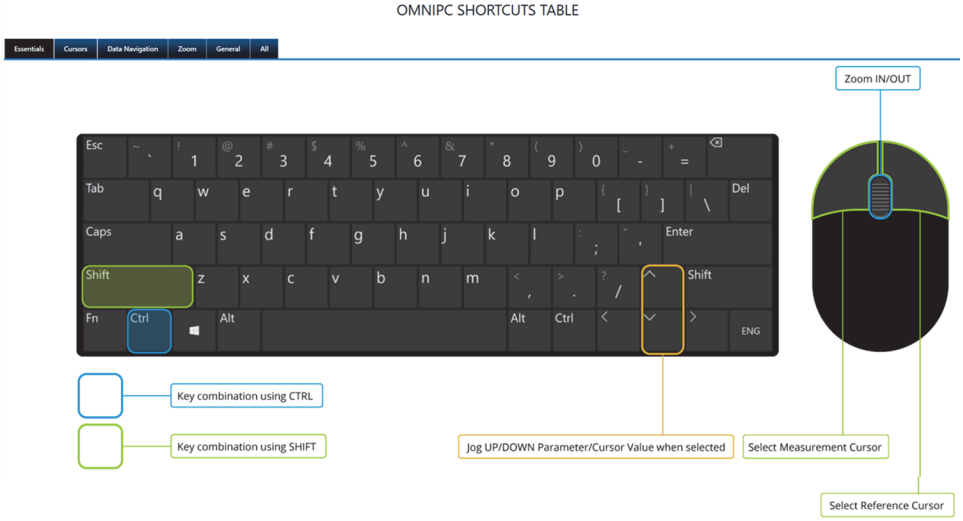
Enables you to change the language and the units (metric/imperial). System and legal information are also available to read.

Help

Opens in a new window displaying the list of shortcuts available in the OmniPC software. Learning these mouse and keyboard shortcuts can help you increase your productivity (see Figure 8-1 on page 215).

TIP

You may select **View** > **Scan Plan View** to see the scan plan.



DESCRIPTION	STANDARD SHORTCUT
Set Data Cursor	Mouse Left DoubleClick
Set (and move) reference cursors (all)	SHIFT + Mouse Left Click
Set (and move) measure cursors (all)	SHIFT + Mouse Right Click
Jog Selected UP	↑
Jog Selected DOWN	↓
Open	CTRL O
Escape from Zoom Mode	ESC
Zoom IN concentric > When hovering Views	CTRL + Mouse Wheel Up
Zoom OUT concentric > When hovering Views	CTRL + Mouse Wheel Down
Reset All Zoom	CTRL 0

Figure 8-1 OmniPC Shortcuts Table

9. Olympus Scientific Cloud (OSC) Connection

To use the X3 Remote Collaboration Service (X3 RCS), you need to create an account on the OSC Platform and your OmniScan X3 unit needs a valid connection to the Olympus Scientific Cloud (OSC).

The unit does not need to be connected to the OSC to download new software versions in the OmniScan X3 Launcher (see Figure 1-2 on page 22). To connect to the OSC, make sure that the OmniScan X3 unit is connected to the Internet.

TIP

For a step by step procedure, refer to the OSC and X3 RCS Registration Guide available on the X3 Remote Collaboration Service page.

To display the OSC connection settings, tap **Preferences** > **Connectivity Settings** > **OSC Connect** (see Figure 9-1 on page 218 and Figure 9-2 on page 218)

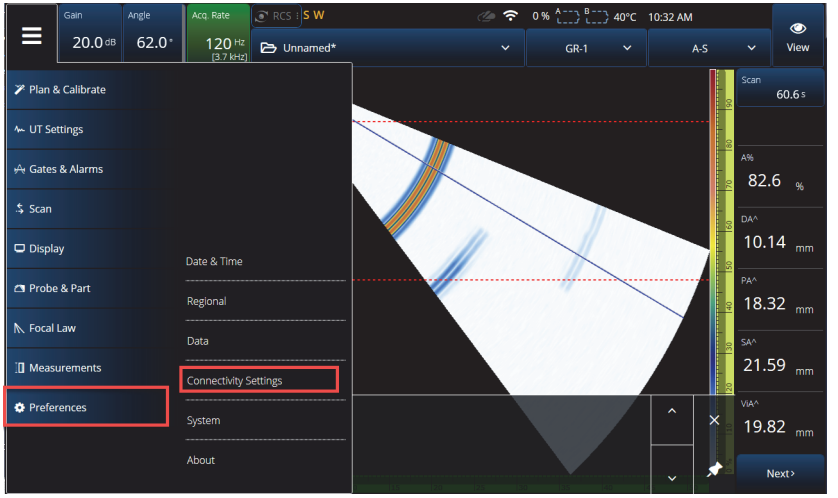


Figure 9-1 Connectivity Settings menu

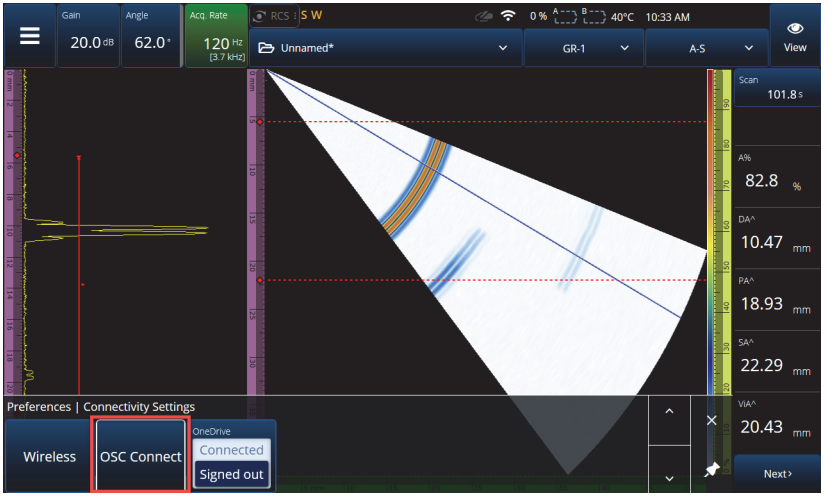


Figure 9-2 OSC Connect menu

9.1 OSC Connection Status

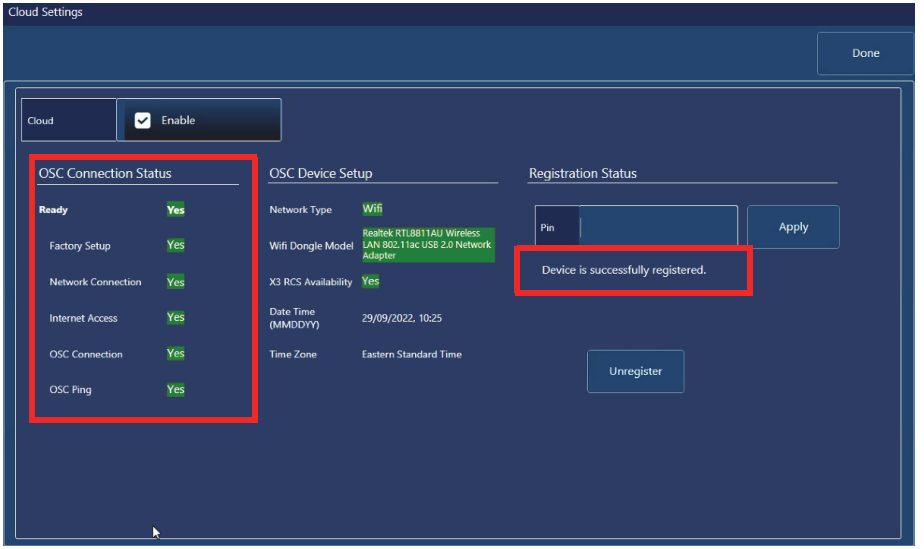


Figure 9-3 OSC Connection Status and Registration Status message

- **Factory setup**
  - Indicates whether the factory setup of the instrument was correctly configured, meaning that the provisioning process has been completed:
    - Green means that it was correctly configured in a factory, so the provisioning data on the device is coherent and present.
    - Red means that the provisioning data on the device is absent or not coherent.
- **Network Connection**
  - Indicates whether the OmniScan X3 unit's wireless or Ethernet network connection is properly connected:
    - Green means that your unit is currently connected to a hotspot wireless network or Ethernet network connection (connector not available at the time of printing).
    - Red means that your unit is not connected to a network.
- **Internet Access**

  - Indicates that the chosen network connection enables access to the Internet. The indicator may be red if the Internet access requires double authentication or is protected by a firewall:
    - Green means the unit has access to the Internet via the chosen network connection.
    - Red means the unit does not have access to the Internet via the network connection.

- **OSC Connection**
  - Indicates whether the OmniScan X3 unit is successfully connected to the server or IOT Hub.
    - Green means that the unit is successfully connected to the IOT Hub with the data stored on the device.
    - Red means that the unit failed to connect to IOT Hub with the data stored on the device.
- **OSC ping**
  - Indicates whether the OmniScan X3 unit successfully sends and receives messages to and from the OSC:
    - Green means that the unit successfully sends and receives messages.
    - Red means that the unit cannot send and receive messages.

9.2 OSC Device Setup

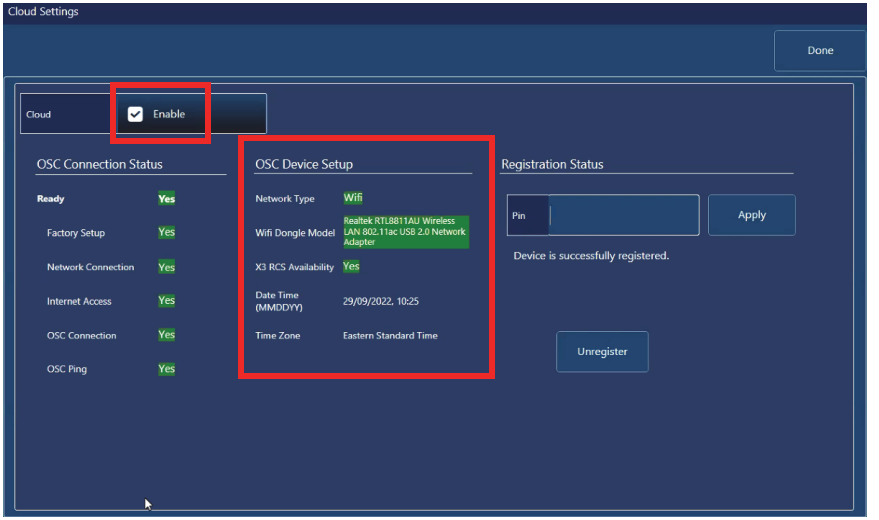


Figure 9-4 OSC Device Setup

- **Network Type**:
  - Indicates whether WiFi or Ethernet is in use (displays green).
    - Red indicates that there is no network connection.
- **WiFi Dongle Model** (if applicable):
  - Name of the USB dongle(s) currently in use. Two or more may be inserted.
  - Green indicates that the dongle is officially supported by the OmniScan X3 unit.
  - Red indicates the dongle is not officially supported to use OSC features.
- **X3 RCS Availability**:
  - Indicates whether or not the instrument has access to the X3 Remote Collaboration Service (X3 RCS) under its tenant.
- **Date Time (MMDDYY)**:
  - Shows the date/time format: dd/mm/yyyy, hh:mm.
- **Time Zone:**
  - Shows the currently selected time zone.

9.2.1 Cloud Enable Check Box

When your OmniScan X3 is connected to the Internet, you must check **Enable** to allow the unit to connect to the OSC (see Figure 9-4 on page 221).

When the **Ready** and **Enable** statuses both say **Yes**, the OmniScan X3 unit is ready to connect to the OSC.

9.2.2 Registration Status

Depending on the **Registration Status** message (see Figure 9-3 on page 219), follow the actions described in Table 90 on page 222.

Table 90 OSC Connect Registration statuses

Registration status message	Action
No registration request found for the device. Please register the device on the Olympus Scientific Cloud.	See "No Registration Request Found" on page 222.
Please authenticate the registration request by entering your 4 digit pin.	Once the OSC provides you with a PIN, the status changes in the OmniScan X3 unit. Enter your four digit PIN and click **Apply**.
Device is successfully registered.	When you get this message, the device registration is completed. Select **Done** in the upper-right corner to exit the menu.

If there are issues with the connection to the OSC, you can select **Unregister** on the OmniScan X3 unit and also unregister the device on the OSC website (must be done by an administrator of the account), and then start the connection process again.

9.2.3 No Registration Request Found

This message indicates that you must register your OmniScan X3 unit's serial number with your OSC account. The serial number is indicated on the bottom of the OmniScan X3 unit. You can also find it by selecting the information (**i**) button in the bottom-left corner of the **OmniScan Launcher**. Take note of the number.

To register the unit on the OSC, you need to use a computer with Internet access (but keep the OmniScan X3 unit turned on and close by).

- If you need to create a new account on the OSC, go to www.olympus-ims.com, and click **Cloud Log in** to create your account (see Figure 9-5 on page 223).

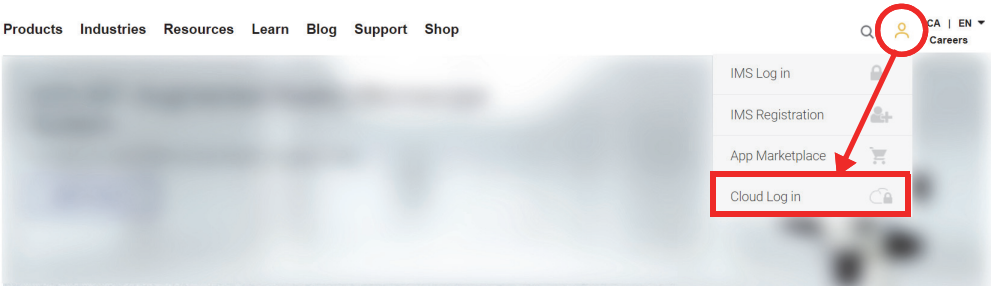


Figure 9-5 Cloud Log in

- If you need to log into a company account, an account administrator must send you an email invitation. A company account is used to share X3 RCS licenses and keep track of all the registered devices in the company.

— After you are logged into your OSC account, go to the **My Devices** section, and click **Add Device.**

— Enter the OmniScan X3 *serial number*. Once added, you will be provided a PIN. At this point, the **Registration Status** in the OmniScan X3 unit should change (see Table 90 on page 222).

10. OmniScan X3 Remote Collaboration Service (X3 RCS)

The X3 Remote Collaboration Service (X3 RCS) is a subscription-based service embedded in the MXU software to save time and costs. This service enables you to consult collaborators and get critical support while in the field. Powered by Zoom, the service allows:

- Live screen sharing
- Remote control
- Annotations

If you are in the field, you can also use the X3 RCS with your cell phone to access a meeting with the following features:

- Video and audio communication
- Collaborator screen sharing

Licenses for the X3 RCS are shared among users in the same Olympus Scientific Cloud (OSC) account. With one license, every user has access to the license, but only one can use it at a time. When the meeting is done, the license is released for another user.

TIP

For a step by step procedure, refer to the OSC and X3 RCS Registration Guide available on the X3 Remote Collaboration Service page.

10.1 Requirements

To use the X3 RCS, you need the following:

- A valid wireless network.
- A valid wireless LAN dongle inserted in the OmniScan X3 unit. The recommended model is the LM Technologies LM808-0406 or LM808-0407 depending on your region. Follow the instructions in the OmniScan X3 user's manual to know how to insert the wireless dongle.
- The device must be registered on the OSC and synchronized with it.
- The X3 RCS license must be activated in the OSC account by the administrator of the account.

NOTE

iPhone users: When setting up a Personal Hotspot, make sure that the phone's Device Name and Hotspot Password only use alphanumerical characters (letters and numbers).

10.2 Activation

The X3 RCS is installed automatically in the MXU software, but is not activated by default.

To activate the X3 RCS

1. Contact an Evident sales representative who will provide a PIN for activating the X3 RCS license once the device is registered.
2. Create or join an existing account on the OSC (an administrator creates the account and invites all other users). Go to www.olympus-ims.com and click **Cloud Log in** as shown in Figure 10-1 on page 227.

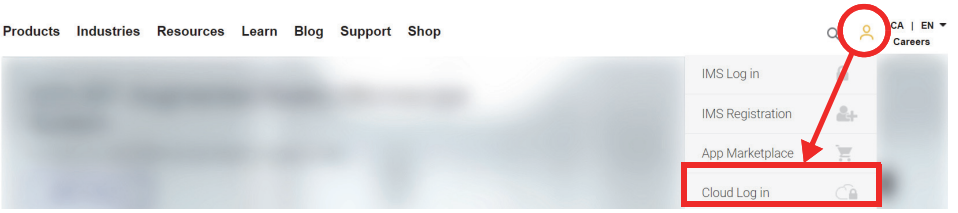


Figure 10-1 Cloud Log in

3. All devices that are going to use the X3 RCS must be registered in the account. On the OCS, go to the **My Devices** section and add an OmniScan X3 unit with its serial number. When entering the serial number on the OSC, you will get a PIN.
4. Boot the OmniScan X3, make sure it is connected to a wireless network by taping the **WIRELESS** button.
5. Enter the PIN in the **OSC CONNECT** menu in the **OmniScan X3 Launcher**. *This* **OSC CONNECT** *PIN is not the same as the X3 RCS license activation PIN.*
6. If the PIN is correctly entered, the device should now be registered on the OSC.
7. In **My Apps** section in the OSC, the administrator of the account can now add the X3 RCS, and activate it with the PIN provided by Evident.

10.3 X3 RCS Statuses

You can access the X3 RCS menu by tapping the RCS icon in the top bar. The RCS icon has four color statuses as shown in Figure 10-2 on page 227:

Remote Control Service Status	Control Appearance
RCS not available (no subscription or no Wi-Fi or no OSC connection)	RCS
RCS available, meeting not started	RCS
RCS available, meeting started	RCS
RCS available, meeting started, and X3 is remote controlled	RCS

Figure 10-2 X3 RCS four statuses

Tapping on the RCS icon opens the X3 RCS menu. If the service is available, the first thing to do is to tap **Start Meeting** to create a meeting directly on the OmniScan X3 unit (see Figure 10-3 on page 228).

Figure 10-3 RCS interface example

Table 91 RCS interface description

Interface item	Description
Stop Sharing/Share Screen	At the beginning of the meeting, the OmniScan X3 screen is automatically shared to all participants invited. Tap this button to stop sharing or to resume sharing. It is recommended to stop sharing before another collaborator shares its own screen.
QR code	Touch the QR code to enlarge it. Scan this code with your cell phone to join the meeting. The Zoom video application must be installed on your phone. Joining with a phone gives additional functions, such as audio and video communication and the ability to show something to others using the phone's camera.
Invite	To invite collaborators by email, enter an email address, then click on the **+** button. Repeat this to add more participants. When ready, tap **Send Invite**. If you made an error in the address, you can delete the address and add the participant again. Click **Update Invite** to send the invitation again.
End Meeting	When the user ends the meeting, everybody is forced out of the meeting.

While the meeting is started and the OmniScan X3 unit screen is shared, a green border is displayed around the screen.

10.4 Remote Control

Collaborators can annotate directly on the OmniScan X3 unit screen without requesting remote control.

Collaborators can also ask to remote control the OmniScan X3 unit. A message will pop, informing the user that someone is requesting remote control. To give control, the user must tap the RCS icon, and click **Accept**. The participant now has access to remote control the instrument. In addition to having access to all the MXU software interface, the collaborator can also virtually access the membrane key on the side of the OmniScan X3 unit screen (see Figure 10-4 on page 229).

Figure 10-4 OmniScan X3 unit shortcuts

NOTE

You can only request Remote Control from Zoom on a PC. Cell phones and tablets do not have access, but this is a limitation of the Zoom video application that is used to communicate with the OmniScan X3 unit, rather than a X3 RCS limitation.

10.5 Zoom Application

Zoom (www.zoom.us) is an application that can be installed on a cell phone, tablet, or PC. You can also use Zoom in a web browser. You can use all the zoom versions to join a meeting with the OmniScan X3 unit.

Some devices have more restrictions, for example it is not possible to remote control the OmniScan X3 unit from a cell phone.

From Zoom, you can:

- Join a meeting from the OmniScan X3 unit.
- Share audio and video with other participants (the user must use another device as audio and video are not active on the OmniScan X3 unit).
- Invite other participants. It can be easier to type email addresses from a laptop than from the OmniScan X3 unit touchscreen.
- Edit participant settings.
- Annotate the OmniScan X3 unit screen.
- Send reactions.
- Chat with other participants (not seen on the OmniScan X3 unit screen).
- Request remote control.

Invite Others From the Application

To invite other participants from Zoom, the user must first join the meeting created by the OmniScan X3 unit. Then, click **Participant**, and then click **Invite**. You can now share the invitation to this meeting with others (see Figure 10-5 on page 231).

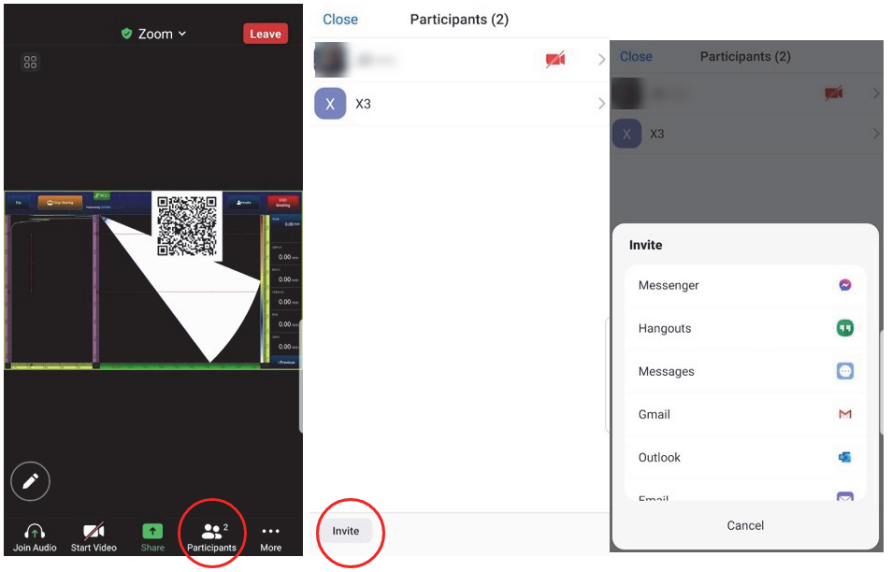


Figure 10-5 Invite other participant from the Zoom application

10.6 Typical Workflow

Once connected to Internet and the RCS application activated, the OmniScan X3 user can request help from collaborators.

To request help

1. Tap the RCS icon.
2. Click **Start Meeting**.
3. Tap **Invite**.
4. Add the email address, then click on **+**.
5. If needed, repeat step 4 for each participant to invite. When done, tap **Send Invite**.
6. Tap the QR code to enlarge it and scan it with your cell phone to get a link to join the meeting.
7. Wait for everyone to join the meeting. The OmniScan X3 user can use video and audio from their cell phone.

8. If a participant requests control, tap the RCS icon to give control.
9. When done, click **End Meeting** to leave and close the meeting. The RCS license can now be used by another OmniScan X3 user in the same OSC account.

List of Figures

List of Tables