TomoView
User's Manual

Software Version 2.10

DMTA-20029-01EN [U8778536] — Revision B

September 2022

EVIDENT CANADA, INC., 3415, Rue Pierre-Ardouin, Québec (QC) 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 2.10
Part number: DMTA-20029-01EN [U8778536]
Revision B
September 2022

Printed in Canada

Table of Contents

List of Abbreviations **11**

Important Information — Please Read Before Use **13**
Intended Use 13
Software Compatibility 13
Safety Symbols 14
Safety Signal Words 14
Note Signal Words 15
Warranty Information 15
Technical Support 16

Part One: Getting to Know TomoView **17**

Introduction **19**

1. Getting Started **21**
1.1 Minimum Computer Requirements 21
1.2 Compatible Data Acquisition Units 22
1.3 TomoView Editions 23
1.4 Security Hardware Key 28
1.5 Installing TomoView 30
1.6 About the Bootp Server 31
1.7 Starting a FOCUS LT to Connect to TomoView 34
1.8 Starting an OmniScan and Connecting to TomoView 36
1.9 Starting TomoView 41

2. TomoView User Interface **45**
2.1 TomoView User Interface and Software Principles 45
2.2 TomoView Interface 47

2.3 Main Window 49
2.4 Menu Bar 49
2.4.1 File Menu 50
2.4.2 Toolbars Menu 54
2.4.3 Part & Overlay Menu 55
2.4.4 View Menu 56
2.4.5 Layout Menu 58
2.4.6 Processing Menu 60
2.4.7 Tools Menu 62
2.4.8 Components 64
2.4.9 Window Menu 66
2.4.10 Help Menu 66
2.5 Toolbars 67
2.5.1 Component Toolbar 67
2.5.2 Document Toolbar 69
2.5.3 View Toolbars 70
2.6 The TomoView Manager Dialog Box 73
2.7 Document Windows 74
2.8 Layouts 75
2.9 The Status Bar 76
2.10 UT Settings Dialog Bar 79
2.11 Scan and Mechanical Settings Dialog Box 79
2.12 The View and Pane Properties 80
2.13 Docking Dialog Boxes 81

3. TomoView Concepts and Operational Mode 83
3.1 About the TomoView Modes 84
3.2 About Groups 85
3.3 About the Advanced Calculator 87
3.4 About Scan Sequences 87
3.5 About Probe Orientation Conventions 89
3.6 About Layouts 95
3.7 About Views 96
3.7.1 Data View Types 99
3.7.1.1 Basic Views 100
3.7.1.2 Volumetric Views 102
3.7.1.3 Scrolling Views 107
3.7.2 View Contextual Menu 109
3.8 About Readings and Information Groups 114
3.9 About Gates 115
3.10 UT Settings Dialog Box 117
3.10.1 General Tab 118

3.10.2 Gate Tab ... 123
3.10.3 TCG Tab ... 127
3.10.4 DGS tab ... 129
3.10.5 Digitizer Tab ... 129
3.10.6 Pulser/Receiver Tab ... 134
3.10.7 Probe Tab ... 137
3.10.8 Alarms Tab ... 141
3.10.9 I/O Tab ... 142
3.10.10 Transmitter Tab ... 145
3.10.11 Receiver Tab ... 146
3.11 Scan and Mechanical Settings Dialog Bar ... 148
3.11.1 Description of the Sequence tab ... 148
3.11.2 Scan Controls Tab ... 152
3.11.3 Encoders Tab ... 155
3.11.4 Options Tab ... 159
3.12 Scanner Parameters (Master Mode Only) ... 162
3.12.1 MCDU Control Tab ... 162
3.12.2 MCDU I/O Tab ... 168
3.13 View and Pane Properties Dialog Bar ... 172
3.13.1 Information Tab ... 172
3.13.2 Display Tab ... 174
3.13.3 Echo Dynamics Tab ... 183
3.13.4 Overlay Tab ... 186
3.13.5 Palette Tab ... 187
3.13.6 Data Source Tab ... 190
3.13.7 Parameters Tab ... 191
3.13.8 Units Tab ... 201
3.13.9 View Linking Tab ... 208
3.13.10 Rebounds Tab ... 208
3.13.11 Strip Tab ... 209
3.14 Preferences Dialog Box ... 210
3.14.1 General Settings Tab ... 211
3.14.2 Linking Tab ... 214
3.14.3 Colors Tabs ... 216
3.14.4 Folders Tab ... 217
3.15 About the Advanced Calculator ... 218
3.16 About the Expert Mode ... 220
3.17 About the Master and Slave Modes ... 220
3.18 File Formats ... 221

Part Two: Working with TomoView ... 225

4. Setup Creation 227
4.1 Working with Setup Files 227
4.1.1 Loading an .acq Setup File 227
4.1.2 Loading an SRD Setup File 228
4.1.3 Saving an .acq Setup File 229
4.1.4 Default Setup Files 230
4.2 Working with Groups 230
4.2.1 Adding and Configuring a Phased Array Group 231
4.2.2 Adding and Configuring a Conventional UT Group 239
4.2.3 Adding and Configuring a TOFD Group 242
4.2.4 Switching Between Groups 243
4.2.5 Renaming Groups 244
4.2.6 Deleting Groups 244
4.3 Calibrating a Phased Array Group 245
4.3.1 Calibrating the Beam Delays 245
4.3.2 Calibrating the Sensitivity 248
4.4 Calibrating a Conventional UT Group 249
4.5 Calibrating a TOFD Group in Setup Mode 251
4.6 Working with Overlay Drawings 252
4.7 Working with the TCG Curve 254
4.7.1 Constructing a TCG Curve for a Conventional UT Group 254
4.7.2 Displaying or Hiding the TCG Curve 257
4.7.3 Constructing a TCG Curve for a Phased Array Group 258
4.7.4 Importing/Exporting a TCG Curve 260
4.8 Working with Layouts 261
4.8.1 Applying a Template Layout File 261
4.8.2 Displaying or Changing the View in a Pane 263
4.8.3 Saving Layout Files and Defining New Template Layouts 263
4.9 Working with the DGS Curve 267
4.9.1 Configuring the DGS for a Conventional UT Group 267
4.9.2 Calibrating the DGS for a Conventional UT Group 268
4.9.3 Configuring the DGS for a Phased Array Group 272
4.9.4 Displaying DGS Readings 274
4.9.5 Changing the DGS Curve Colors 275
4.10 Working with the Readings 276
4.10.1 Reading Management 276
4.10.2 Reading Examples 277
4.10.3 Hiding or Showing Readings at the Top of a View 280
4.10.4 Customizing the Reading Color and Font 280
4.11 Working with Gates 282
4.11.1 Gate Adjustments 283
4.11.2 About Gate Synchronization 285

4.11.3 Gates in the Analysis Mode 285
4.11.4 Gates and Amplitude C-Scans 286
4.11.5 Gate Usage Example 287
4.12 Working with the Alarms 288
4.13 Working with the Inputs/Outputs 288
4.14 Working with the Firing Sequencer 289
4.15 Working with the Conditional A-scan 292

5. Performing Acquisitions 295
5.1 Working with the Different Scan Types 295
5.1.1 One-Line Scan Sequence 295
5.1.2 Free Running Sequence 297
5.1.3 Bidirectional Scan 297
5.1.4 Unidirectional Scan 300
5.1.5 Helicoidal Scan 302
5.1.6 Angular Sequence 304
5.1.7 Custom Sequence 306
5.2 Working with Encoders 307
5.2.1 About Encoder Types 307
5.2.2 Calibrating an Encoder 309
5.3 Working with the Position-Dependent Setup 311
5.3.1 About the Position-Dependent Setup Module 311
5.3.2 Position Dependent Setup Dialog Bar 312
5.3.3 Use of the Position-Dependent Setup Module 313
5.3.4 Limitations 314
5.4 Defining the Automatic Savings Options 315

6. Basic Analysis Using TomoView 317
6.1 Working with Data Files in TomoView 317
6.1.1 Opening a TomoView Data File 317
6.1.2 Importing OmniScan Data Files 319
6.1.3 Merging Data Files 322
6.2 Working with the Indication Table and Report Component 324
6.2.1 Adding an Indication to the Indication Table 324
6.2.2 Producing an Ultrasonic Inspection Report 328
6.2.3 Changing the Logo in the Inspection Report 330

7. Advanced Analysis Using TomoView 333
7.1 Working with the Thickness C-Scan Process 333
7.2 Working with the Volumetric Merge Process 335
7.2.1 Performing a Volumetric Merge 336

7.2.2 Performing an Automatic Volumetric Merge 342
7.2.3 Performing an Automatic Volumetric Merge by Group 342
7.3 Customizing Readings 342
7.4 Working with the Amplitude Drop Sizing Utility 342
7.4.1 Defining or Modifying Flaw Sizing Settings 343
7.4.2 Determining the Location and Dimensions of Flaw Indications 344
7.5 Working with the Gain Information Component 349
7.6 Working with the TOFD Manager Component 350
7.6.1 Description of the TOFD Manager Dialog Box 352
7.6.1.1 Analysis Tools Group Box 352
7.6.1.2 Lateral Wave Processing Group Box 354
7.6.1.3 Focalization Process Group Box 355
7.6.2 Performing TOFD Calibration in Analysis Mode 355
7.6.3 Performing Data Processing of TOFD Files 356
7.6.3.1 Lateral Wave Synchronization 356
7.6.3.2 Lateral Wave Removal 358
7.6.3.3 SAFT Processing 361
7.7 Working with the C-Scan Merge Process 363
7.8 Working with the SNR Analysis Component 368
7.8.1 Calculating the Area of an Indication Using SNR Analysis Utility 368
7.8.2 SNR Analysis Conventions 373
7.9 Working with the Soft C-Scan Process 375
7.10 Working with the Binarizer Component 378
7.11 Working with Color Palettes 383
7.11.1 Modifying Color Palettes 383
7.11.2 Optimizing a Color Palette for Corrosion Visualization 385
7.12 Working with the Image Analysis Tools 388
7.12.1 Measurement Cursors and Measures 391
7.12.1.1 Relationship Between Cursors 391
7.12.1.2 Moving a Cursor with the Mouse 392
7.12.1.3 Moving a Cursor with the Keyboard 392
7.12.1.4 Moving Cursors with a Fixed Gap 392
7.12.1.5 Creating a Link Between Cursors 393
7.12.1.6 Displaying Cursor Parameters 393
7.12.2 Zone Tool 394
7.12.2.1 Creating a Zone 395
7.12.2.2 Resizing a Zone 395
7.12.2.3 Moving a Zone 395
7.12.2.4 Hiding a Zone 395
7.12.2.5 Displaying Zone Parameters 396
7.12.3 3-D Cursor 397
7.12.3.1 Creating a 3-D Cursor 397

7.12.3.2 Moving a 3-D Cursor 397
7.12.3.3 Hiding a 3-D Cursor 397
7.12.3.4 Displaying 3-D Cursor Parameters 398
7.13 Working with the A-scan Resynchronization Process 398
7.14 Working with the Matrix Filters Process 400
7.15 Working with the Cylindrical Correction 401
7.15.1 Description of the Cylindrical Correction Tool 401
7.15.2 Case of a Skew Angle of 0° or 180° 404
7.15.3 Case of a Skew Angle of 90° or 270° 409
7.15.4 Using Cylindrical Corrections 415
7.15.5 Calibrating the Wedge 415
7.15.6 Analyzing Cylindrical Readings in TomoView 420
7.16 Working with the Export Data Utility 422
7.17 Using the Hysteresis Correction Module 424
7.18 Working with the Log-to-Linear Data Conversion Process 427

8. Shortcut Keys 429

9. Troubleshooting 435
9.1 TomoView Only Starts with the File and the Help Menus 435
9.2 About Firewalls and TomoView 435
9.3 Resolving Communication Problems Between the TomoView and the Acquisition Unit 436
9.4 Configuring the Network Adaptor 439
9.5 Handling Large Data Files with East Asian Languages 440
9.5.1 Setting English as the Default Input Language 441
9.5.2 Starting TomoView to Handle Large Data Files with East Asian Languages 443

10. Additional Features 445
10.1 Working with the Import PASS Files Utility 445
10.1.1 Importing and visualizing an .mnp File 445
10.1.2 Saving PASS Data in an .rdt File 447
10.1.3 Determining the Characteristics of an Ultrasonic Beam 448
10.2 Changing the Color of an Element 449
10.3 Configuring Links Between TomoView Elements 450
10.4 Working with the Excel Exchanges Process 452
10.5 Editing System Preferences 460
10.6 Working with the Screen Capture Utility 461
10.7 Working with the Beam Simulation Component 463
10.7.1 Enabling a Beam Simulation View 464

10.7.2 Displaying a Beam Simulation View 464
10.7.3 Description of the Beam Simulation Dialog Box 465
10.8 User Fields 471
10.9 Working with the Fast Fourier Transforms (FFT) Component 473

List of Figures 475

List of Tables 487

List of Abbreviations

BSC	beam simulation configuration
DHCP	dynamic host configuration protocol
FFT	Fast Fourier transform
FTP	file transfer protocol
IP	internet protocol
LAN	local area network
LW	lateral wave
MFC	Microsoft foundation class
ML	material loss
PA	phased array
PID	proportional integral derivative (controller)
PRF	pulse repetition frequency
RAM	random-access memory
RDTIFF	R/D Tech proprietary file format
SAFT	synthetic aperture focusing technique
SNR	signal-to-noise ratio
TOFD	time-of-flight diffraction
UAC	user account control
US	United States
USB	universal serial bus
UT	ultrasonic testing

Important Information — Please Read Before Use

Intended Use

TomoView software is designed to perform nondestructive inspections on industrial and commercial materials.

Software Compatibility

Refer to the following table to confirm that TomoView is compatible with the ancillary equipment being used.

Table 1 Evident Data Acquisition Units compatible with TomoView

Family	Models
MultiScan	MultiScan MS5800 (acquisition in UT mode only)
OmniScan	OmniScan MX OmniScan MX2 OmniScan iX (in Analysis mode only)
TomoView	FOCUS LT FOCUS LT Rackmounted

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.

 High voltage warning symbol:

This symbol is used to alert the user to potential electric shock hazards greater than 1,000 volts. All safety messages that follow this symbol shall be obeyed to avoid possible harm.

Safety Signal Words

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

 **DANGER**

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

 **WARNING**

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

CAUTION

The CAUTION signal word indicates a potentially hazardous situation. It calls attention to an operating procedure, practice, or the like, which, if not correctly performed or adhered to, could 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 safety symbols could appear in the documentation of the instrument:

IMPORTANT

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

NOTE

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

TIP

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

Warranty Information

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

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

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

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

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

Technical Support

Evident is firmly committed to providing the highest level of customer service and product support. If you experience any difficulties when using our product, or if it fails to operate as described in the documentation, first consult the user's manual, and then, if you are still in need of assistance, contact our After-Sales Service. To locate the nearest service center, visit the Service Centers page on the Evident Scientific Web site.

Part One: Getting to Know TomoView

Introduction

TomoView is powerful and versatile software for nondestructive ultrasonic testing (UT). TomoView features functions for the acquisition of ultrasonic signals, for real-time imaging of these signals, and for the analysis of previously acquired data files.

TomoView has many advanced functions and features to efficiently acquire and analyze data during UT inspections. An ideal tool for either on-site or laboratory work, TomoView runs under Microsoft Windows 7, Windows Vista or Windows XP Professional.

TomoView handles data files up to 2 GB in size for a 64-bit operating systems (OS) and up to 1 GB for a 32-bit OS, allowing you to perform complex inspections not possible with standalone inspection instruments.

TomoView offers a flexible graphical user interface allowing you to customize, save, and retrieve layouts containing various views, thus facilitating your work during UT inspections.

In addition to generic graphical features such as measurement cursors, zooming operations, and color-palette modifications, TomoView also provides automated tools to facilitate industrial standard compliant advanced data analysis. You can simply outline flawed areas and configure TomoView to automatically find and display the peak amplitude and the length of the flaw. The TomoView reporting functions allow you to include flaw information in an indication table, together with essential TomoView setup parameters.

Using TomoView, you can analyze previously acquired files or files that are being acquired. You can select and zoom on portions of a view, and perform flaw measurements, while the same file is still being filled with acquisition data.

1. Getting Started

This chapter contains information to allow you to quickly begin using TomoView.

1.1 Minimum Computer Requirements

The minimum computer requirements for TomoView software are:

- 2 GB RAM memory or higher (4 GB recommended for large data files and high-speed acquisitions)
- 100 GB free hard disk space for optimal performance in analysis and acquisition.
- 1280 × 1024 or higher display adaptor and monitor resolution
- One USB port for the HASP security hardware key
- One 100 Mbit/s network adaptor dedicated to the acquisition unit (for acquisition). The computer needs a second network adaptor if you want to simultaneously connect it to a local area network and to a data acquisition unit.
- A keyboard and a pointing device
- One of the following operating systems:
  - Microsoft Windows XP, XP Pro, XP Family Edition (Service Pack 2 or higher)
  - Microsoft Windows Vista Business, Home Basic, Enterprise, Home Premium, Starter, Ultimate but the user account control (UAC) is not supported
  - Microsoft Windows 7

IMPORTANT

You can install TomoView on a drive other than the drive used for Windows. In that case, the TomoView installer still requires 150 MB where Windows resides, to install the hardware security key driver, the Direct X updates, the Windows temporary installation, and the MFC (Microsoft Foundation Class) files.

NOTE

In this document, TomoView screen captures were taken on a computer running Windows 7. The look of the screen captures could differ if you are using TomoView on a computer running another operating system.
For most examples presented in this document, TomoView was arbitrarily configured to use the metric measurement units (see section 3.14.1 on page 211 to find how to change measurement units).

1.2 Compatible Data Acquisition Units

TomoView operates with Evident ultrasonic instruments to acquire data in the Inspection mode and to analyze data files created with these instruments. Table 2 on page 22 provides the list of the compatible instruments.

Table 2 Evident Data Acquisition Units compatible with TomoView

Family	Models
MultiScan	MultiScan MS5800 (acquisition in UT mode only)
OmniScan	OmniScan MX OmniScan MX2 OmniScan iX (in Analysis mode only)
TomoScan	FOCUS LT FOCUS LT Rackmount

1.3 TomoView Editions

TomoView is available in four editions:

Inspection

A full-featured mode providing all the inspection and analysis functions. With the TomoView **Inspection** edition license, you can also choose to start the other TomoView editions (see section 1.9 on page 41 for details).

Analysis

Provides all analysis functions but no inspection functions.

Lite Weld

Provides a set of analysis functions optimized for weld applications (see Figure 1-1 on page 24):

- Conventional UT (pulse-echo and TOFD) analysis based on the B-scan
- Phased array angle beam analysis based on the S-scan
- Analysis of one-line scan inspections

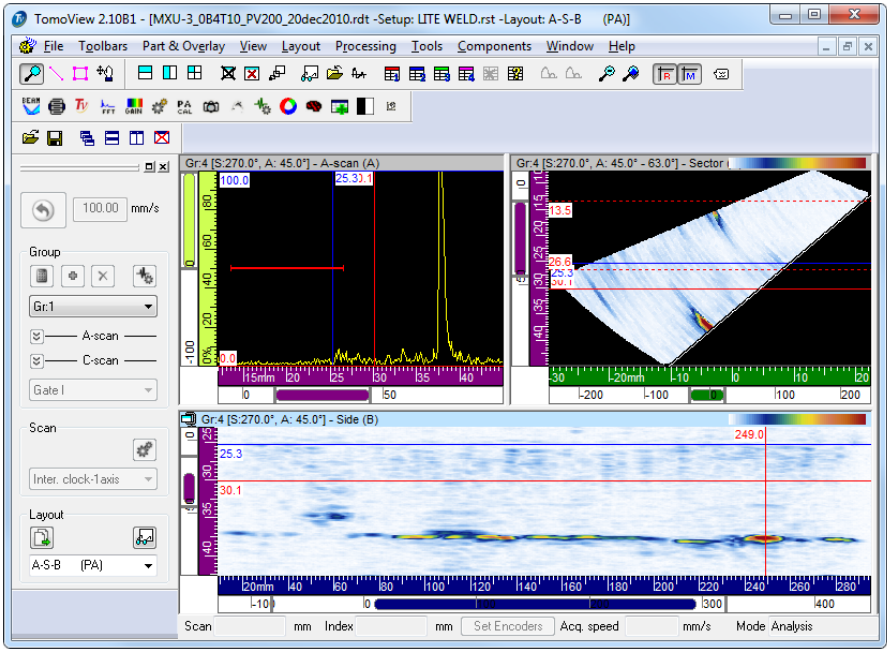


Figure 1-1 TomoView Lite Weld edition display example

Lite Aero

Provides a set of analysis functions optimized for aerospace applications (see Figure 1-2 on page 25):

- Conventional UT analysis based on the C-scan
- Phased array 0-degree analysis based on the C-scan
- Analysis of raster scan inspection

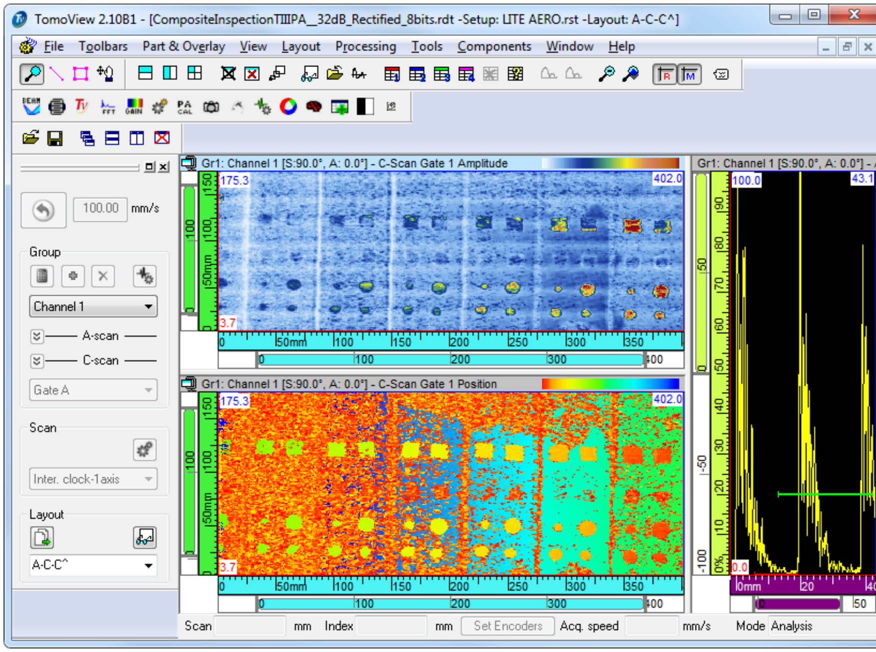


Figure 1-2 TomoView Lite Aero edition display example

TomoViewer

A free viewer that allows you to visualize TomoView data files.

At startup, you need to select which TomoView edition you want to run (see Figure 1-3 on page 26). Table 3 on page 26 presents the features available for each edition.

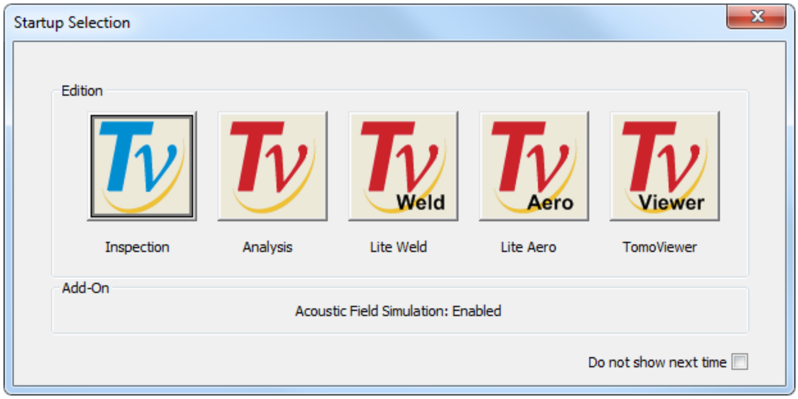


Figure 1-3 The Startup Selection dialog box

Table 3 TomoView edition feature comparison

Features	TomoVIEWER	TomoView			
		LITE Weld	LITE Aero	Analysis	Inspection
Ability to switch between metric and US Customary Units	✓	✓	✓	✓	✓
Zoom in/out	✓	✓	✓	✓	✓
Ability to import OmniScan data files	✓	✓	✓	✓	✓
Software gain adjustment	✓	✓	✓	✓	✓
Ability to view conventional UT groups	✓	✓	✓	✓	✓
Ability to view phased array groups	✓	✓	✓	✓	✓
Predefined display layouts	✓	✓	✓	✓	✓
Color contrast adjustment	✓	✓	✓	✓	✓
Ability to view TOFD groups	✓	✓		✓	✓
Volumetric Merge tool (automatic or manual)	✓	✓		✓	✓
Ability to view indication table	✓	✓	✓	✓	✓
Ability to preview and print reports	✓	✓	✓	✓	✓
Ability to modify/create color palette		✓	✓	✓	✓
Built-in report generator		✓	✓	✓	✓
Ability to display rebounds (skips)		✓	✓	✓	✓
Ability to save custom layouts		✓	✓	✓	✓
Ability to display custom weld overlay (.dxf)		✓	✓	✓	✓

Table 3 TomoView edition feature comparison *(continued)*

Features	TomoVIEWER	TomoView			
		LITE Weld	LITE Aero	Analysis	Inspection
Sound axis calibration		✓	✓	✓	✓
Ability to display and edit indication table		✓	✓	✓	✓
Selectable information groups (readings)		✓	✓	✓	✓
C-Scan Merge tool		✓	✓	✓	✓
Zone tool for statistical measurements		✓	✓	✓	✓
Data-File Merge tool		✓	✓	✓	✓
Off-line gate adjustment		✓	✓	✓	✓
Ability to add/modify/delete indication		✓	✓	✓	✓
Multiple-group combined display		✓		✓	✓
Phased array and TOFD combined display		✓		✓	✓
Predefined weld overlay display		✓		✓	✓
Ability to add binarized C-scan images			✓	✓	✓
SNR measurement on C-scans			✓	✓	✓
Extra software C-scans		✓	✓	✓	✓
Off-line A-scan resynchronization			✓	✓	✓
TOFD manager (LW removal, resynchronization)				✓	✓
Ability to modify probe offsets and skew angles				✓	✓
Ray tracing				✓	✓
Ability to open multiple files simultaneously				✓	✓
Ability to export data group in .txt files				✓	✓
Layout creation				✓	✓
3-D cursor				✓	✓
FFT calculation				✓	✓
Hysteresis correction				✓	✓
Log to linear conversion				✓	✓
Microsoft Excel exchange				✓	✓
Ability to import PASS files				✓	✓
Data acquisition capabilities					✓

The **About TomoView** dialog box, accessible by selecting **Help > About** on the menu, reports which edition of TomoView is currently running (see Figure 1-4 on page 28).

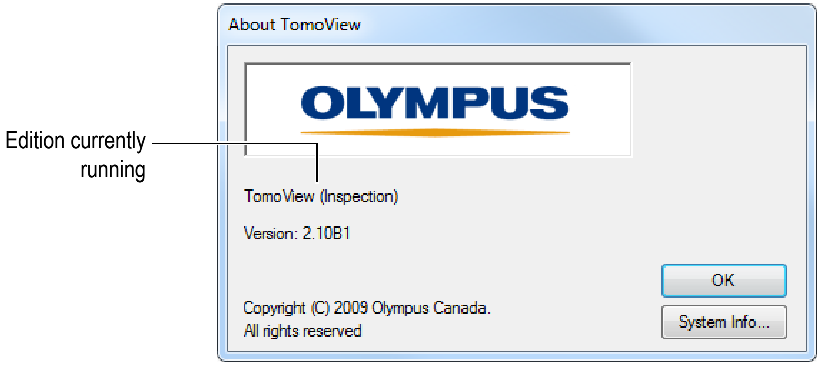


Figure 1-4 The About TomoView dialog box indicating the TomoView edition

1.4 Security Hardware Key

TomoView needs to detect an HASP security hardware key connected to the computer to operate. The HASP security USB hardware key supplied with your TomoView copy contains the authorization code needed to operate the TomoView edition that you purchased.

Before starting TomoView, connect the HASP hardware key to the USB port of your computer. You can still use a parallel port key from a previous TomoView version (see Figure 1-5 on page 28) as long as it contains authorization codes to run TomoView 2.10.

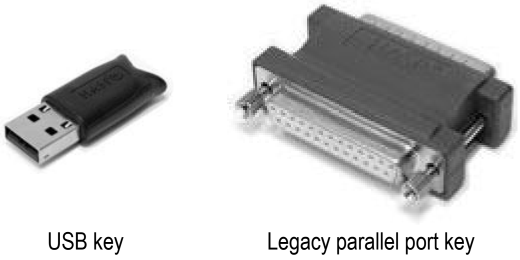


Figure 1-5 The HASP security hardware key models

When you start TomoView with no security hardware key connected to the computer, TomoViewer is the only enabled edition in the **Startup Selection** dialog box (see Figure 1-6 on page 29).

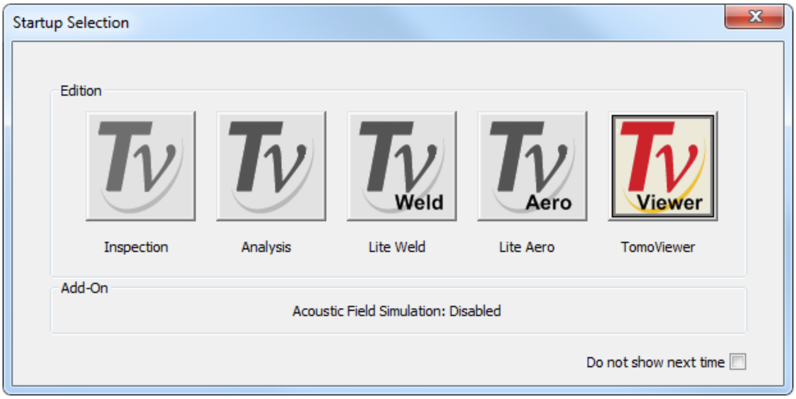


Figure 1-6 The Startup Selection dialog box with no security hardware key

If you disconnect the security hardware key while TomoView is running, the message shown in Figure 1-7 on page 29 appears 30 seconds later. After clicking **OK**, when needed, TomoView proposes to save unsaved data, and then closes.

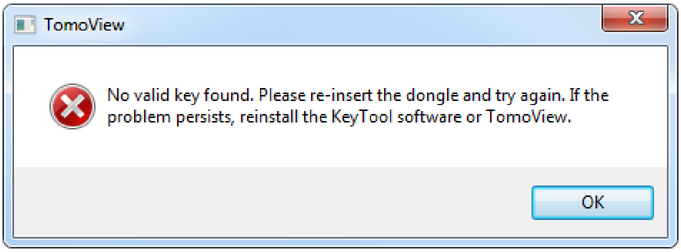


Figure 1-7 Missing hardware key message box

When you purchase both the Inspection and Analysis editions of TomoView, two security hardware keys are delivered, one for each edition.

1.5 Installing TomoView

The installation of TomoView is simple.

To install TomoView

1. On the computer on which you want to install TomoView, log on with a user account that has administrator rights.
2. Run the TomoView installer program from the Evident distribution disk.
3. Follow the on-screen TomoView installer wizard steps.
   The wizard installs TomoView, the Advanced Calculator, and the Bootp Server.
4. You need to disable the sleep mode on the computer to prevent a connection loss with the data acquisition unit:
   a) For Windows 7, on the Windows taskbar, select **Start > Control Panel > Hardware and Sound > Power Options**, and then click **Change Plan Settings.**
   b) In the pop-up menu that appears, set **Turn off the display** and **Put the computer to sleep** both to **Never.** (see Figure 1-8 on page 30).
   c) Click **Save Changes**.

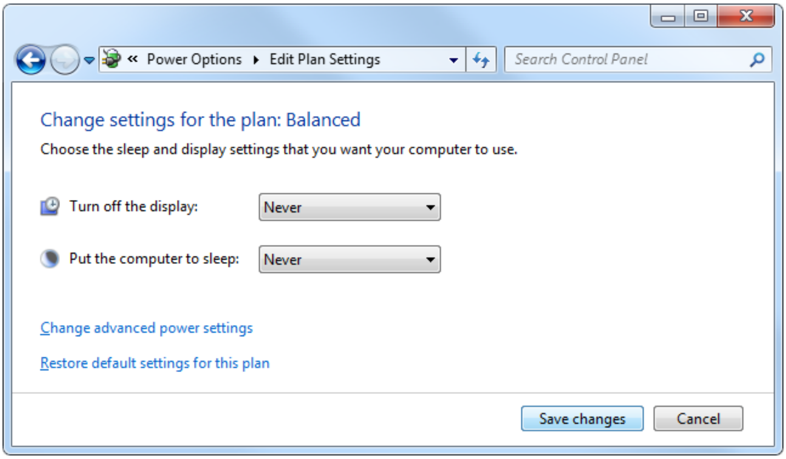


Figure 1-8 Configuring power options for Windows 7

OR

- For Windows XP, on the Windows taskbar, select **Start > Control Panel > Power Options**, and then select **Never** for both **System standby** and **System hibernates** (see Figure 1-9 on page 31).

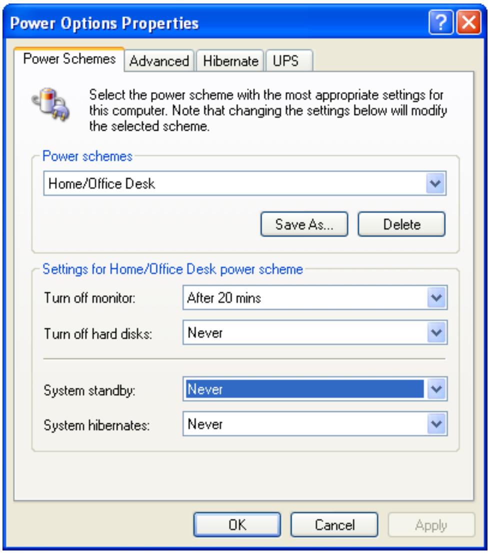


Figure 1-9 Configuring power options for Windows XP

NOTE

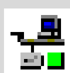
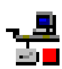
The TomoView installer configures the Windows firewall to allow the communication between the acquisition unit and the TomoView and Bootp Server programs. If you use a third-party firewall on the computer running TomoView, refer to section 9.2 on page 435 for configuration information.

1.6 About the Bootp Server

The TomoView installer automatically installs the Bootp Server. The role of the Bootp Server is to provide a valid IP address to a starting Evident data acquisition unit connected to the computer.

The Bootp Server is configured to automatically to start with Windows. The Bootp Server Monitor icon () appears in the Notification Area of the Windows taskbar (see Figure 1-10 on page 32). The Bootp Server Monitor icon includes an indicator showing the state of the server (see Table 4 on page 32).

Table 4 Bootp Server Monitor states

Icon	Bootp Server state
	The Bootp Server is operating normally with no active pod.
	The Bootp Server is operating normally with at least one active pod.
	The Bootp Server is not operating normally. Refer to section 9.3 on page 436 for troubleshooting information.

A tooltip appears when you bring the mouse pointer over the Bootp Server Monitor icon (see Figure 1-10 on page 32).

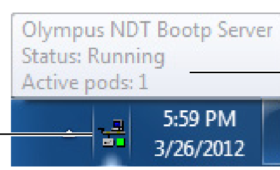


Figure 1-10 The Bootp Server Monitor icon and tooltip

The Bootp Server contextual menu appears when you right-click the Bootp Server Monitor icon (see Figure 1-11 on page 33).

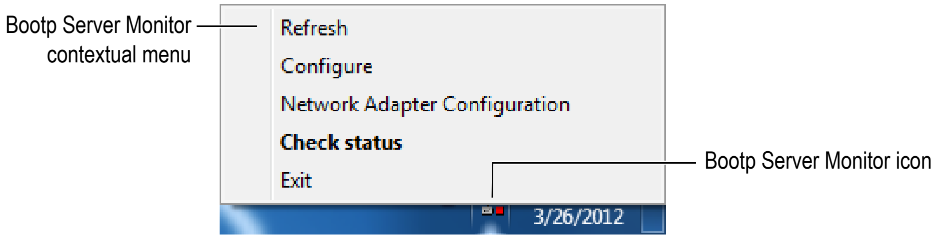


Figure 1-11 The Bootp Server Monitor icon and contextual menu

You can use the contextual menu commands to configure and diagnose the operation of the Bootp Server. For example, when the Bootp Server Monitor icon includes a red indicator (error condition), click **Check Status** in the contextual menu. The **Bootp status** dialog box opens, displaying diagnostic information (see Figure 1-12 on page 33). Refer to section 9.3 on page 436 for more troubleshooting information.

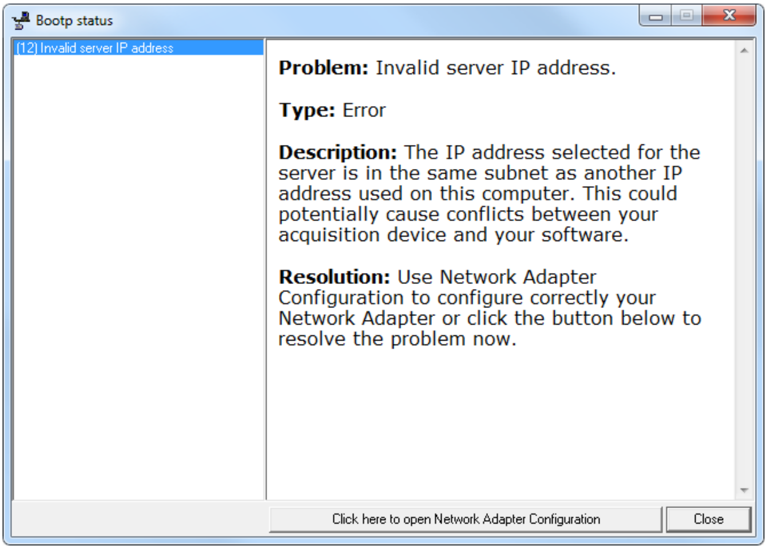


Figure 1-12 Bootp status dialog box content example

1.7 Starting a FOCUS LT to Connect to TomoView

TomoView can acquire ultrasonic data using a FOCUS LT or a FOCUS LT Rackmount unit.

To start a FOCUS LT to connect to TomoView

1. Start the computer but do not start TomoView yet.
2. Ensure that the power is turned off on the FOCUS LT unit.
3. Connect a crossover Ethernet cable to the network adaptor of the FOCUS LT unit.
4. Connect the other end of the crossover Ethernet cable to the network adaptor dedicated to the FOCUS LT on the computer.

IMPORTANT

When your computer has more than one network adaptor, ensure to connect the crossover Ethernet cable to the network adaptor configured to connect to the acquisition unit.

---

5. Turn on the data acquisition unit:
   - For a FOCUS LT, press the power button on the front panel (Figure 1-13 on page 35), and then wait for the STBY indicator to stop blinking, indicating that the unit is ready to operate.

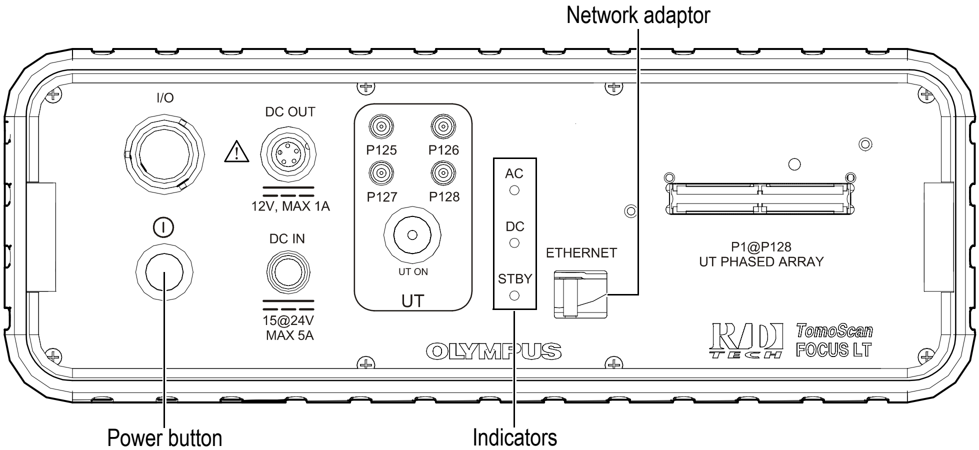


Figure 1-13 The FOCUS LT front panel

OR

For a FOCUS LT Rackmount, press the power button on the front panel (Figure 1-14 on page 35), and then wait for the READY indicator to stop blinking and to turn green, indicating that the unit is ready to operate.

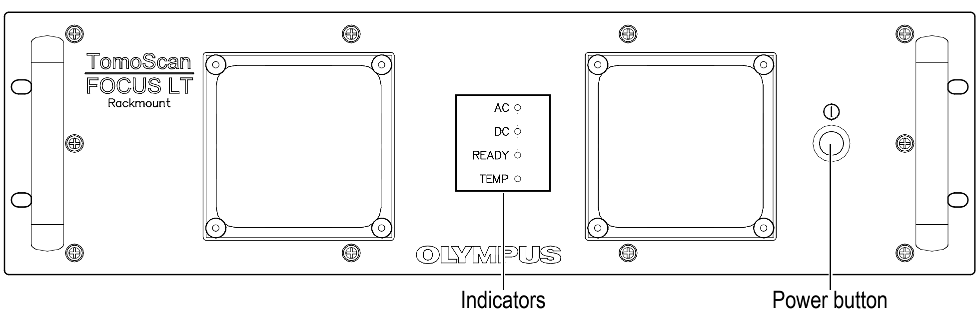


Figure 1-14 The FOCUS LT Rackmount front panel

6. On the computer, bring the pointer over the Bootp Server Monitor icon () in the Navigation Area of the Windows taskbar.
   A tooltip appears (see Figure 1-15 on page 36).

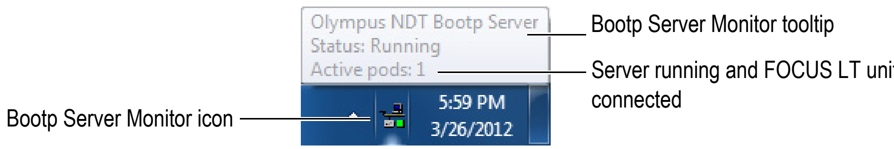


Figure 1-15 The Bootp Server icon when ready for operation

7. From the tooltip information, confirm that the Bootp Server is running and that it recognizes the FOCUS LT.
8. Start TomoView (see section 1.9 on page 41).

1.8 Starting an OmniScan and Connecting to TomoView

TomoView can acquire ultrasonic data using an OmniScan MX or MX2 unit that runs the TomoView Control program.

When you use TomoView to acquire ultrasonic data with an OmniScan unit, you need to start the OmniScan before starting TomoView.

To start an OmniScan to connect to TomoView

1. Start the computer but do not start TomoView yet.
2. Ensure that the power is turned off on the OmniScan unit.
3. Connect a crossover Ethernet cable to the OmniScan network adaptor (see Figure 1-16 on page 37).

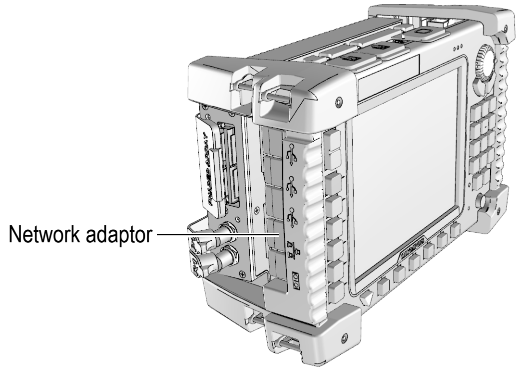


Figure 1-16 The network adaptor on the OmniScan MX

4. Connect the other end of the crossover Ethernet cable to the appropriate network adaptor on the computer.

IMPORTANT

When your computer has more than one network adaptor, ensure to connect the crossover Ethernet cable to the network adaptor configured to connect to the acquisition unit.

5. On the OmniScan unit (see Figure 1-17 on page 38):
   a) Ensure that a memory card containing the TomoView Control application is inserted into its slot on the right side of the unit.
   b) On the front panel, press the green Power key to turn on the power of the OmniScan.
   The **OmniScan** initial screen appears (see Figure 1-18 on page 38).

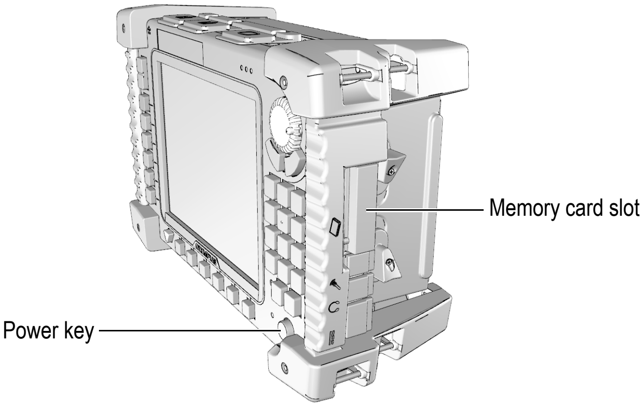


Figure 1-17 The OmniScan MX unit

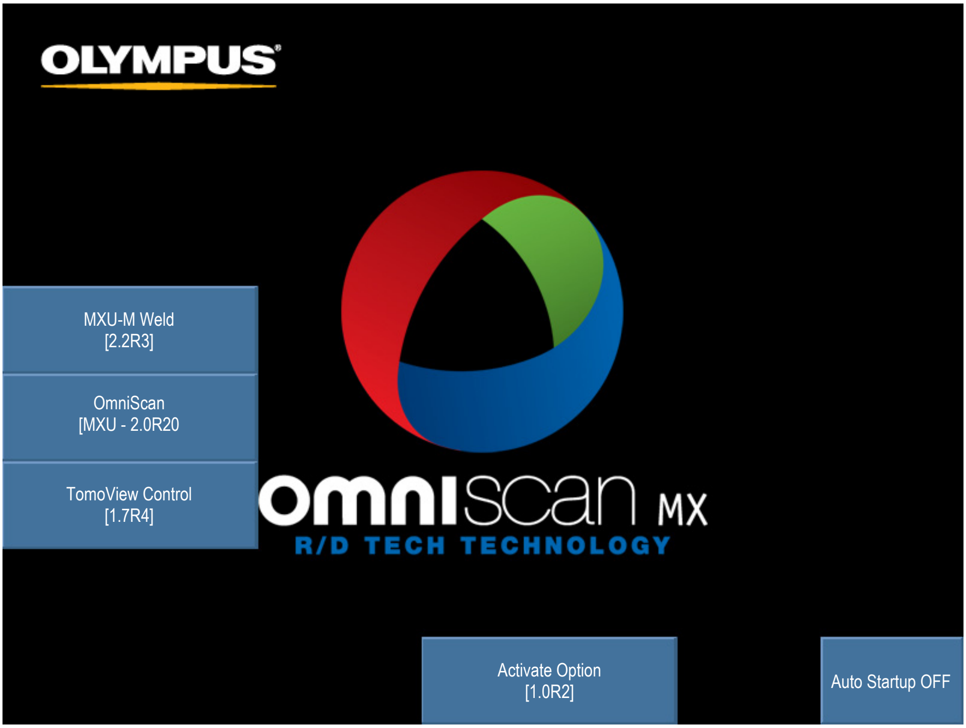


Figure 1-18 Example of the OmniScan initial screen

c) Choose **TomoView Control**.

d) If a dialog box appears requesting an option key number, type your new option key number (see Figure 1-19 on page 39) to activate the TomoView Control program on the current module.
If you do not have an option key number, and you want to upgrade your system, please contact your local Evident sales representative.

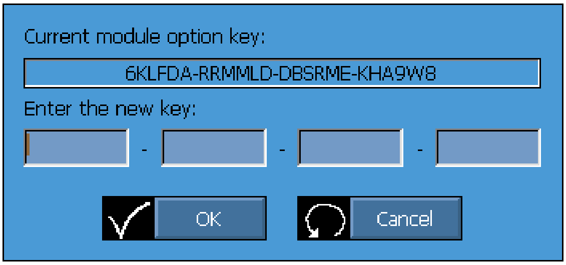


Figure 1-19 The dialog box to type a new option key number

The application appears on the OmniScan screen showing a log of successful operations (Figure 1-20 on page 40).

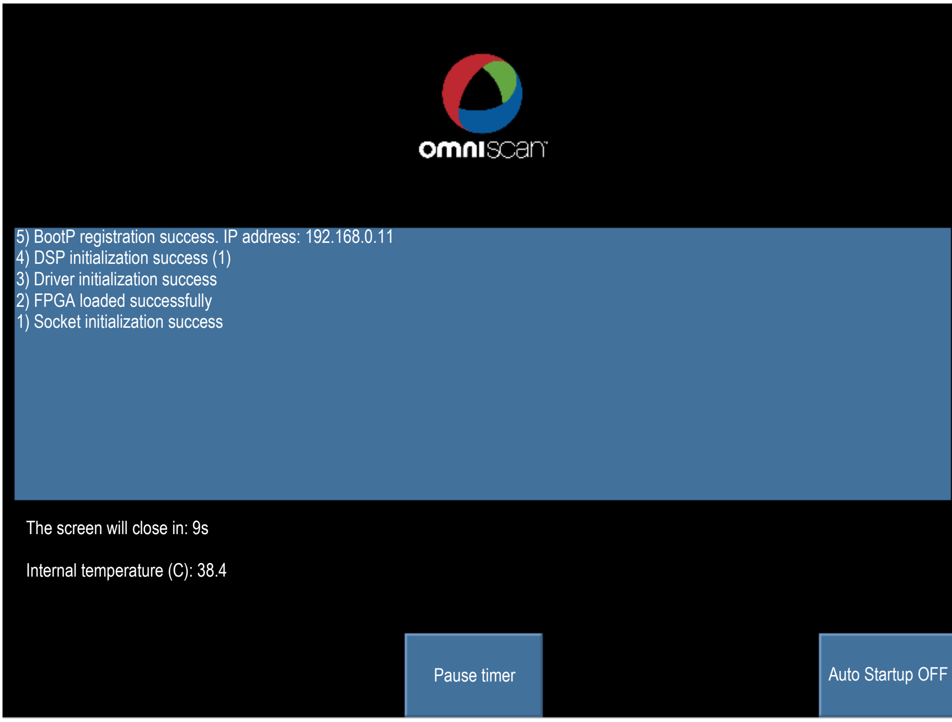


Figure 1-20 Example of the TomoView Control application screen

If an operation fails, a logged message appears in red. Review the message and take the necessary action to correct the problem.

NOTE

By default, the screen saver turns the screen off after 10 seconds. Press any function or parameter key to reactivate the screen. Select **Pause timer** to stop the screen saver count down.

6. On the computer, bring the pointer over the Bootp Server Monitor icon () in the Navigation Area of the Windows taskbar.
   A tooltip appears (see Figure 1-21 on page 41).

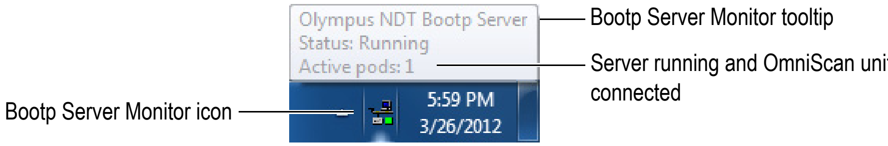


Figure 1-21 The Bootp Server icon when ready for operation

7. From the tooltip information, confirm that the Bootp Server is running and that it recognizes the OmniScan.
8. Start TomoView (see section 1.9 on page 41).

1.9 Starting TomoView

TomoView operates with or without an acquisition unit. When you use TomoView without an acquisition unit, you can only perform analysis tasks on existing data.

To start TomoView

1. Connect the security hardware key to the appropriate port of the computer. TomoView needs to detect the security hardware key to operate, with or without being connected to an acquisition unit.

2. Turn on the computer and wait for Windows to complete its starting process. Do not start TomoView yet.

IMPORTANT

The computer must be operational before you turn the acquisition unit power on, as the acquisition unit gets its IP address from the computer.

---

3. Connect the data acquisition unit to the appropriate network adaptor of the computer, and then start it (see section 1.8 on page 36 for an OmniScan and section 1.7 on page 34 for a FOCUS LT).
4. Start TomoView by:

   a) Double-clicking the TomoView icon  on the Windows desktop.

   OR

Clicking **Start > All Programs > Evident NDT > TomoView 2.10** on the Windows taskbar.

5. In the **Startup Selection** dialog box that appears (see Figure 1-22 on page 42), click the button associated with the desired TomoView edition.

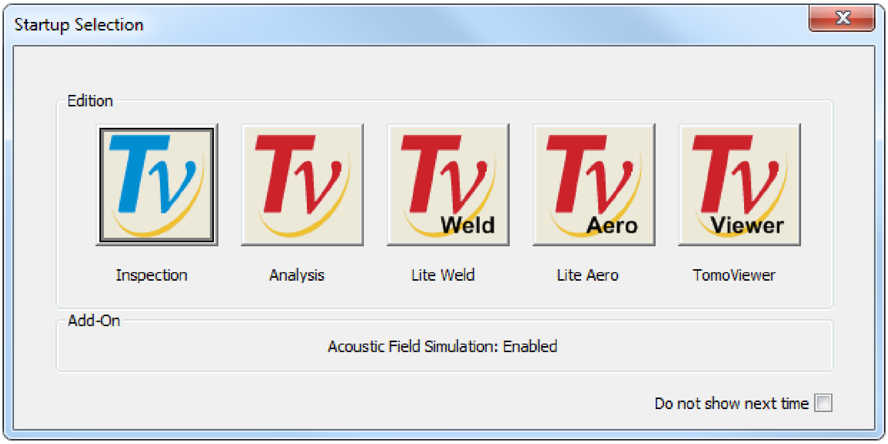


Figure 1-22 The Startup Selection dialog box

TIP

If you do not want the **Startup Selection** dialog box to appear each time you start TomoView, select the **Do not show next time** check box.

To reactivate the **Startup Selection** dialog box, select **File > Preferences > General Settings** tab in TomoView, and then deselect **Startup Selection** in the **Dialog Bypass** group box.

The TomoView splash screen appears briefly indicating that the application is starting up.

6. When you start the **Inspection** edition, the **Select Device Configuration** dialog box appears (see Figure 1-23 on page 43). In the dialog box, do one of the following:

   - Select the acquisition unit or units that you want to include in your configuration, and then click **OK**.

NOTE

With its multipod capability, TomoView can acquire data from up to three FOCUS LT units in parallel, offering up to a threefold increase in acquisition speed.

OR

When your computer is not connected to an acquisition unit, click **Cancel** to use TomoView only in the Analysis mode. In this case, the Setup and Inspection modes will be unavailable.

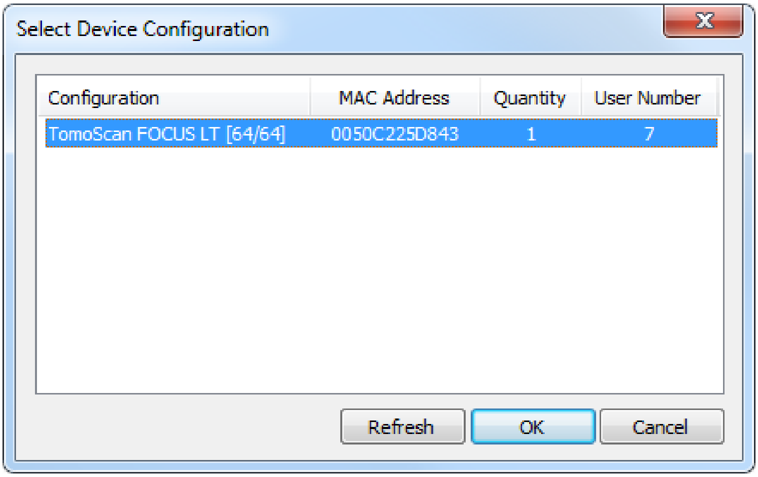


Figure 1-23 Example of the Select Device Configuration dialog box

NOTE

The list in the **Select Device Configuration** dialog box is empty when TomoView fails to detect hardware devices either because supported acquisition units are not connected, are not turned on, or are not correctly installed. Refer to section 9.3 on page 436 for troubleshooting information.

7. In the **Configuration Selection** dialog box that appears (see Figure 1-24 on page 44), select one of the following setup loading options, and then click **OK**.

Open the last configuration

Select this option to load the last configuration used, whose name is indicated in the box. By default, the box indicates the name of the default configuration (Default_PA.acq, Default_UT.acq and more).

Open an existing configuration

Select this option to access the **Open** dialog box. You can use this dialog box to browse through the folders and choose a configuration file (file name extension .acq).

Create a new configuration

Select this option to start a new configuration from a default configuration.

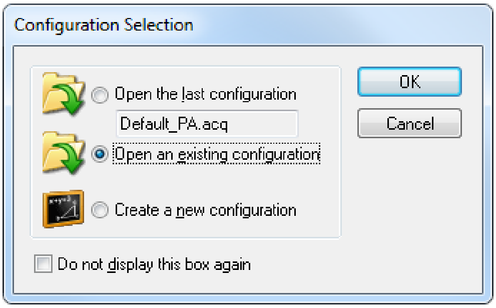


Figure 1-24 The Configuration Selection dialog box

NOTE

A configuration file (.acq) is a complete description of the TomoView workspace. The file includes the acquisition unit hardware setup and the TomoView layout environment.

If you click **Cancel**, the default configuration loads (Default_PA.acq for a phased array instrument or Default_UT.acq for a UT conventional instrument).

Once the selected setup is loaded, the TomoView window appears.

2. TomoView User Interface

2.1 TomoView User Interface and Software Principles

TomoView provides features needed to perform nondestructive inspections based on ultrasonic technology. It combines setup, inspection, and analysis functions in one software package. TomoView can also be used in a stand-alone mode to analyze previously acquired data.

The TomoView user interface (see Figure 2-1 on page 46) provides toolbars and menus for quick access to main commands. Using TomoView, you can conveniently present data imaging in multiple simultaneous views, such as in the example in Figure 2-1 on page 46. In this example, one document window is divided into four splitter windows. Each splitter window contains one or more (in the case of the upper right window) panes. Each pane contains one data view.

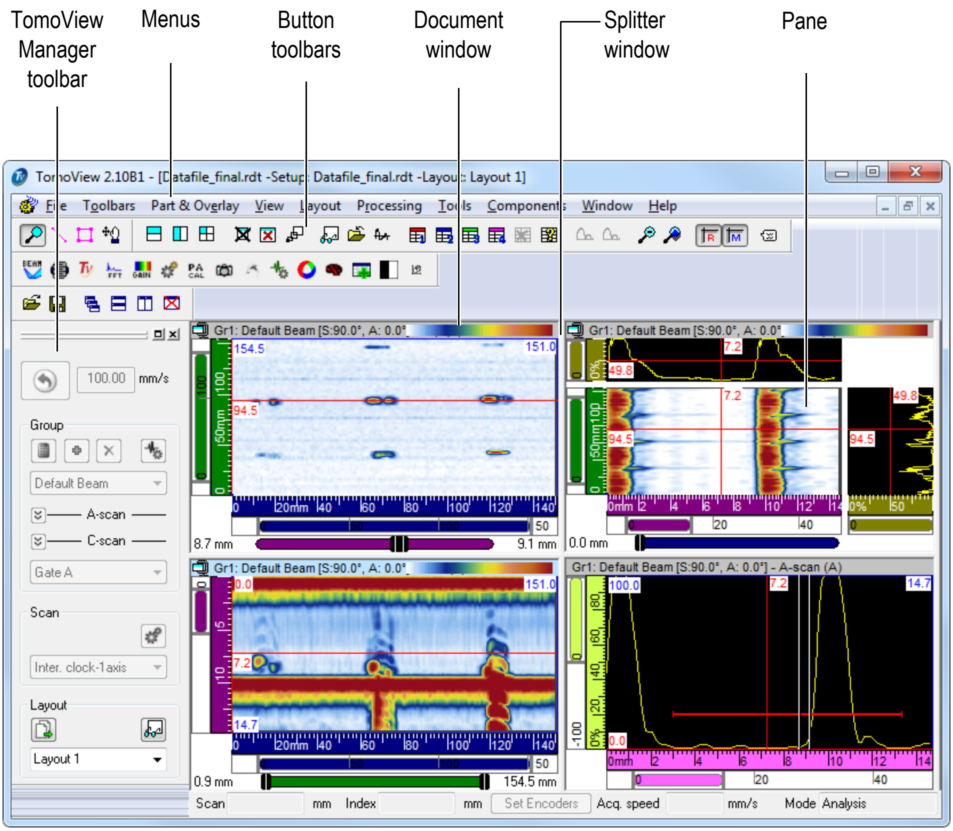


Figure 2-1 TomoView user interface example

Document windows

In TomoView, a document is related to the concept of an "inspection recording" and the associated data shown in a document window. You can use the application to consult and/or edit many document windows simultaneously.

Splitter windows

You can subdivide a document window into splitter windows and panes. Splitter windows have the advantage of being easily resized while maintaining the "tiled" nature of the various panes in the window.

Window panes

The window *pane* is each of the nonoverlapping areas in a splitter window. Document windows can be split into any number of panes. Each pane can be user defined to contain any of the possible view types generated by TomoView. For further information concerning the different types of views, see section 3.7.1 on page 99.

In TomoView, panes also offer flexibility for the location of readings: cursor positions, statistics, scale settings, etc. These fields can be dynamically placed and customized within any view. For further information concerning the readings, see section 4.10 on page 276.

Layout

A layout is a complete set of display-related settings. You can save and load a layout to quickly return to a desired view configuration. Layout configurations are included in data files so that recorded data can be viewed through the original layouts that existed at data-recording time, or through the current system layouts.

Template layouts are also provided as a reference for the most common inspection types. A menu on the TomoView Manager toolbar provides quick access to existing template layouts installed in TomoView (see section 4.8 on page 261 for more information on using layouts).

Setup

A setup file is a complete set of TomoView settings and contains the settings that an operator can access while using the TomoView interface. Setup files may contain one or more layouts, and files can be saved and restored on demand.

Normally, a setup represents a procedure for a specific application that is to be performed with the equipment.

2.2 TomoView Interface

This section provides a description of the TomoView interface. Figure 3-1 on page 83 shows an example of a data file representation and identifies the main TomoView interface elements.

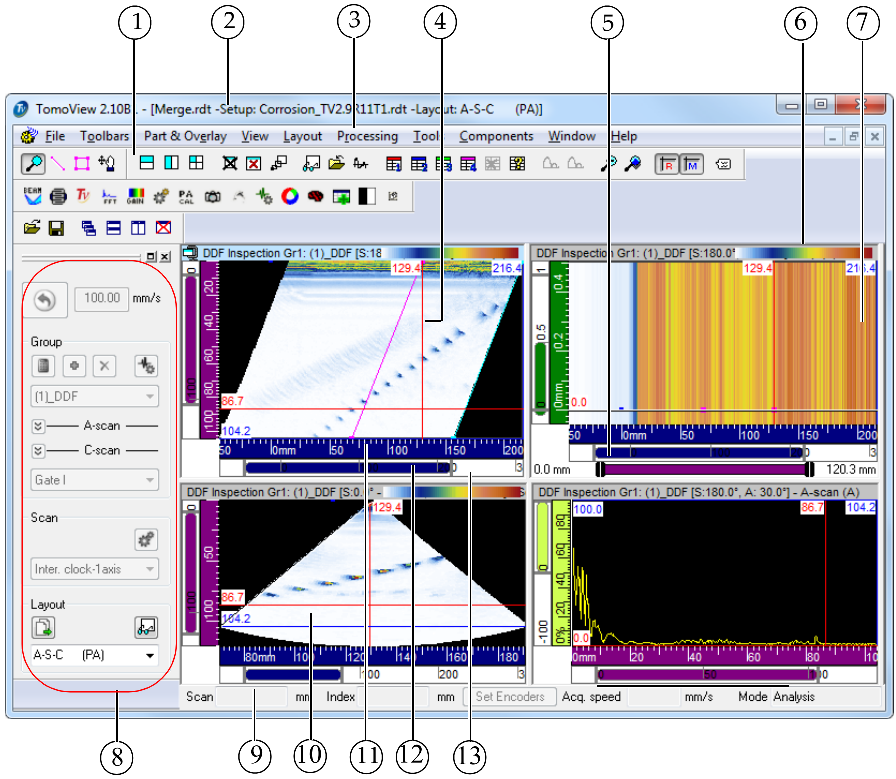


Figure 2-2 The TomoView interface

The main TomoView interface elements shown in Figure 2-2 on page 48 are the following:

1. Toolbars
2. Title bar of the main window
3. Menu bar
4. Reference cursor (red) with label
5. Title bar of the active pane
6. Color palette
7. Pane (data display area)
8. **TomoView Manager** toolbar
9. Status bar
10. Measurement cursor (blue) with label
11. Ruler
12. Zoom bar
13. Gate selector

2.3 Main Window

The main window (see Figure 2-2 on page 48) contains the following elements:

Title bar

Displays the program icon, the program identification, and the name of the active file, as well as control buttons on the right end of the bar.

Menu bar

Displays the names of the various menus.

Toolbar

Displays command buttons, which can be clicked to carry out various commands.

Document windows

There is a document window, or shell window, for each file opened. Document windows provide a primary view of file content. The windows can be overlapping, in which case the topmost window is the active one. The document windows can also be arranged as tiles or in a cascade by means of **Window** menu commands. The active window is always identified by a highlighted title bar.

2.4 Menu Bar

This section describes the TomoView menu bar. The menu bar (see Figure 2-3 on page 49) is the horizontal bar displayed at the top of the main window, below the title bar, and provides access to commands. To open a menu, click the menu name. To activate a menu command, click its name in the menu, or type the keyboard shortcut if there is one indicated.

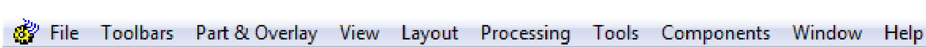


Figure 2-3 The Menu bar

The following subsections describe the commands available in each menu. Note that, depending on the operation mode and the type of hardware connected, the content of the menus could differ from what is presented in this manual.

2.4.1 File Menu

The **File** menu (see Figure 2-4 on page 50) contains the commands that manage data files and the general preferences of the software.

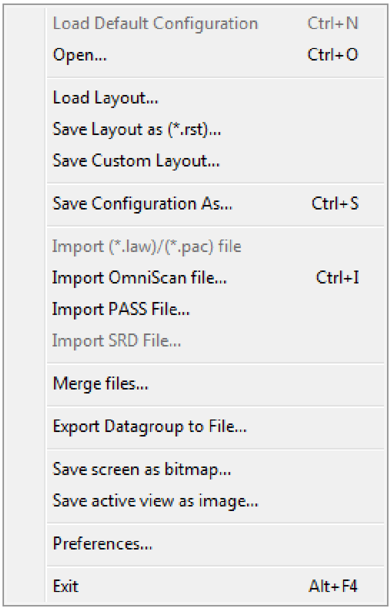


Figure 2-4 The File menu

The **File** menu commands are the following:

Load Default Configuration

Loads the default configuration used.

Open

Opens a standard **Open** dialog box, from which you can select and load a setup or a data file.

Load Layout

Opens a standard **Open** dialog box, from which you can select and open a layout file.

Save Layout As (*.rst)

Opens a **Save As** dialog box, from which you can save the current layout and define it as the default layout.

Save Custom Layout

Opens a **Save Custom Layout** dialog box (see Figure 2-5 on page 51), from which you provide a name for the current layout and save it as one of the five custom layouts.

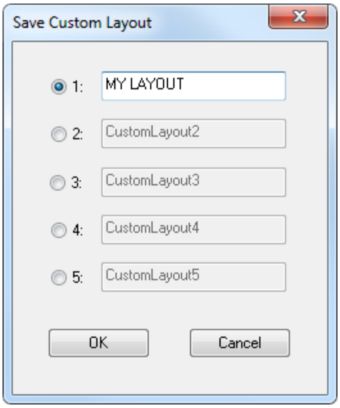


Figure 2-5 The Save Custom Layout dialog box

Save Configuration As

Opens a standard **Save As** dialog box, from which you can browse through folders, and save the active setup.

Import (*.law)/(*.pac) file

Opens a dialog box (see Figure 2-6 on page 52) from which you can select and load a *.law or a *.pac file used to configure the beam parameters for either the current Group, or a new one.

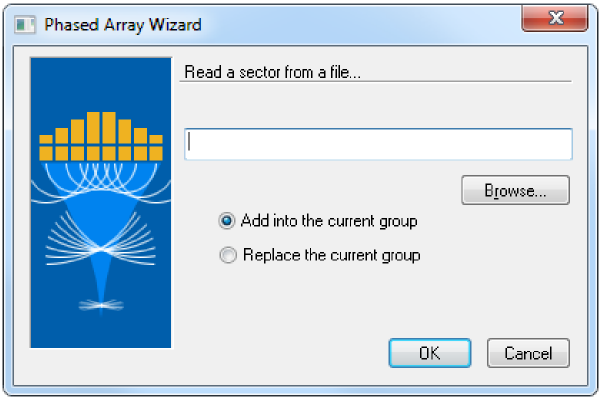


Figure 2-6 Importing .law or .pac files dialog box

Import OmniScan File

Opens a standard **Open** dialog box, from which you can select and load an OmniScan data file. This menu item is equivalent to the **OmniScan File Converter** button on the **Main** toolbar (see Table 5 on page 68 for details).

Import PASS File

Opens a standard **Open** dialog box, from which you can select and load a PASS file. PASS files are used to simulate the beam propagation inside various types of materials and geometries. See section 10.1 on page 445 for more information.

Import SRD File

Opens a standard **Open** dialog box, from which you can select and load an SRD file. SRD are legacy setup files used with older acquisition units. See section 6.1.3 on page 322 for more information.

Merge Files

Opens the dialog box shown in Figure 2-7 on page 53, from which different data files can be selected and merged into a single data file. See section 6.1.3 on page 322 for more information.

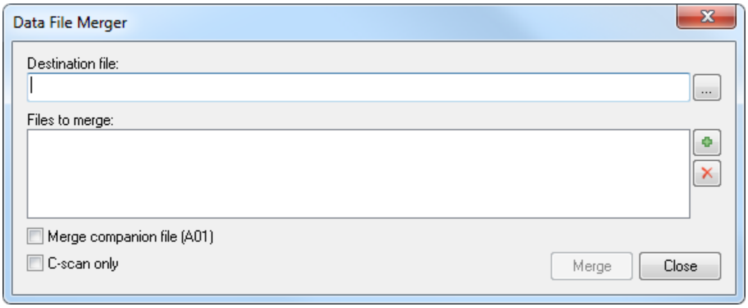


Figure 2-7 Data File Merge dialog box

Export Datagroup to File

Opens the dialog box shown in Figure 2-8 on page 53, which can be used to export an A-scan or a C-Scan to a *.txt file. See section 7.16 on page 422 for more information.

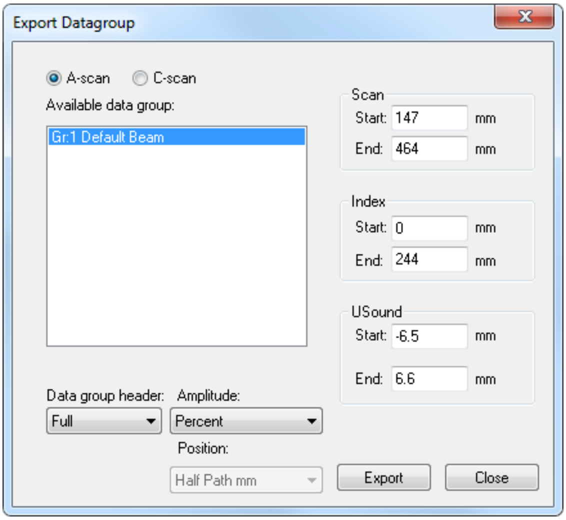


Figure 2-8 Export Datagroup dialog box

Save Screen as Bitmap

Opens a menu which allows you to take snapshots of the screen which then can be saved on the computer as *.bmp files. See section 10.6 on page 461 for more information.

Save Active View as Image

Takes a snapshot of the current view and saves it on the computer as either a *.bmp or *.tif file.

Preferences

Opens the **Preferences** dialog box, used to set various basic parameters related to general use of the software (see section 3.14 on page 210 for details).

Exit

Closes opened data files and quits TomoView.

2.4.2 Toolbars Menu

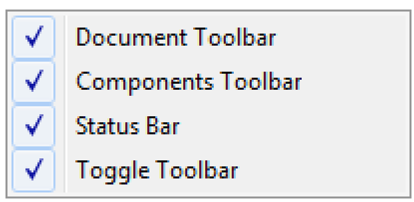


Figure 2-9 The Toolbars menu

The **Toolbars** menu (see Figure 2-9 on page 54) contains commands relating to the display of toolbars. The **View** menu commands are the following:

Document Toolbar

Alternately displays and hides the document toolbar.

Components Toolbar

Alternately displays and hides the **Component** toolbar.

Status Bar

Alternately displays and hides the status bar at the bottom of the views.

Toggle Toolbar

Alternately displays and hides the **View** toolbars.

2.4.3 Part & Overlay Menu

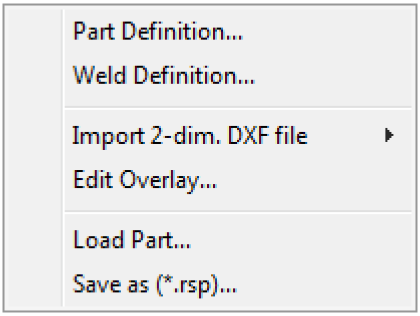


Figure 2-10 The Part & Overlay menu

The **Part & Overlay** menu (see Figure 2-10 on page 55) contains the commands used to manage and edit parts and overlays. The **Part & Overlay** menu commands are the following:

Part Definition

Opens the **Part Definition** dialog box, from which you define the geometry of flat or cylindrical parts.

Weld Definition

Opens the **Predefined Weld** dialog box, from which you select a weld type and define its geometry.

Import 2-Dim DXF file

Specimen submenu

Opens the **Select file name** dialog box, from which you browse to find and open a DXF file containing a drawing of the specimen.

Weld submenu

Opens the **Select file name** dialog box, from which you browse to find and open a DXF file containing the drawing of a weld.

Edit Overlay

Opens the **Edit Overlays** dialog box, from which you can rename, relocate, or delete a component.

Load (*.rsp)

Opens a standard **Open** dialog box, from which you can select and load an overlay defined in an .rsp file.

Save As (*.rsp)

Opens a standard **Save As** dialog box, from which you can browse through folders, type a new file name if necessary, and then save the current overlay to an .rsp file.

2.4.4 View Menu

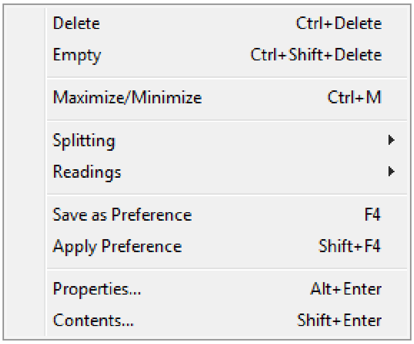


Figure 2-11 The View menu

The **View** menu (see Figure 2-11 on page 56) contains the following commands that manage pane displays, properties, and contents.

Delete

Removes the active pane from the current display.

Empty

Empties the active pane of its view contents.

Maximize/Minimize

Maximizes the size of the active pane by hiding the other panes. Choosing this command when the pane is already maximized restores the pane to its original size and displays the other panes.

Splitting

Opens a submenu that contains various split commands that can be used to create subdivisions in the active pane.

Readings

Opens a submenu that contains various commands that can be used to customize the readings in the active pane.

View Information

Opens a submenu (see Figure 2-11 on page 56) that gives access to various readings to customize options (see section 4.10 on page 276 for more information on how to use the readings).

Save as Preference

Saves the current active pane settings as the default property settings (preferences).

Apply Preference

Applies the default settings (preferences) to the active pane.

Properties

Opens the **View Properties** dialog box, used to consult and modify the various properties of the active pane (see section 3.12 on page 162 for more details).

Contents

Opens the **Contents** dialog box, used to select the data view types to be displayed in the active pane (see section 4.8.2 on page 263 for more details).

2.4.5 Layout Menu

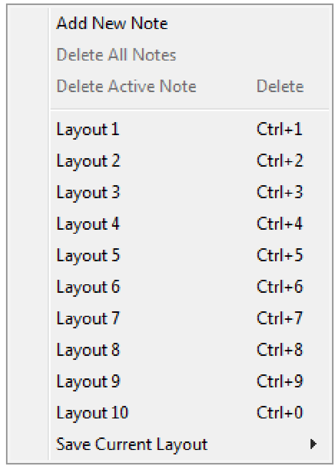


Figure 2-12 The Layout menu

The **Layout** menu (see Figure 2-12 on page 58) contains commands that control toolbar and note display, and manage the layouts. The **Layout** menu commands are the following:

Add New Note

Adds a balloon-shaped pop-up window that is displayed on top of the pane contents to point out and comment on a given element. Note that frames can be freely moved and resized according to your needs. Notes will be added to the printout but cannot be saved.

Delete All Notes

Removes all notes at once from the pane contents.

Delete Active Note

This command (or its shortcut, the DELETE key) removes the active note from the pane contents.

The other menu items are default factory layouts that you can replace with your own.

Layout 1, 2, 3,…, 8, 9, 10

These commands represent the available layouts. Choosing a command from this list applies the corresponding layout to the active window.

Save Current Layout

Opens a submenu (see Figure 2-13 on page 59) listing the available commands used to save layouts. Choosing a command from this list saves the layout of the active window under the corresponding layout number. A dialog box appears, which you can use to save the layout under the name of your choice.

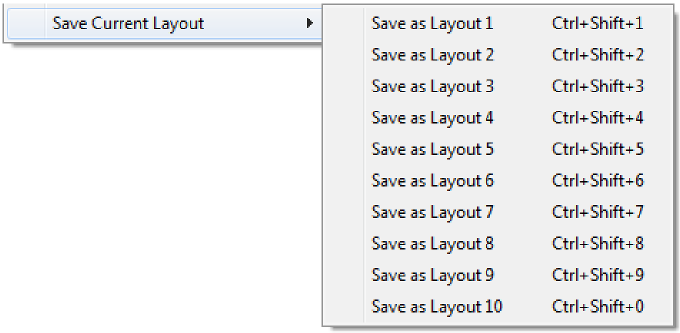


Figure 2-13 The Save Current Layout submenu

2.4.6 Processing Menu

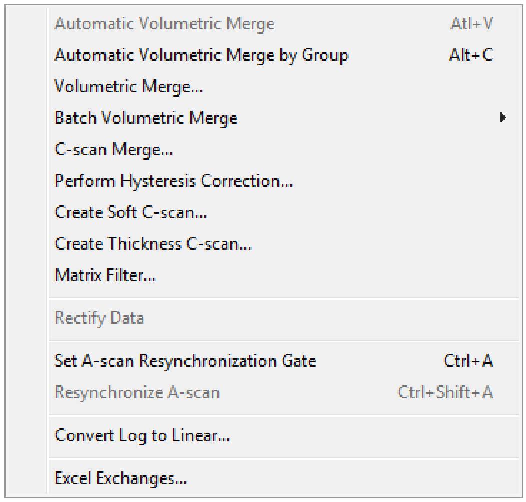


Figure 2-14 The Processing menu

The **Processing** menu (see Figure 2-14 on page 60) contains commands that apply various processing options to the data displayed in the active view. The **Processing** menu commands are the following:

Automatic Volumetric Merge

Performs a volumetric merge of all focal laws for all groups using the default parameters. See section 7.2 on page 335 for more information.

Automatic Volumetric Merge by Group

Performs a volumetric merge of all focal laws for each corresponding group using the default parameters. See section 7.2 on page 335 for more information.

Volumetric Merge

Opens the **Volumetric Merge** dialog box, used to perform a volumetric merge of recorded data in analysis mode. See section 7.2 on page 335 for more information.

Batch Volumetric Merge

Opens a submenu giving access to two commands:

- Create: Opens an **Open** dialog box, used to select and open a batch volumetric merge file.

- Execute: Opens an **Open** dialog box, used you to select and execute a batch volumetric merge file.

C-Scan Merge

Opens the **C-Scan Merge** dialog box, used to perform a C-scan merge of recorded data in analysis mode. See section 7.7 on page 363 for more information.

Perform Hysteresis Correction

Opens the **Hysteresis Correction** dialog box, which can be used to improve the visualization of previously recorded ultrasonic data from bidirectional scanning sequences by drastically reducing the effect of the mechanical backlash of the manipulator on the scan axis. See section 7.17 on page 424 for more information.

Create Soft C-Scan

Opens the **Create Soft C-Scan** dialog box, which can be used to create position and amplitude C-Scan data from previously recorded A-scan data. See section 7.9 on page 375 for more information.

Create Thickness C-Scan

Opens the **Create Thickness C-Scan** dialog box, which can be used to create thickness C-Scan data by performing subtractions on data originating from two gates. See section 7.1 on page 333 for more information.

Matrix Filters

Opens the **Matrix Filters** dialog box, which can be used to reduce the noise observed on the C-Scan data in order to provide a much clearer representation of the data for analysis in noisy environments. See section 7.14 on page 400 for more information.

Rectify Data

Applies a software rectification to the signal data in the active view.

Set A-Scan Resynchronization Gate

Defines the resynchronization gate (purple) in the A-scan display using the reference (red) and the measurement (blue) cursors. See section 7.13 on page 398 for more information.

Resynchronize A-Scans

Uses the resynchronization gate parameter to align A-scan peaks to zero depth. See section 7.13 on page 398 for more information.

Convert Log to Linear

Converts A-scan logarithmic data into linear data. See section 7.18 on page 427 for more information.

Excel Exchanges

Opens the **Excel Exchanges** dialog box, used to dynamically send the parameters available in the **Readings** to a Microsoft Excel spreadsheet. The results of the computation in Excel can then be dynamically sent back to TomoView as **Custom info fields** of the **View Information.** See section 10.4 on page 452 for more information.

2.4.7 Tools Menu

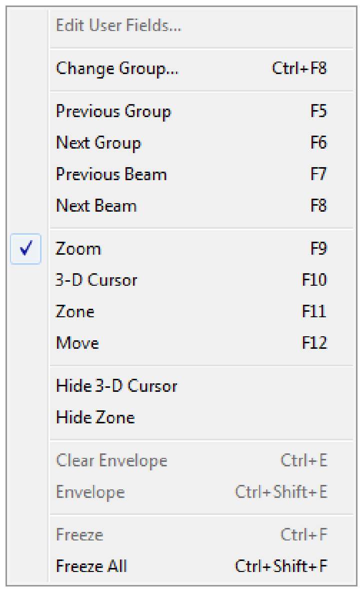


Figure 2-15 The Tools menu

The **Tools** menu (see Figure 2-15 on page 62) contains commands that set display parameters for the active view. The **Tools** menu commands are the following:

Edit User Fields

Opens the **Edit User Fields** dialog box, used to define and edit the user fields (see section 4.8 on page 261 for more details).

Change Group

Opens the **Change Group** dialog box, used to select a channel from a list.

Previous Group

Decreases the Current Group value by one unit in the order followed for the Group Creation.

Next Group

Increases the Current Group value by one unit in the order followed for the Group Creation.

Previous Beam

Decreases the Current Beam value by one unit in the regular beam numbering order.

Next Beam

Increases the Current Beam value by one unit in the regular beam numbering order.

Zoom

Enables the zoom tool, used to select and zoom in a specific region on a view.

3-D Cursor

Enables the 3-D Cursor tool (also called *segment*), used to make 3-D measurements. See section 7.12.3 on page 397 for more information.

Zone

Enables the zone tool, used to select a region on a view. See section 7.12.2 on page 394 for more information.

Hide 3-D Cursor

Hides the 3-D Cursor tool in all views.

Hide Zone

Hides the zone tool in all views.

2.4.8 Components

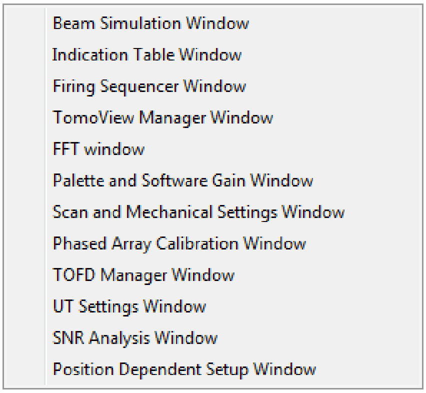


Figure 2-16 The Components dialog box

The **Components** menu (see Figure 2-16 on page 64) contains commands that set display parameters for the active view. The **Components** menu commands are the following:

Beam Simulation Window

Opens the Beam Simulation Window, used to simulate the trajectories and dimensions of ultrasonic beams generated by conventional and phased array probes in a user-defined geometry. See section 7.7 on page 363 for more information.

Indication Table Window

Opens the Indication Table Window, used to manage indication information and to create reports. See section 6.2 on page 324 for more information.

Firing Sequencer Window

Opens the Firing Sequencer Window, used to modify the order in which the ultrasonic beams are fired. See section 4.14 on page 289 for more information.

TomoView Manager Window

Opens the TomoView Manager Window, used to select the mode and access the group, scan, and layout parameters. See section 2.6 on page 73 for more information.

FFT Window

Opens the FFT Window, used to calculate the Fast Fourier transformation. See section 10.9 on page 473 for more information.

Palette and Software Gain Window

Opens the Palette and Software Gain Window, used to set the software gain and to dynamically change the color palette's maximum and minimum values. See section 7.5 on page 349 for more information.

Scan and Mechanical Settings Window

Opens the Scan and Mechanical Settings Window, used to define the scan and perform the settings for the mechanical scanning system. See section 3.11 on page 148 for more information.

Phased Array Calibration Window

Opens the Phased Array Calibration Window, used to calibrate the phased array beams. See section 4.3 on page 245 for more information.

TOFD Manager Window

Opens the TOFD Manager Window, used to analyze time-of-flight diffraction inspection data. See section 7.6 on page 350 for more information.

UT Settings Window

Opens the UT Settings Window, used to configure the ultrasonic setup parameters. See section 3.10 on page 117 for more information.

SNR Analysis Window

Opens the SNR Analysis Window, used to compute the signal-to-noise ration indicators. See section 7.8 on page 368 for more information.

Position Dependent Setup

Opens the Position Dependent Setup Window, used to synchronize the different setup and data files on predefined encoder positions. See section 5.4 on page 315 for more information.

2.4.9 Window Menu

Figure 2-17 The Window menu

The **Window** menu (see Figure 2-17 on page 66) contains commands that manage windows. The **Window** menu commands are the following:

Cascade

Arranges the windows so that they overlap. The title bar of each window remains visible, making it easy to select any window.

Tile

Arranges the windows side by side, so that each window is completely visible and none overlap.

Close All Files (.rdt)

Closes all currently opened data files.

Opened Window

Choosing a name from this list makes an opened document window active. All document windows currently opened are listed, including those that have been reduced to icons.

2.4.10 Help Menu

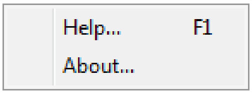


Figure 2-18 The Help menu

The **Help** menu (see Figure 2-18 on page 66) contains commands that provide access to information about TomoView.

Help

Opens a window providing access to complete TomoView documentation.

About

Opens a dialog box indicating which version and edition of TomoView you are running. See Figure 2-19 on page 67.

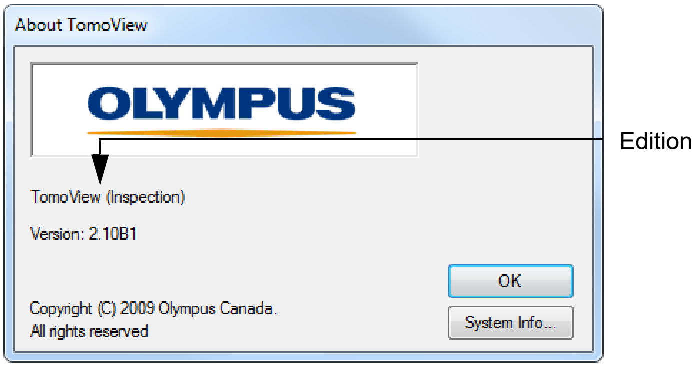


Figure 2-19 The About TomoView dialog box

2.5 Toolbars

The TomoView toolbars contain buttons that are used as shortcuts for frequently accessed dialog boxes or commands.

The toolbars are located by default at the top of the window but can be dragged anywhere as a floating toolbar on the screen. Inversely, you can drag and dock a floating toolbar to a window edge.

To hide or show a toolbar

◆ On the **Toolbars** menu, select or unselect the desired toolbar.

2.5.1 Component Toolbar

The Component toolbar, shown in Figure 2-20 on page 68 contains buttons to access basic TomoView components (see Table 5 on page 68 for details).

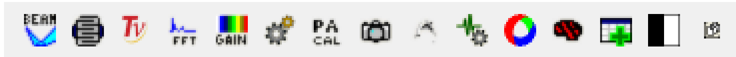


Figure 2-20 The Component toolbar

Table 5 The Component toolbar buttons

Icon	Component	Description
	Beam Simulation	To simulate the trajectories and dimensions of ultrasonic beams generated by conventional and phased array probes in a user-defined geometry.
	Firing Sequencer	To modify the order in which the ultrasonic beams are fired.
	TomoView Manager	To select the mode and access the group, scan, and layout parameters.
	Fast Fourier Transform	To calculate the Fast Fourier transform FFT.
	Gain Information	To set the software gain and dynamically change color palette maximum and minimum values.
	Scan and Mechanical Settings	To define the scan and to perform the settings for the mechanical scanning system
	Phased Array Calibration	To calibrate all phased array beams.
	TOFD Manager	To analyze time-of-flight diffraction inspection data.
	Screen Capture	To take a capture of the current display, which then can be saved as a **.bmp** file.
	UT Settings	To configure the ultrasonic setup parameters.
	OmniScan File Converter	To create a TomoView data file from an OmniScan data file (.opd, .oud) and to open the TomoView file (.rdt).

Table 5 The Component toolbar buttons *(continued)*

Icon	Component	Description
	SNR Analysis Utility	To compute the signal-to-noise ratio indicators.
	Indication Table	To manage indication information and create reports.
	Binarizer	To convert a view into a two-color threshold-based representation.
	Position Dependent Setup	To synchronize the different setup and data files on predefined encoder positions.

2.5.2 Document Toolbar

The Document toolbar (see Figure 2-21 on page 69) contains standard document management functions to open and save files, and to manage arrangements of multiple document windows (see Table 6 on page 69).

Figure 2-21 The Document toolbar

NOTE

The **Document** toolbar is not available in the **Lite** edition.

Table 6 The Document toolbar buttons

Icon	Function	Description
	Open	To open a standard **Open** dialog box, used to select and load a data file or a predefined setup.

Table 6 The Document toolbar buttons *(continued)*

Icon	Function	Description
	Save As	To open a standard **Save As** dialog box, used to browse through folders, type a new file name, if necessary, and then save the active setup.
	Cascade	To arrange the document windows in a cascade pattern.
	Tile Horizontally	To arrange the document windows in horizontal tiles.
	Tile Vertically	To arrange the document windows in vertical tiles.
	Close All Data Files	To close all opened data files.

2.5.3 View Toolbars

The **View** toolbars (see Figure 2-22 on page 70) contain various commands related to the management and properties of the views (see Table 7 on page 71 for details).

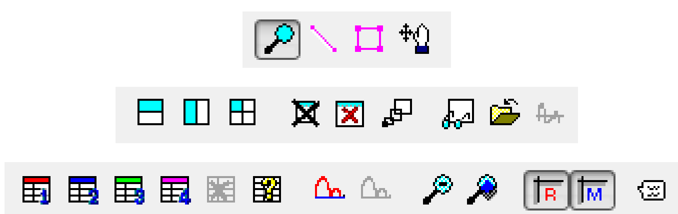


Figure 2-22 The View toolbars

Table 7 The View toolbar buttons

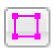
Icon	Name	Function
	Zoom tool	To select and zoom in a specific region on a view.
	3-D Cursor tool	To make 3-D measurements on a C-scan view by clicking and dragging. Also works on Top/Side/End views.
	Zone tool	To select a region on a top, side, end, or C-scan view by clicking and dragging. **Tip**: When the Zone tool is not selected, press and hold the CTRL key, and then click and drag on a view to perform the same task.
	Move tool	To move the graphical data relative to the view frame.
	Split view horizontally in two	To divide the active view into two views with the same horizontal dimension.
	Split view vertically in two	To divide the active view into two views with the same vertical dimension.
	Split view in four	To divide the active view into four views with the same horizontal and vertical dimensions.
	Delete view	To delete the active view.
	Empty view	To remove the contents of the active view.
	Maximize view	To maximize the size of the active view by hiding the other views. Clicking this button when the view is already maximized restores the view to its original size and displays the other views.
	Edit view properties	To open the **View Properties** dialog box, used to configure the parameters of the active view.
	Edit view contents	To open the **Contents** dialog box, used to select the data type and the view type to be displayed in the active view.

Table 7 The View toolbar buttons *(continued)*

Icon	Name	Function
	Toggle echo dynamics curves	To toggle the display of the echo dynamics curve on the active view. This button is available only for top, side, end, and C-scan views.
	Toggle view information group 1	To toggle the display of information group 1 in the active view (reference cursor parameters by default).
	Toggle view information group 2	To toggle the display of information group 2 in the active view (measurement cursor parameters by default).
	Toggle view information group 3	To toggle the display of information group 3 in the active view (by default, parameters related to the combination of the reference and measurement cursors).
	Toggle view information group 4	To toggle the display of information group 4 in the active view (3-D cursor, Zone tool, and acquisition parameters by default).
	Hide view information groups	To hide all information groups displayed in the active view.
	Edit view information groups contents	To open the **Information Groups** dialog box, used to edit information group contents, that is, the parameter sets displayed in the information groups. You can also double-click a reading to open this dialog box.
	Envelope	Toggles the activation of the envelope mode for the online and off-line A-scan view.
	Rectify data	To apply a software rectification to unrectified signal data.
	Fit image to view	To display the complete data contents of the active view.
	Fit inspection domain to view	To display the complete inspection domain in the active view.
	Toggle reference cursor	To toggle the display of the active view reference cursors.

Table 7 The View toolbar buttons *(continued)*

Icon	Name	Function
	Toggle measurement cursor	To toggle the display of the active view measurement cursors.
	Add a note	To add a callout on top of a view to point out and comment on a given element. You can freely move and resize the callout. The callout notes are included in screen captures but cannot be saved.
	US Customary units	In TomoView Lite and TomoViewer editions only, changes length measurement units to US Customary when pressed.

2.6 The TomoView Manager Dialog Box

The **TomoView Manager** dialog box is where you pilot the operations in TomoView. The **TomoView Manager** contains the mode, group, scan, and layout parameters that you need to define before performing an inspection (see Figure 2-23 on page 74).

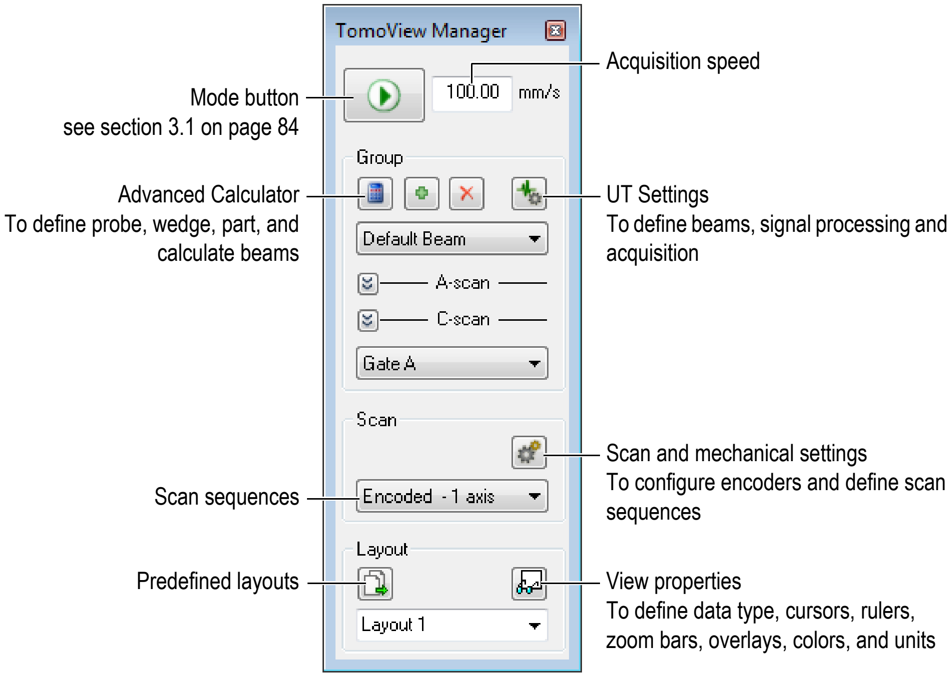


Figure 2-23 Components of the TomoView Manager dialog box

2.7 Document Windows

TomoView is a multi document software. This means that you can open one or more documents at a time. However, only one setup can be open at a time. The data for an opened document appears in a document window (see Figure 2-24 on page 75).

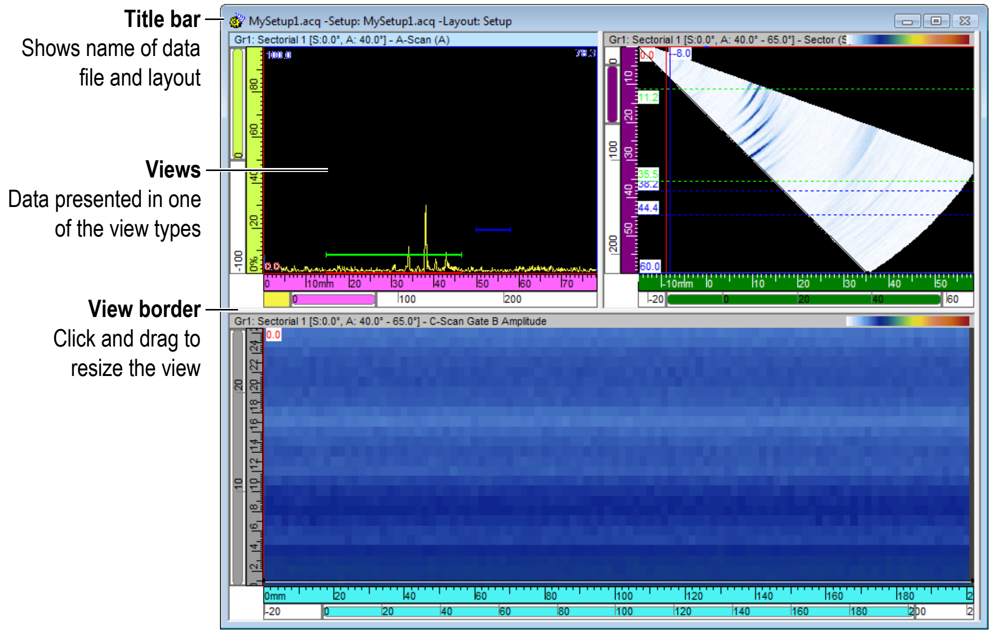


Figure 2-24 Example of a document window containing three views

You can cascade or tile multiple document windows using the Document toolbar (see section 2.5.2 on page 69), or using the **Windows** menu.

You can add, remove, or empty views in a document window using the View toolbars (see section 2.5.3 on page 70), or using the **View** menu. Two or more views appear side by side, never overlapping. You activate any view by clicking on it.

2.8 Layouts

In TomoView, a layout is an organization of two or more views appearing in the document window. Layouts offer extensive flexibility in how you present your inspection data.

TomoView offers a set of ten layouts that are available from the **TomoView Manager** dialog box for quick selection (see Figure 2-25 on page 76). You can also select one of the layouts from the **Layout** menu. A set of ten layouts is saved in an .rst file.

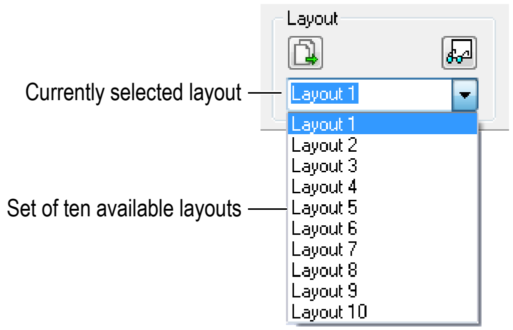


Figure 2-25 The set of ten layouts

2.9 The Status Bar

The status bar appears at the bottom of the document windows (see Figure 2-26 on page 76) when you select **Toolbars > Status Bar** on the menu.

Figure 2-26 The Status Bar

The status bar displays the following information relative to the state of the system:

Scan

Position on the scan-axis from the encoder or from the 1-axis internal clock.

Index

Position on the index-axis from the encoder or from the 2-axis internal clock.

Set Encoders

Sets the encoder position of the **Scan** and **Index** axes to the **Preset** values defined in the **Scan and Mechanical settings** dialog box.

Acq. speed

Maximum acquisition speed with the current settings, directly related to current PRF value as specified by the following equation:

$$\text{Acq. speed} = \text{PRF} \cdot \text{Scan resolution}$$

where:

PRF:

The pulse repetition frequency is set in the **UT Settings** dialog box > **Digitizer** tab > **PRF** group > **Current** parameter.

Scan Resolution:

The scan resolution is set in the **UT Settings** dialog box > **Digitizer** tab > **Data** group > **Resolution** parameter.

TIP

You can adjust the **Acq. speed** by changing the **PRF** value (in the **Digitizer** tab of the **UT Settings** dialog box) and the **Resolution** value (in the **Scan** tab of the **Scan and Mechanical Settings** dialog box). Note however that a too high **Acq. speed** value can generate ghost echoes. For example, with a scanner that can move a probe up to 100 mm/s and a scan resolution of 2 mm, in the **TomoView Manager**, set the **Acq. speed** to 100 mm/s. TomoView automatically sets the **PRF** value to 50 Hz. For manual encoder inspections, set the **Acq. speed** value between 50 mm/s and 100 mm/s.

Mode

Current mode (from Setup, Inspection, or Analysis).

Al.

Alarm indicator state for the Alarm 1, Alarm 2, and Alarm 3 (from left to right) described in Table 8 on page 78. The alarm indicators appear only when you define at least one alarm in the **Alarms** tab of the **UT Settings** dialog box.

Table 8 Alarm (Al.) indicator states

Indicator color	Alarm state
	Active
	Not active
	Not defined

Link

Indicator color reporting the state of the communication with the acquisition unit as given in Table 9 on page 78.

Table 9 Link indicator states

Indicator color	Communication with acquisition unit
	Correctly established
	Not established
	No attempt

Acq. Unit

Internal temperature of the acquisition unit (for OmniScan and FOCUS LT only). The background color of the value changes when it is out of the normal value range as shown in Table 10 on page 78.

Table 10 Interpretation of OmniScan MX and FOCUS LT internal temperature

Status	Internal temperature range [°C]	Indicator example
Normal operation	0 to 59.9	47.7°C
High temperature	60 to 61.9	60.4°C
Overheating warning	62 to 64.9	63.1°C

Table 10 Interpretation of OmniScan MX and FOCUS LT internal temperature

Status	Internal temperature range [°C]	Indicator example
Critical overheating	65 to 67.9	
Automatic shutdown	≥ 68	N/A

2.10 UT Settings Dialog Bar

Clicking the UT Settings button () on the **Main** and **TomoView Manager** toolbars toggles the visibility of the **UT Settings** dialog box. The **UT Settings** dialog box contains eight basic tabs: **General**, **Gate**, **TCG**, **Digitizer**, **Pulser/Receiver**, **Probe**, **Alarms**, and **I/O** (see Figure 2-22 on page 70). When you use a phased array unit such as the Focus LT, this dialog box also includes two additional tabs: **Transmitter** and **Receiver**. For more details on the different tabs of the UT Settings dialog box, see section 3.10 on page 117.

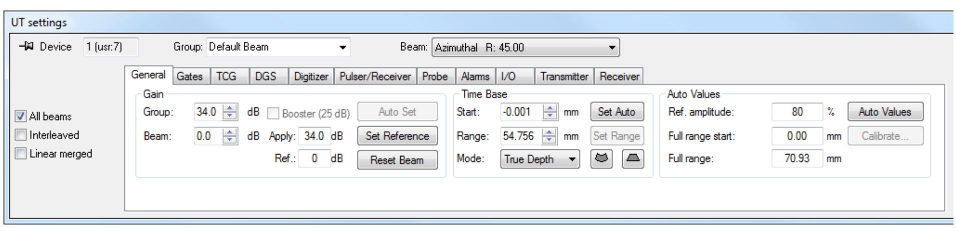


Figure 2-27 The UT Settings dialog box

2.11 Scan and Mechanical Settings Dialog Box

Clicking the Scan and Mechanical Settings button () on the **Main** and **TomoView Manager** toolbars toggles the visibility of the **Scan and Mechanical Settings** dialog box that contains four basic tabs: **Scan**, **Scan Controls**, **Encoders**, and **Options** (see Figure 2-28 on page 80). When working in Master mode, this dialog also includes two additional tabs: **MCDU Control** and **MCDU I/O**. For more details on the different tabs of the Scan and Mechanical Settings dialog box, see section 3.11 on page 148.

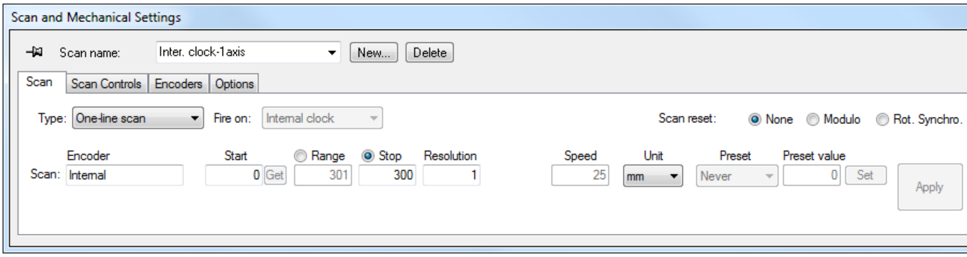


Figure 2-28 The Scan and Mechanical Settings dialog box

2.12 The View and Pane Properties

Clicking the View and Pane Properties button () on the **Main** and **TomoView Manager** toolbars toggles the visibility of the **View and Pane Properties** dialog bar. The **Info, Display, Echo Dynamics, Overlay, Palette, Data Source, Parameters, Units**, **View Linking**, **Rebounds**, and **Strip** tabs are available in the Pane Properties dialog box, depending on the data type contained in the active view. For more details on the different tabs of the **View and Pane Properties** dialog bar, see section 3.12 on page 162.

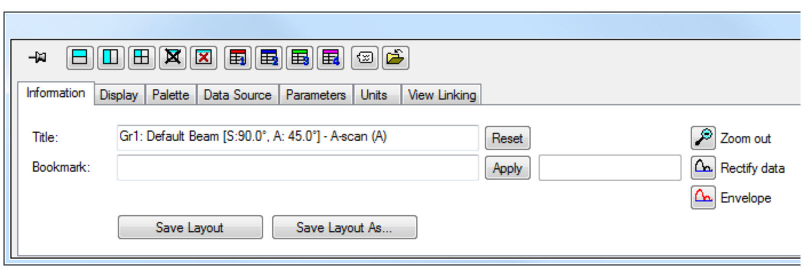


Figure 2-29 The View Properties dialog box

2.13 Docking Dialog Boxes

TomoView allows you to dock the main dialog boxes on a window border. A docking dialog box aligns itself with an edge of the window. By dragging its title bar, you can move the docking dialog box anywhere on the screen as a floating dialog box. Inversely, you can drag and dock the floating dialog box to one of the window edges. See Figure 2-30 on page 81.

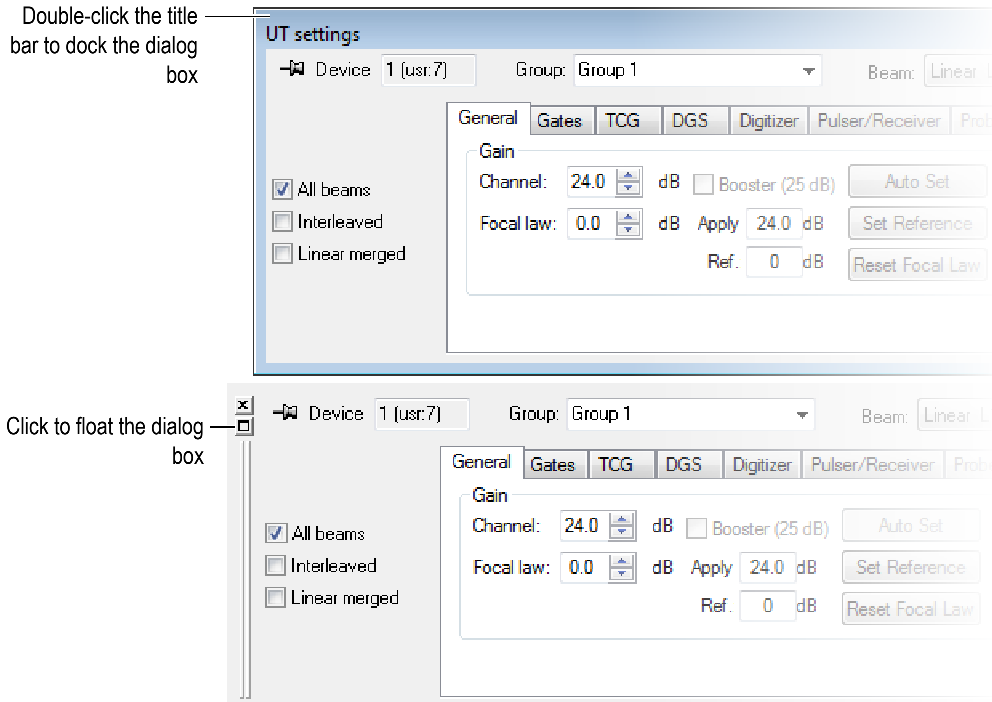


Figure 2-30 Floating (top) and docking (bottom) of a dialog box

Using the *thumbtack* function, you can specify if the dialog box remains displayed, or not when you open other docking dialog boxes. For this purpose, you can click the thumbtack icon, located in the upper-left corner of the dialog box, to toggle between the two following options:

Tacked option icon ()

The dialog box remains open when you open other docking dialog boxes.

Untacked option icon ()

The dialog box closes when you open other docking dialog boxes.

3. TomoView Concepts and Operational Mode

The TomoView user interface shown in Figure 3-1 on page 83 provides toolbars, dockable dialog boxes, a data display, and a status bar.

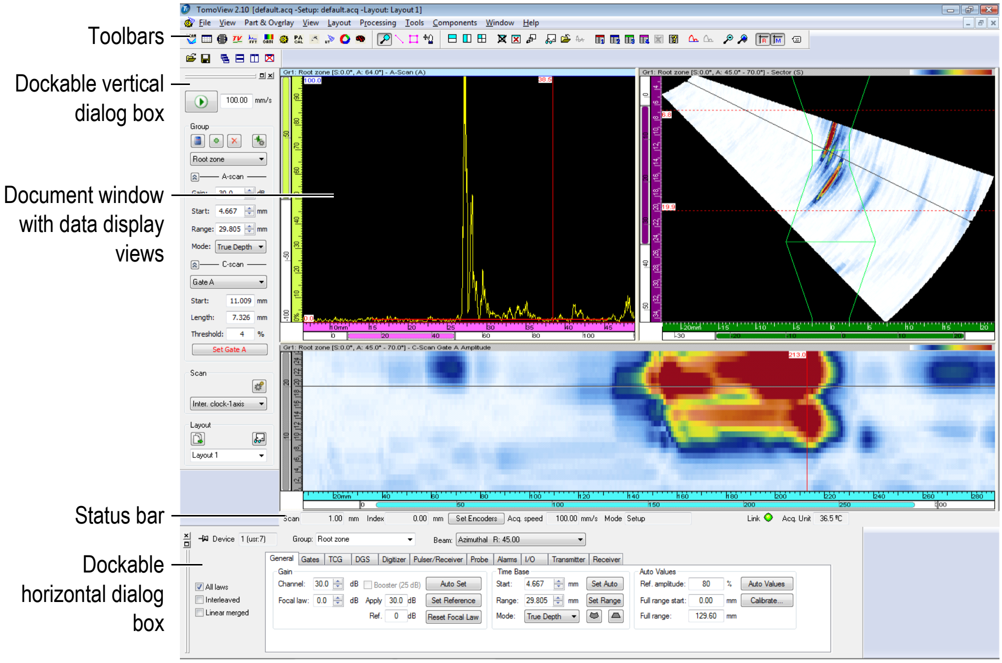


Figure 3-1 The main elements of the TomoView user interface

3.1 About the TomoView Modes

TomoView has three modes:

Setup

Mode in which you set up the various hardware and software parameters (ultrasonic, scan, and window layout settings). TomoView starts in the **Setup** mode when it is connected to an acquisition unit.

Inspection

Mode in which you perform data acquisition. The **Inspection** mode is available only when TomoView is connected to an acquisition unit.

Analysis

Mode in which you carry out analysis and produce reports for recorded data. TomoView starts in the **Analysis** mode when it is not connected to an acquisition unit.

To go from one mode to the another, on the **TomoView Manager**, click the mode button (see Figure 3-2 on page 85). The mode button changes depending on the current mode.

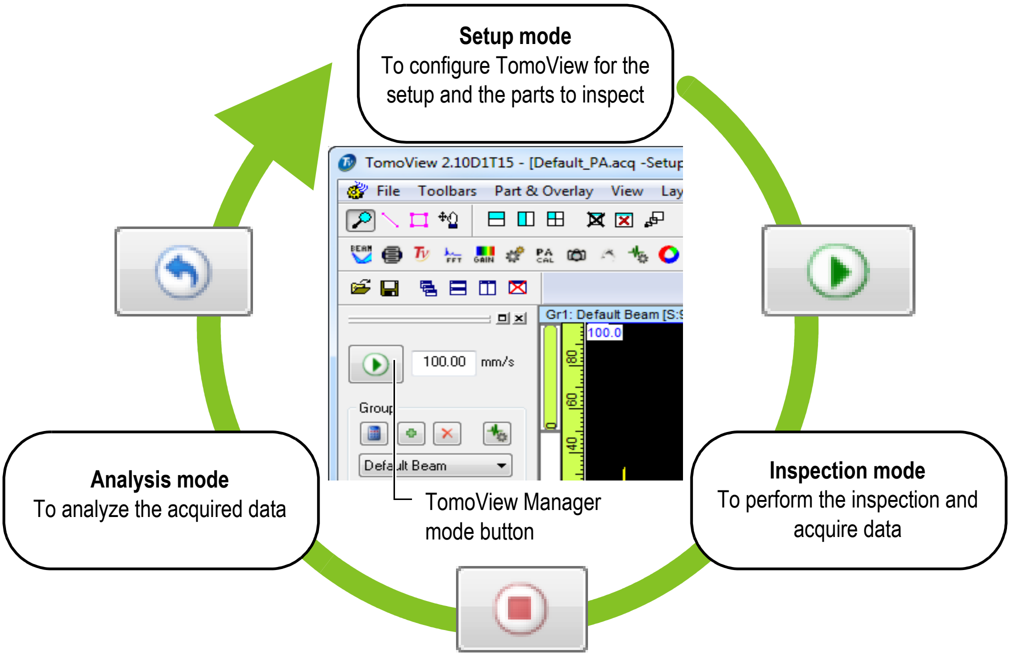


Figure 3-2 Switching between modes

The status bar, visible when selecting **Toolbars > Status Bar** on the menu, includes the **Mode** parameter that indicates the current mode (see Figure 3-3 on page 85).

Figure 3-3 The mode indicated in the status bar

3.2 About Groups

In TomoView, a *group* is a named configuration of parameters required to generate one or more ultrasonic beams. A group can use conventional or phased array probes. A group can pulse and receive on the same probe or use two different probes to pulse on one and receive on the other. A probe can be used by more than one group.

Assembling different beams in one group allows you to set the same parameters for all the beams at one time. This also allows you to display images built from all beams (ex: sectorial scan). Depending on the application, it might be helpful to use different settings for different beams (ex: different band-pass filters), justifying the creation of one group per beam.

You can create, delete, select, and configure groups from the **TomoView Manager** dialog box (see Figure 3-4 on page 86).

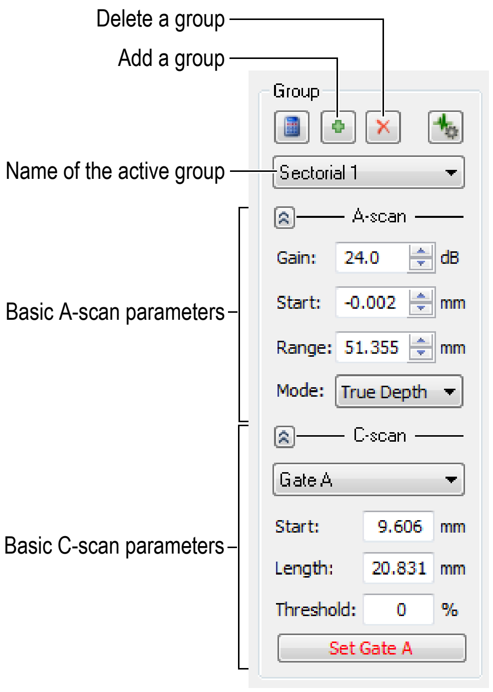


Figure 3-4 The Group area in the TomoView Manager dialog box

For example, you can create a first phased array group to generate a linear scan, a second group to generate a sectorial scan, and then display them concurrently in a layout (see Figure 3-5 on page 87).

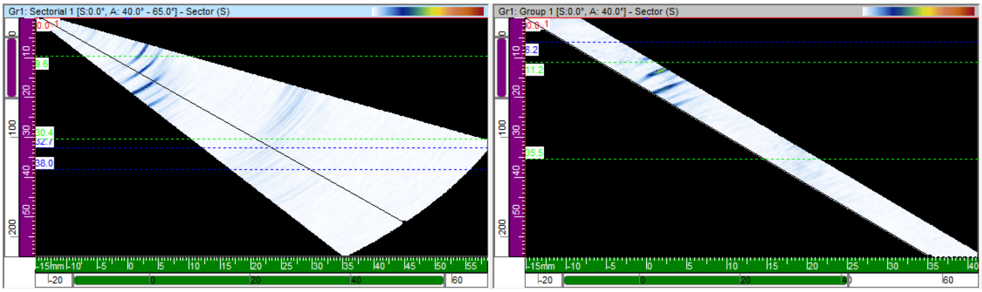


Figure 3-5 Example of two scans from two different groups

3.3 About the Advanced Calculator

The Advanced Calculator is an Evident software that comes with TomoView. You start the Advanced Calculator from the **TomoView Manager** dialog box, by clicking the Beam Calculator button ( ). From TomoView, use the Advanced Calculator to specify the probe and wedge used in the inspection, the geometry and material of the inspected part, and the beam configuration (see Figure 3-159 on page 219). The Advanced Calculator calculates the beams and returns the information to TomoView.

Refer to the *Advanced Calculator User's Manual* (p/n: DMTA080-01EN) for more details.

3.4 About Scan Sequences

In TomoView, you can configure scan sequence parameters and save them as a named scan. You can quickly select a scan sequence from the TomoView Manager. TomoView comes with useful predefined scan sequences (see Figure 3-6 on page 88).

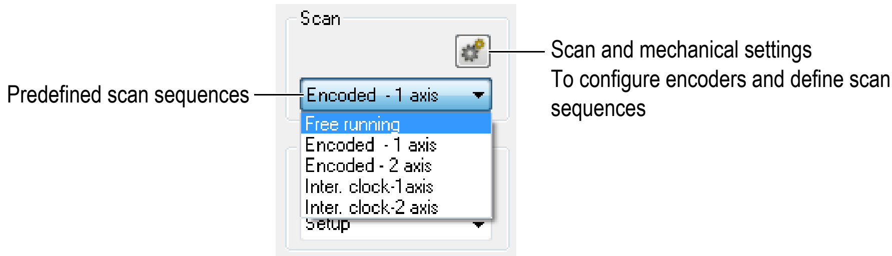


Figure 3-6 Predefined scan sequences available from the TomoView Manager

You can edit, delete, or create a scan sequence configuration (including predefined scan sequences) using the **Scan and Mechanical Settings** dialog box (see Figure 3-7 on page 88). The scan sequence data is saved in the hardware setup (.acq) file.

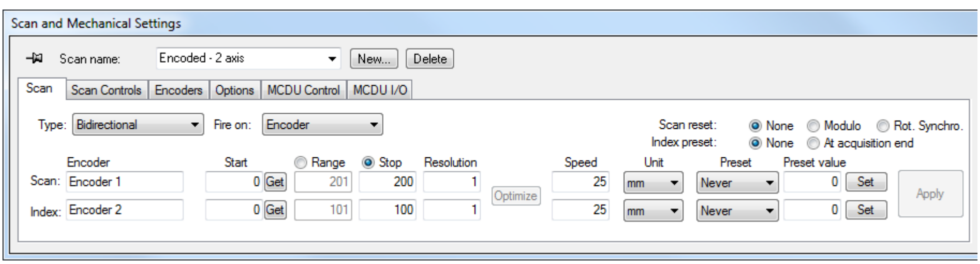


Figure 3-7 The Scan tab of the Scan and Mechanical Settings dialog box for the Encoded - 2 axis predefined sequence

The predefined scan sequences are:

Free running

Scan where data is acquired at the rate specified under **PRF** on the **Digitizer** tab of the **UT Settings** dialog box. The data is recorded at only one position, at the origin of the scan and index axes.

Encoded - 1 axis

Scan using one position encoder to determine the position during the acquisition along a linear path. The data is recorded at every interval (corresponding to the resolution setting) along the path from the start position to the end position of the scan axis.

Encoded - 2 axis

Scan using two position encoders to determine the position during the acquisition on a bidimensional surface-scan. The data is recorded at every interval (corresponding to the resolution setting) along the path from the start position to the end position of both the scan and index axes.

Inter. clock - 1 axis

Scan using the internal clock (PRF) to determine the position during the acquisition along a linear path. The data is recorded at every interval (corresponding to the resolution setting) along the path from the start position to the end position of the scan axis.

Inter. clock - 2 axis

Scan using the internal clock (PRF) to determine the position during the acquisition on a bidimensional surface-scan. The data is recorded at every interval (corresponding to the resolution setting) along the path from the start position to the end position of both the scan and index axes.

TIP

When you modify or delete predefined scan sequences, you can restore them by opening a default hardware setup (.acq) file.

3.5 About Probe Orientation Conventions

This section describes the conventions used in TomoView for the probe and the wedge orientations relative to the axes.

The probes and wedges are illustrated schematically as shown in Figure 3-8 on page 90. The probe element number of a phased array generally increases from the back to the front of the probe/wedge assembly.

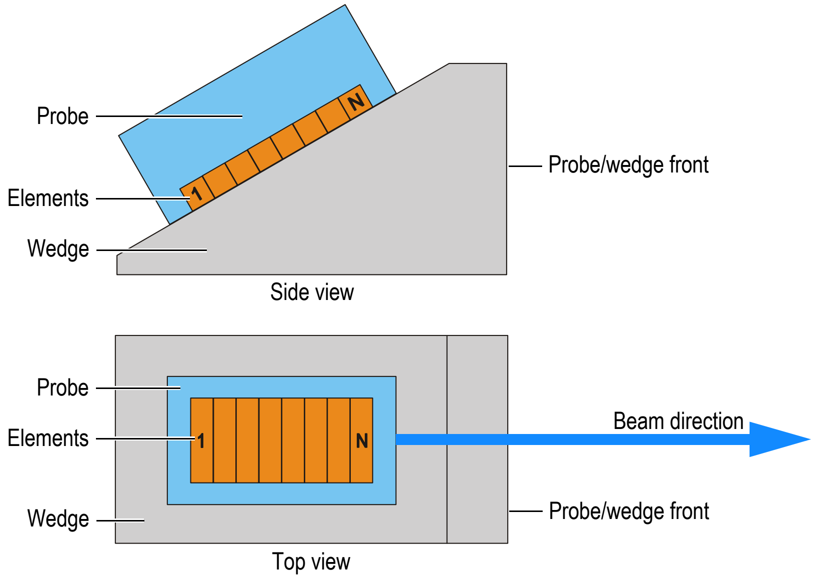


Figure 3-8 Example of a probe and a wedge illustration

NOTE

In rare applications where the probe connector or wire is physically interfering with other inspection setup components, you can mount the probe in a reverse position on the wedge. To notify TomoView about this, in the **Advanced Calculator**, select the **Reverse primary axis** check box under the **Probe** area. The check box is automatically selected when you select a reversed wedge model.

The probe skew is defined as the angle between the primary axis of the probe and the scan-axis. The skew has a value of 0° when the beam direction points parallel to the scan axis in the positive direction. The skew angle increases clockwise.

In the example shown in Figure 3-9 on page 91, the angle beam probe is moving on the inspected part along the scan axis following a raster scan pattern. The beam direction is parallel to the scan axis. Consequently, the probe skew is equal to 0°.

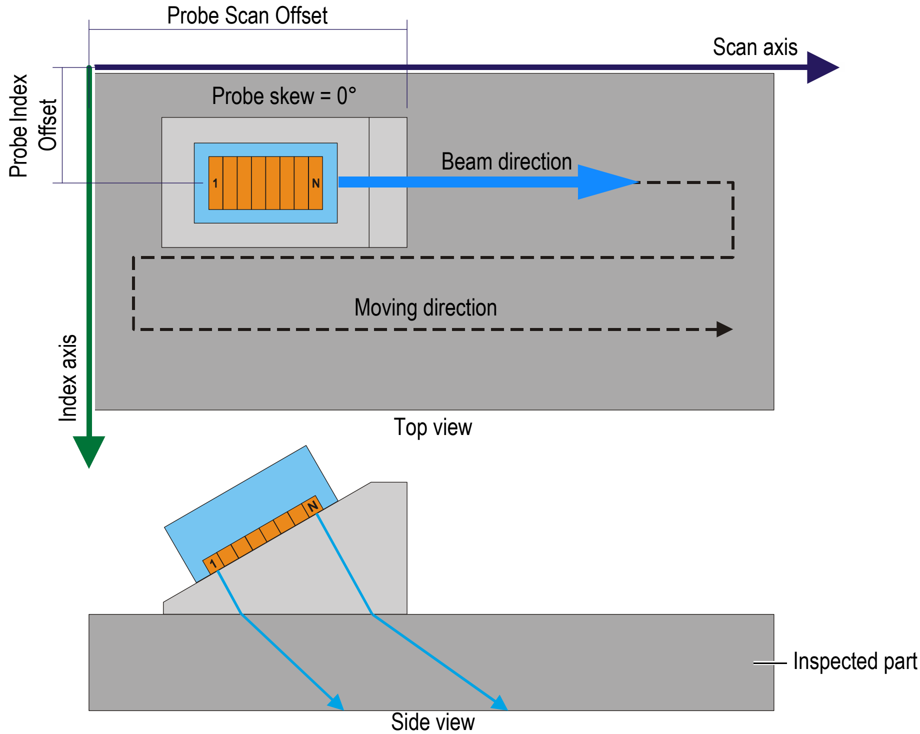


Figure 3-9 Raster scan plate inspection with 0° probe skew

In the example shown in Figure 3-10 on page 92, the probe is moving on the inspected part along the scan axis. The beam direction is along the ultrasound axis but the beam electronic scanning direction is parallel to the index axis. Consequently, the probe skew is equal to 90°.

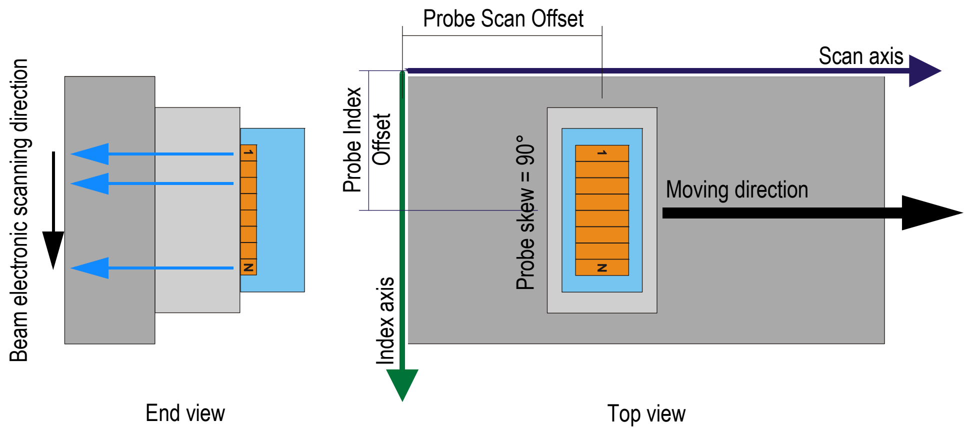


Figure 3-10 Plate inspection with 90° probe skew

In the example shown in Figure 3-11 on page 92, the angle beam probe is moving on the inspected part along the scan axis and the beam direction is parallel to the scan axis. Consequently, the probe skew is equal to 0°.

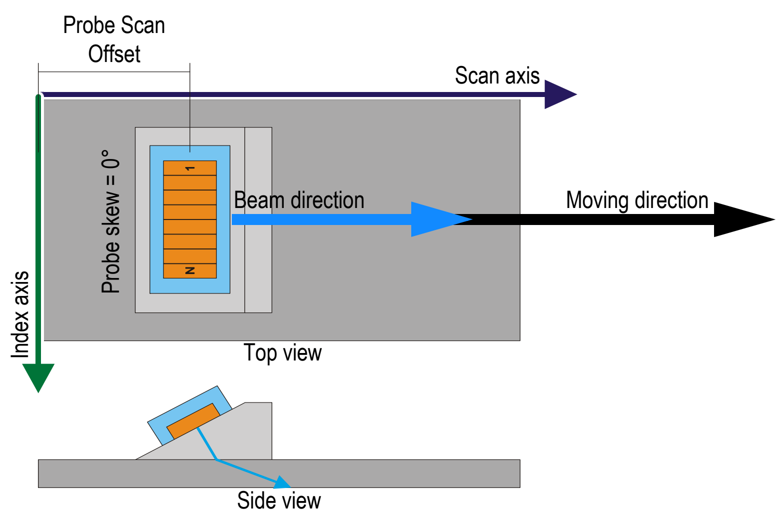


Figure 3-11 Rivet inspection with 0° probe skew

In the example shown in Figure 3-12 on page 93, a mechanical scanner is holding two angle beam PA probes and two angle beam UT probes. The four angle beam probes are moving on the inspected part along the scan axis and the along the weld. The direction of the beam is parallel to the index axis. Consequently, the probe skews are equal to 90° or to 270°.

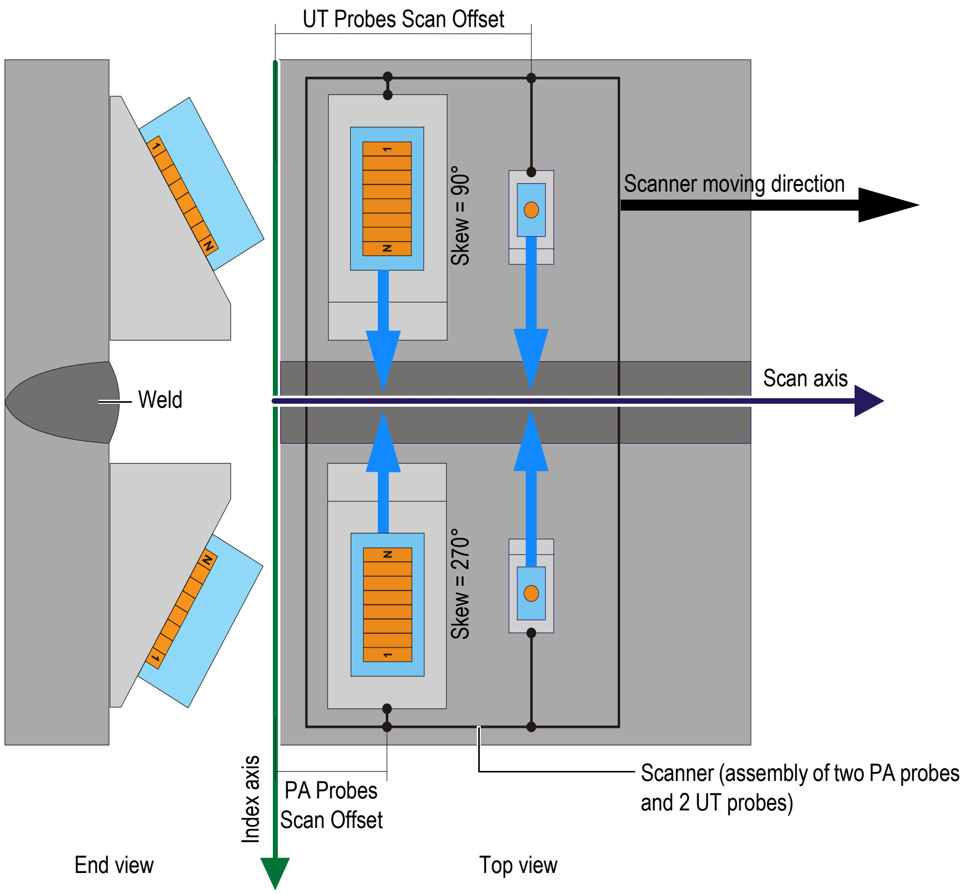


Figure 3-12 Weld inspection using a scanner with 90° and 270° probe skews

In the example shown in Figure 3-13 on page 94, the angle beam probes are moving on the inspected disc or wheel circumference along the scan axis. The beam direction is parallel to the scan axis. Consequently, the probe skews are equal to 0° or to 180°.

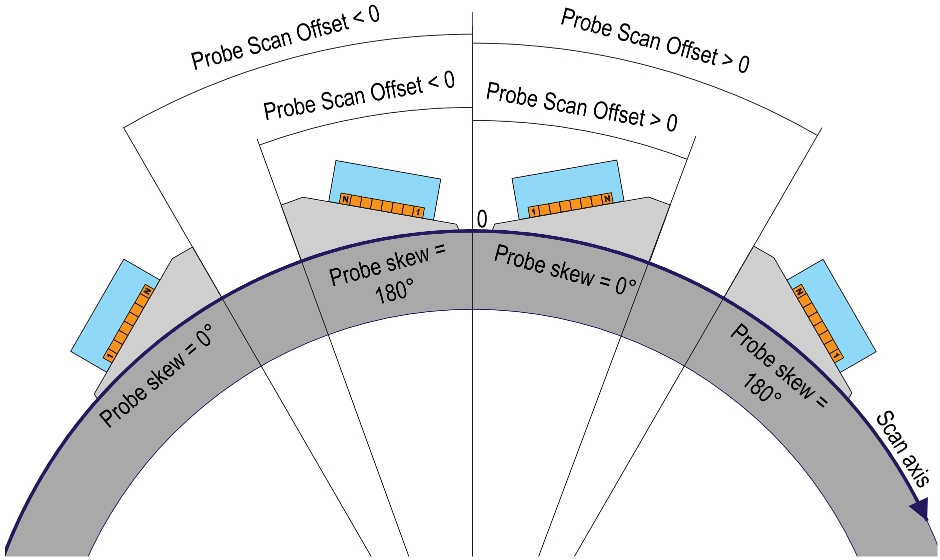


Figure 3-13 Disc or wheel inspection with 0° and 180° probe skews

In the example shown in Figure 3-14 on page 95, two facing angle beam PA probes are moving on the inspected tube along the scan axis along the weld. The scan axis, represented by the symbol ⊗ in Figure 3-14 on page 95, is oriented in the third dimension perpendicular to the plan of the figure and increases toward inside the page. The beam directions are parallel to the index axis. Consequently, the probe skews are equal to 90° and 270°.

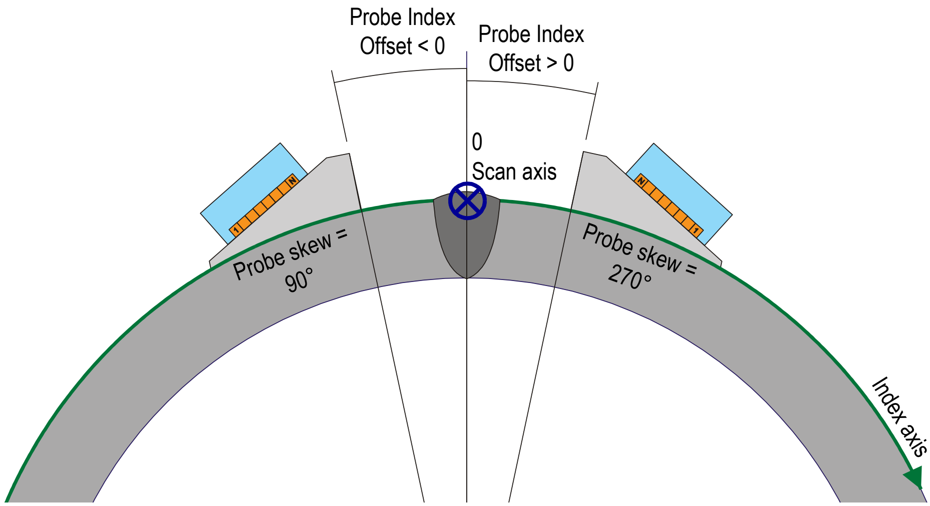


Figure 3-14 Tube weld inspection with 90° and 270° probe skews

3.6 About Layouts

In TomoView, a layout is an organization of two or more views appearing in the document window. Layouts offer extensive flexibility in how you present your inspection data.

TomoView offers a set of ten layouts that are available from the **TomoView Manager** dialog box for quick selection (see Figure 3-15 on page 96). You can also select one of the layouts from the **Layout** menu. A set of ten layouts is saved in an .rst file.

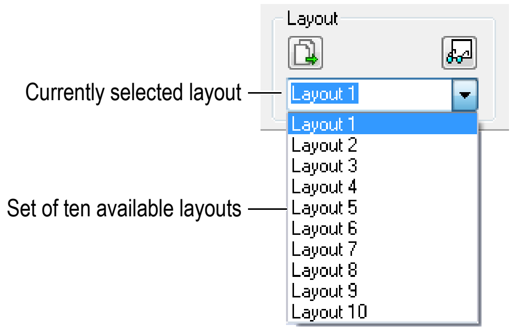


Figure 3-15 The set of ten layouts

The predefined layouts button of the **TomoView Manager** dialog box allows you to quickly load a set of predefined layouts adapted to an application (see Figure 3-16 on page 96).

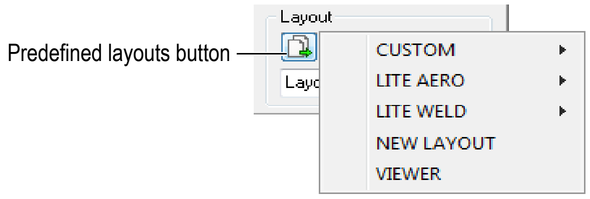


Figure 3-16 Selecting predefined layouts

3.7 About Views

A view displays the data for the current document in one of the data view types. Figure 3-17 on page 97 gives an example of a view showing an A-scan data view type.

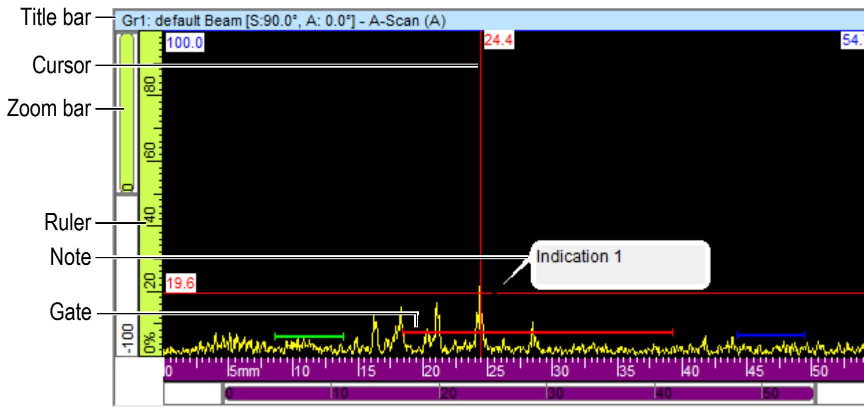


Figure 3-17 Example of an active A-scan view

A view contains the following elements:

Title bar

The title bar of the active view is highlighted with a light-blue background as shown in Figure 3-18 on page 97.

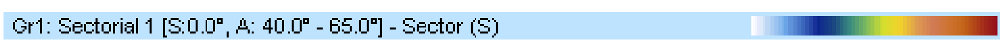


Figure 3-18 Example of the title bar of an active view

The title bar contains information describing the data in the view:

<Group Number><Group Name>[S: <Skew Angle>, A: <Beam Angles>]

where:

<Group Number>: Sequential number identifying the group (e.g.: Gr1).

<Group Name>: Name of the group (ex: Sectorial 1)

<Skew Angle>: Skew angle of the current group (e.g.: 90°) taking into account the skew angle of the probe and of the beam.

<Beam Angles>: Angle or range of beam angles (e.g.: 40° to 60°)

Rulers

The rulers are scales displayed at the left and at the bottom of a view. The color of the ruler identifies the axis. The units and number of precision digits can be adjusted using the **View and Pane Properties > Units** tab.

Zoom bar

Zoom bars appear at the left and bottom edges of a view. Each zoom bar contains a zoom box, which enables you to set the visible part of the view. The zoom box shows the relative position and the proportion of data currently visible in the data display area relative to the contents of the entire inspection domain. The color of the zoom bar identifies the axis on the different views. You can resize the zoom bar by dragging its ends to zoom in or out and scroll the zoom bar to see other parts of the data. You can use the mouse wheel with or without the Ctrl key to scroll the zoom bars.

Grid

The grid consist of thin horizontal and vertical lines that are displayed in the curve area to facilitate measurements and correspondence with the rulers. Depending on the distance between lines, the grid might be coarse, medium, or fine. The grid can be activated and customized using the **View and Pane Propertie**s **> Display** tab.

Cursors

The cursors are thin horizontal and vertical lines that are used to measure the data displayed in views and to identify a region in the view. A label indicates the exact measure of each cursor. Two cursor types are available for a view: reference and measurement cursors.
You can quickly display the reference cursor by double-clicking in a view with the left mouse button. Similarly, double-click in a view with the right mouse button to display the measurement cursor.

Notes

A note is a callout text box note that points to an element of the view (see Figure 3-17 on page 97 for an example). The Note icon button () can be used to add a new note.

3.7.1 Data View Types

The data view types are graphical representations of the ultrasonic data. There are three types of ultrasonic views:

1. Basic views:
   - A-scan
   - S-scan
2. Volumetric views:
   - Side (B)
   - Top (C)
   - End (D)
   - Polar
3. Scrolling views:
   - Scrolling B-scan
   - Scrolling strip chart (position)
   - Scrolling strip chart (amplitude)

You can select the data view type for a view by selecting the view, and then selecting **View > Contents** on the menu or pressing Shift + Enter to open the **Contents** window (see example in Figure 3-19 on page 99). In the **Contents** dialog box, the available data view types depend on various parameters including the acquisition type (conventional  or phased array ) and the mode (Setup, Inspection, or Analysis).

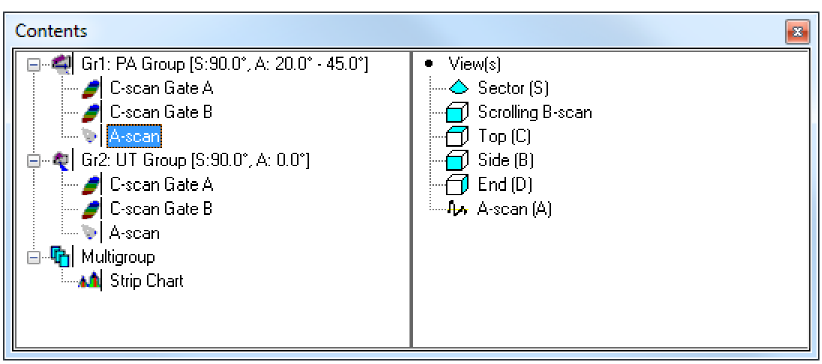
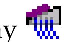


Figure 3-19 Example of data view types available for phased array data in the Contents window

TIP

A view appears empty when its content (conventional  or phased array ) is not available in the current data file.

3.7.1.1 Basic Views

The following is a description of each type of view:

A-scan View

The A-scan view is the basis for all other views. It is a representation (view) of the received ultrasonic pulse amplitude versus time of flight (ultrasonic path), or a waveform. The **Online A-Scan** view (see example in Figure 3-20 on page 100) is a real-time display of the received ultrasonic pulse amplitude versus the time of flight. A peak in the signal is associated with the echo of a defect or a discontinuity in the specimen. Peaks at the start and the end of the ultrasonic axis are generally associated with the echo of the specimen's entry surface and the back-wall.

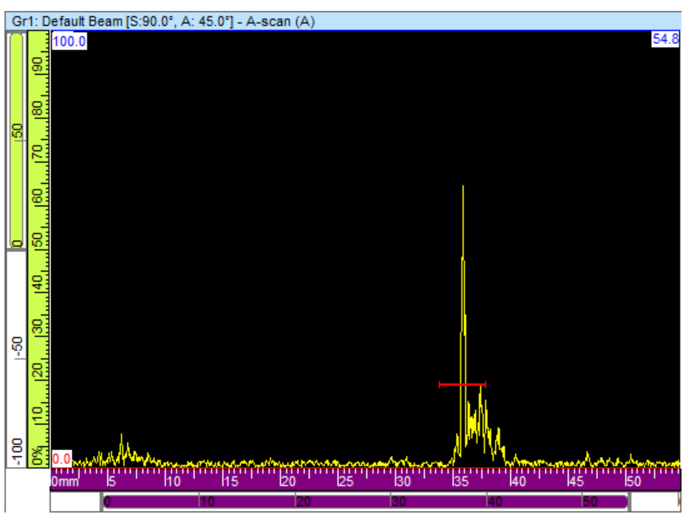


Figure 3-20 Example of an Online A-Scan view

Sectorial Views

NOTE

Sectorial views are only available for phased array channels.

Sectorial views are representations of the juxtaposition of the A-scans associated with each focal law of a phased array scan. They present a 2-D view of the sector covered by the scan. Figure 3-21 on page 101 shows the three types of sectorial representation for a given data set.

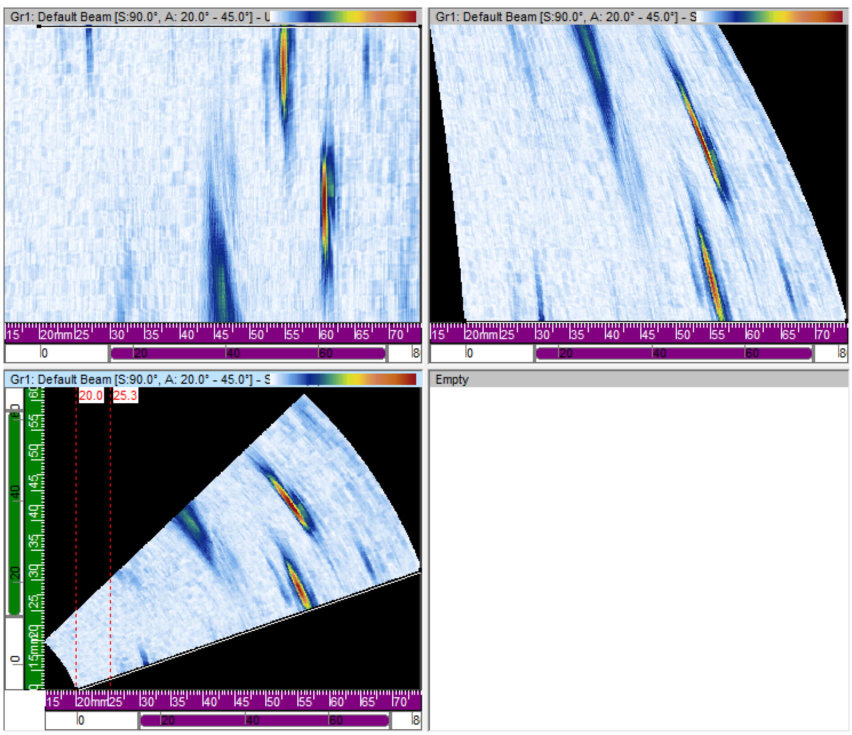


Figure 3-21 Example of Uncorrected (top-left), USound corrected (top-right) and VC sectorial (bottom-left) scans

The three sectorial scan view types are:

Uncorrected sectorial scan

View where the A-scan of each focal law (see top-left view of the example in Figure 3-21 on page 101) is represented by a horizontal line on which the amplitude is color-coded. The displayed real-time data shows the focal laws stacked (vertically in the example) in the order that they were generated.

Sectorial scan (Usound corrected)

View similar to the uncorrected sectorial scan except that the A-scans are corrected for delay and true depth so that their positions are accurate relative to the ultrasonic axis (see top-right view of the example in Figure 3-21 on page 101).

VC sectorial scan

Volume-corrected view similar to the uncorrected sectorial scan except that the A-scans are corrected for delay and refracted angle so that the positions are accurate relative to the ultrasonic and scan axes. A typical S-scan sweeps through a range of angles using the same focal distance and elements. The horizontal axis corresponds to the projected distance (test-piece width) from the exit point for a corrected image and the vertical axis corresponds to the depth (see bottom-left view of the example in Figure 3-21 on page 101).

When an azimuthal scan is defined by the calculator, the sectorial-scan view represents an angular sector where each line of this view corresponds to the A-scan of a different angle. Therefore, when a linear scan is defined, the sectorial-scan view represents the beam movement. Each line then corresponds to an A-scan of different aperture. Finally, when a depth scan is defined, the sectorial-scan view then represents the beam focusing at different depths. Each line then corresponds to a different A-scan.

3.7.1.2 Volumetric Views

Volumetric views are color-coded images built from successive A-scans projected on different planes defined by the ultrasound, scan, and index axes. The most important views, similar to 2-D projections of a technical drawing, are presented in Figure 3-22 on page 103.

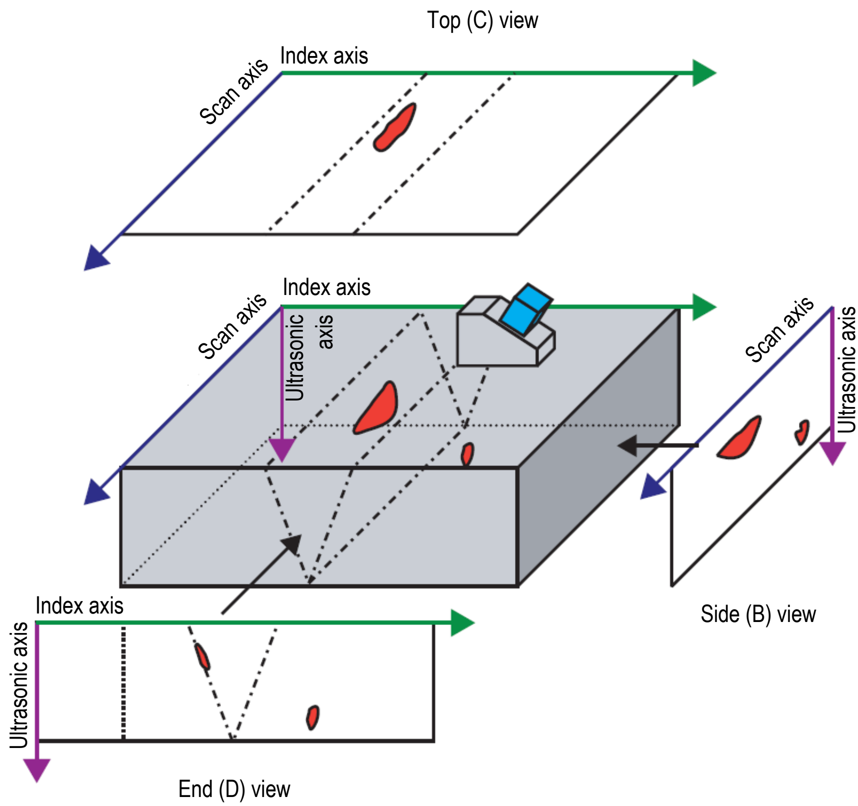


Figure 3-22 Example of ultrasonic views [Top (C), Side (B), and End (D)] with probe skew angle of 90°

In Figure 3-22 on page 103, if the probe skew angle is 0° (or 180°), the Side (B) view becomes the End (D) view, and vice versa. The Side (B) view is defined by the depth and probe-movement axes. The End (D) view is defined by the depth and the electronic-scan axis.

Side (B)

The **Side (B)** view (see Figure 3-23 on page 104) is a two-dimensional graphical representation of the recorded data. One of the axes is the scan axis; the other is the uncorrected ultrasonic (USound) path. The position of the displayed data is related to the encoder positions at the moment of acquisition. At a given position on the projected image, the color corresponds to the maximum amplitude at this position as detected in the considered index-axis range.

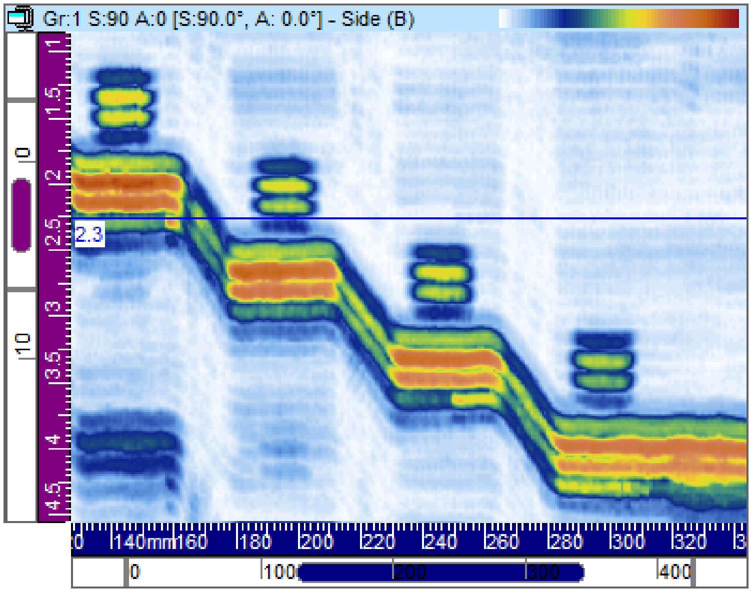


Figure 3-23 Example of Side (B) view

Top (C)

The **Top (C)** view (see Figure 3-24 on page 105) is a two-dimensional graphical representation of the recorded data displayed as a top view of the test specimen. One of the axes is the scan axis; the other is the index axis. The position of the displayed data is related to the encoder positions at the moment of acquisition. At a given position on the projected image, the color corresponds to the maximum amplitude at this position as detected in the considered true-depth range.

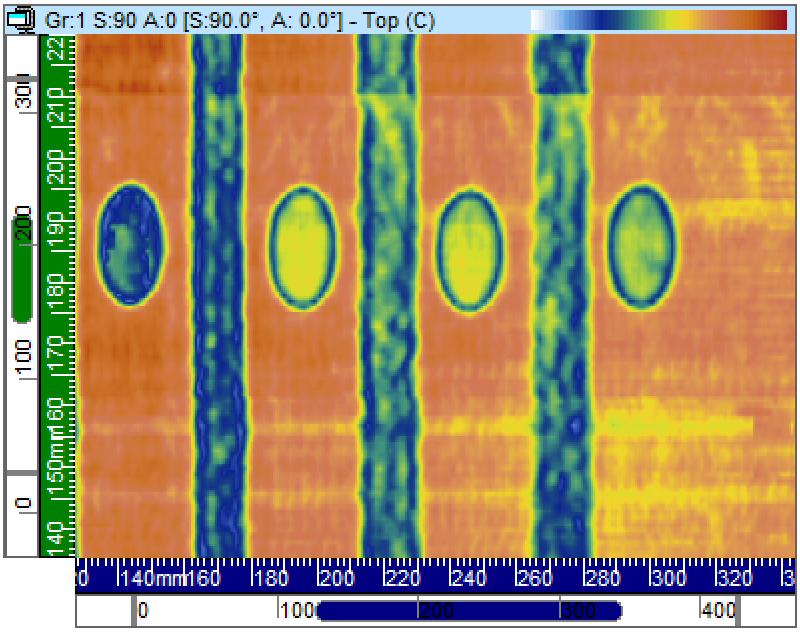


Figure 3-24 Example of Top (C) view

End (D)

The **End (D)** view (see Figure 3-25 on page 106) is a two-dimensional graphical presentation of the recorded data. One of the axes is the defined index axis; the other is the uncorrected ultrasonic (USound) path. The position of the displayed data is related to the encoder positions at the moment of acquisition. At a given position on the projected image, the color corresponds to the maximum amplitude at this position as detected in the considered scan-axis range.

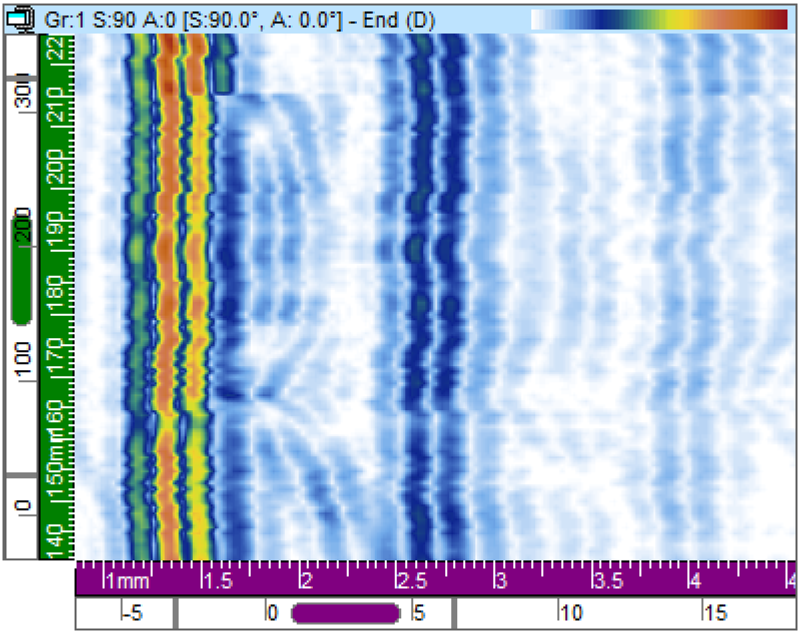


Figure 3-25 Example of End (D) view

Polar View

NOTE

The Polar view is available only when the geometry of the specimen is identified as cylindrical. This can be done by selecting **Part and Overlay > Part Definition** in the menu, and defining the part as **Cylindrical**.

The Polar view (see Figure 3-26 on page 107) is a two-dimensional representation of the recorded data in a realistic cylindrical geometry. It is used in analysis mode. The polar coordinates **ID depth** or **OD depth** (radial coordinate) and **Position** (angular coordinate) are calculated taking into account the previously defined specimen.

Depending on the orientation of the scan axis relative to the cylindrical geometry, and the skew angle of the considered probe, the polar view is the cylindrical equivalent of either the VC-Side (B) view or the VC-End (D) view. Either distance units (mm or in.) or rotational units (°) can be used and displayed in the circumferential direction of the cylinder.

NOTE

The polar view is only supported for skew angles of 0°, 90°, 180°, and 270°; for data with other skew angle values, however, the **Cylindrical Correction View Information** can be used to calculate the correct position and size of indications.

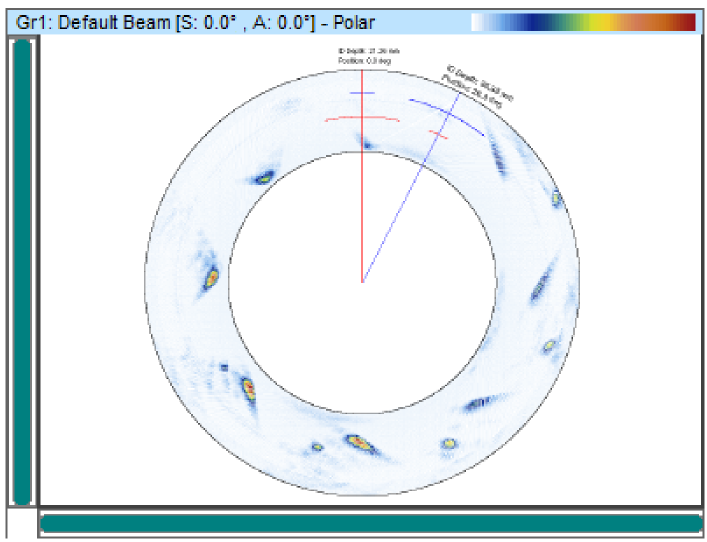


Figure 3-26 Example of a Polar View

3.7.1.3 Scrolling Views

Scrolling views add data dynamically, from an active acquisition unit to a scrolling axis.

Scrolling B-Scan

In the **Scrolling B-Scan** view (see Figure 3-27 on page 108), each A-scan is represented by a horizontal line on which the amplitude is color-coded. The lines are added consecutively in real time, from the bottom, so that the resulting image scrolls up. Therefore, the data view shows the real time vertically, versus the time of flight of the received ultrasonic pulse horizontally.

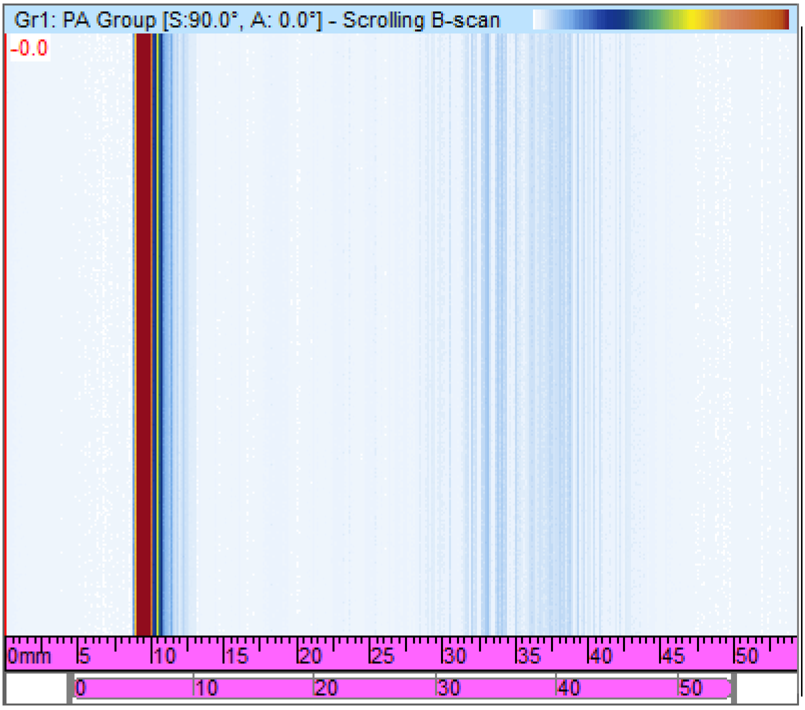


Figure 3-27 Example of Scrolling B-Scan view

Strip Charts - Amplitude and Position Scrolling Views

NOTE

The amplitude and/or position scrolling views are only available when the options for the recording of the selected amplitude and/or position data are enabled in the **Gates** section of the **UT Settings** dialog bar (see section 3.10.2 on page 123).

In an amplitude or position scrolling view, the data of the signal crossing the associated gate is represented by a color-coded **Scrolling View** which is displayed inside a **Strip Chart** view. The **Scrolling View** can be configured using the **Configuration** tab of the **View and Pane Properties** dialog bar (available only when a Strip Chart view is selected).

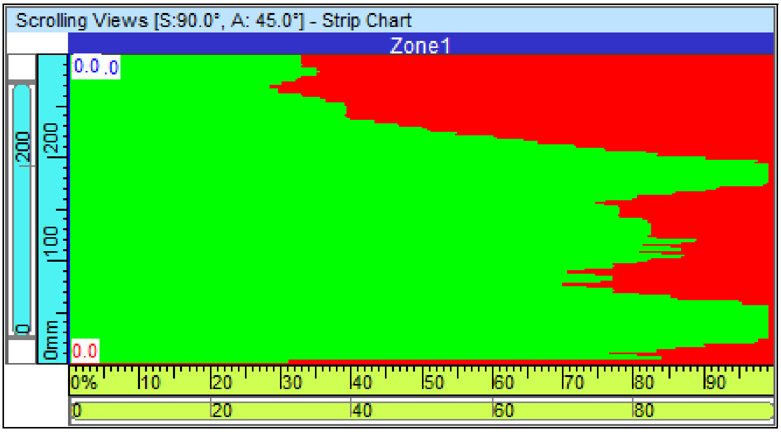


Figure 3-28 Example of Scrolling View

3.7.2 View Contextual Menu

Once a data view type is assigned to a view, display options are available from a contextual menu. The contextual menu appears when you right-click on the title bar of a view (see example shown in Figure 3-29 on page 110).

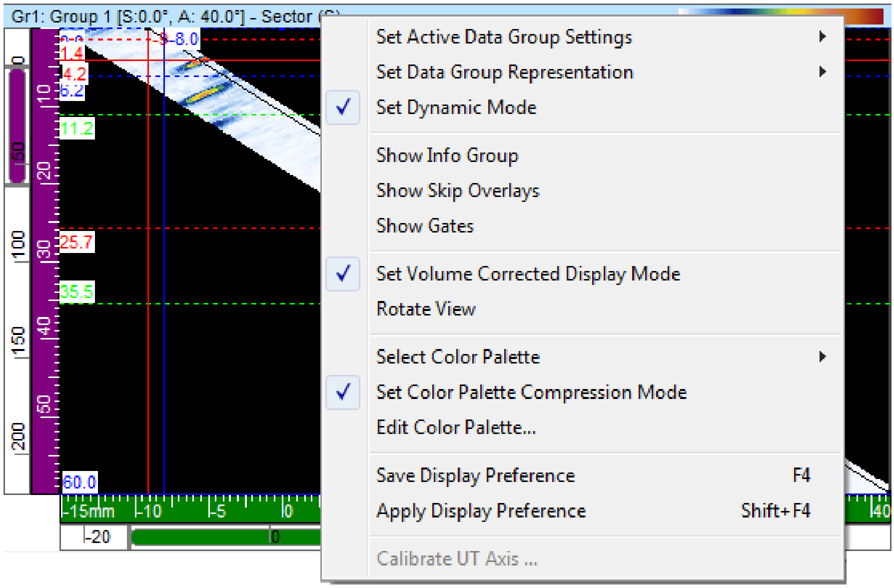


Figure 3-29 Example of the contextual menu for a view

The view contextual menu changes depending on the view type. The possible view contextual menu commands are the following:

Data commands

Set Active Data Group Settings

Provides one or more choices (**Active Group**, **Active Law**, and **Active Gate**) to set the view to show data for the active item (group, beam, or gate), automatically following changes of the active selection.

Set Single Slice (Projection)

Toggles between single- and projection-data display in the view. The command is also available in the **Data Source** tab of the **View Properties** dialog box.

Set Data Group Representation

Provides one or more choices for sectorial scans (**Sector (S)** and **TOF**), and for C-scans (**Stacked**, **Single Beam**, **Strip**, and **Scrolling Strip**).

Restore Initial Gates

Only available in Analysis mode, this selection is used to reposition all gates either for the current group or for all groups to the positions where they were when the data was originally acquired.

Set Dynamic Mode

Only available in setup and Inspection mode, this selection is used to toggle between the dynamic state and the analysis state (offline).

Show commands

Show Info Group

Toggles the appearance of the information groups below the title bar of the view.

Show Gate Selectors

Toggles the appearance of the gate selectors on the selected view.

Show Zoombar and Rulers

Toggles the appearance of the zoom bars and the rulers. The command is also available in the **Display** tab of the **View Properties** dialog box.

Show Echo Dynamics

Toggles the appearance of the echo-dynamics signals next to the relevant axis. The echo-dynamic curves show the maximum amplitude (or minimum position) between the measurement and reference cursors (see example shown in Figure 3-30 on page 112). The command is also available in the **Echo Dynamics** tab of the **View Properties** dialog box.

Show Skip Overlays

Toggles the appearance of the overlay lines representing the skips. The command is also available in the **Overlay** tab of the **View Properties** dialog box.

Show Rebounds

Toggles the appearance of the rebounds (see example shown in Figure 3-30 on page 112).

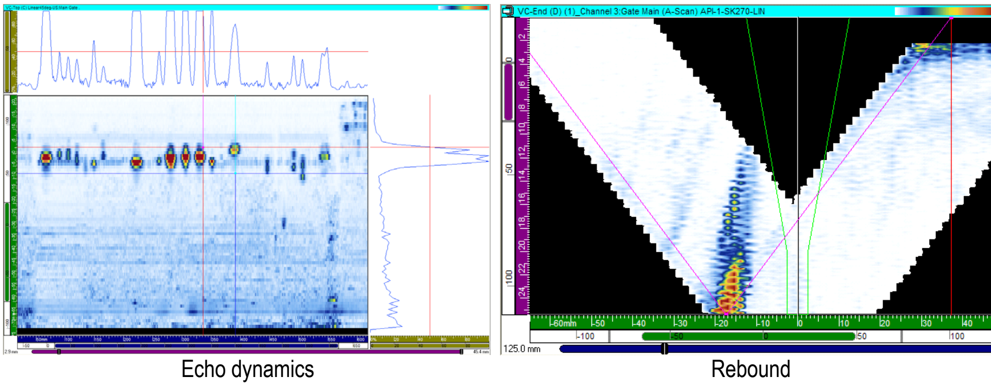


Figure 3-30 Examples of shown echo dynamics curves and rebound

Show Gates

Toggles the appearance of the gates. This option is only available for A-scan and S-scan views. The command is also available in the **Overlay** tab of the **View Properties** dialog box.

Display mode commands

Set 1:1 Ratio Display Mode

Displays the view with the same scale on both axes. The command is also available in the **Display** tab of the **View Properties** dialog box.

Set Smoothing Display Mode

Enables the smoothing function. The command is also available in the **Display** tab of the **View Properties** dialog box.

Set Volume Corrected Display Mode

Toggles the A-scan correction for delay and refracted angle so that the positions are accurate, relative to the ultrasonic and scan axes.

Rotate View

Rotates the data so that the two axes interchange. The command is also available in the **Display** tab of the **View Properties** dialog box.

Color palette commands

The following contextual menu commands are also available from the **Palette** tab of the **View Properties** dialog box. These commands are available on the menu only when the view supports a color palette.

Select Color Palette

Used to select one of the available color palettes (**Rainbow**, **Gray**, **Thickness**, or **Balanced**).

Set Color Palette Compression Mode

Toggles the state of the color palette compression. When you reduce the range for the 256 colors of a palette without compression, you remove colors from the palette. With compression, all palette colors are compressed in the new range.

Edit Color Palette

Opens the **Palette Editor** dialog box, where you can edit existing color palettes or create new color palettes.

Display preferences commands

Save Display Preference

Saves the current view properties as the default properties.

Apply Display Preference

Applies the default view properties to the currently selected view.

TIP

On the keyboard, press F4 and SHIFT-F4 to respectively activate the **Save Display Preference** and the **Apply Display Preference** commands.

Ultrasound axis commands

Reverse USound Axis

Reverses the direction of the ultrasound axis. The command is only available for the A-scan view. The command is also available from the **Display** tab of the **View Properties** dialog box.

Change USound True Depth / Half Path

Toggles the ultrasound axis unit between true depth and half path. The command is only available for the A-scan view when the ultrasound axis is set to either true depth or half path.

Change USound TOFD / Time

Toggles the ultrasound axis ruler between TOFD and time. The command is only available for the A-scan view when the ultrasound axis is set to either time of TOFD.

TIP

You can set the type and the units of **USound** axis in the **Units** tab of the **View Properties** dialog box.

Calibrate UT Axis

Available only in Analysis mode, opens the dialog box for the calibration of the **True Depth**, the **Half Path**, or the **TOFD** on the ultrasound axis. The command is also available from the **Units** tab of the **View Properties** dialog box and in Setup mode, from the **General** tab of the **UT Settings** dialog box.

3.8 About Readings and Information Groups

TomoView computes reading values for various parameters to help you analyze your ultrasonic data. Readings are calculated using cursor, zone, acquisition, or segment (3-D Cursor) parameters.

You can select to display one or more groups of readings at the top of a view (see Figure 3-31 on page 115).

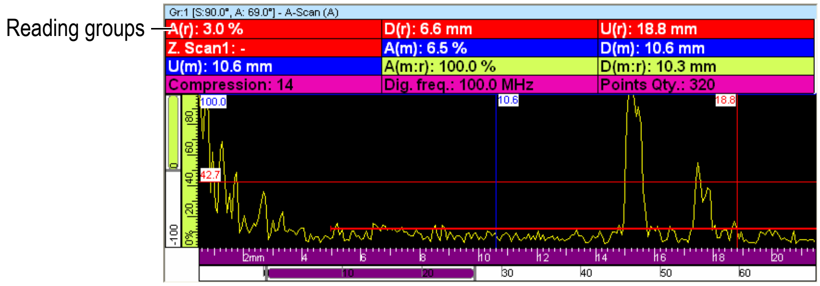


Figure 3-31 Example of reading groups appearing at the top of a view

Refer to section 4.10 on page 276 for more information on how to customize the reading groups.

3.9 About Gates

A gate is a signal processing tool that isolates a time domain region of the received ultrasonic signal on which further processing is performed. In an A-scan view, a gate appears as an horizontal line ending at both ends with short vertical lines. The vertical position of the gate line indicates the signal detection threshold (see Figure 3-32 on page 116). When the rectification of the receiver is set to RF, the threshold of a gate can be positive or negative.

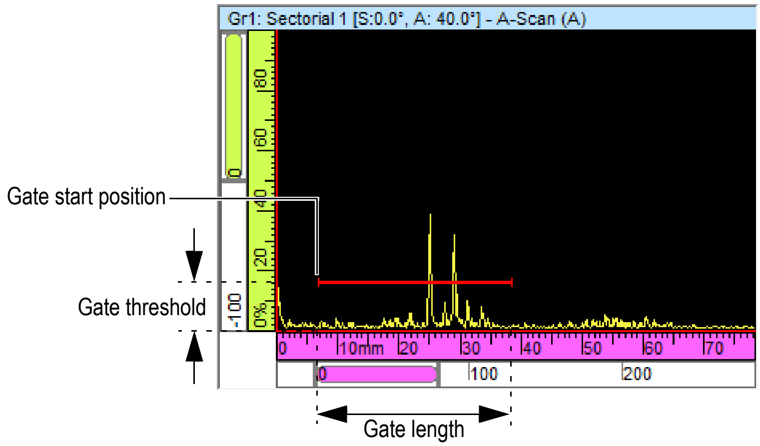


Figure 3-32 Example of gate A in an A-scan

In an S-scan view, the gate start and end positions appear as horizontal dashed lines (see Figure 3-33 on page 116). When the time base mode is set to true depth, the area in between is the gated zone for all beams. When the time base mode is set to half path, the area between the dashed lines is the gated zone for the current beam only. The dashed lines of the gate automatically move to the appropriate location when you change the current beam.

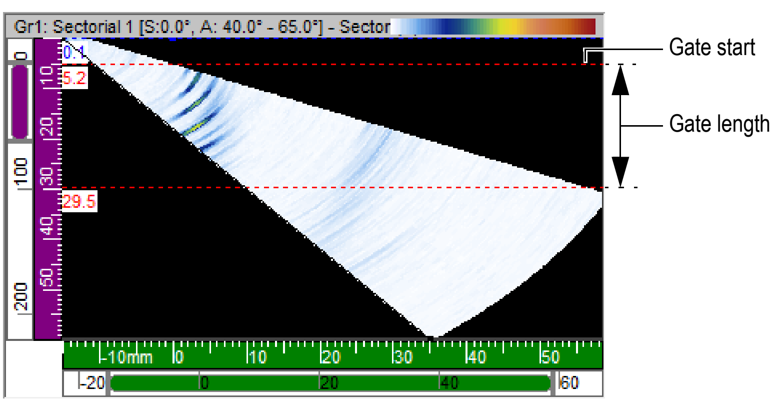


Figure 3-33 Example of gate A in an S-scan

TomoView supports up to five gates (gate I, gate A, gate B, gate C, and gate D), allowing you to perform complex signal processing. The function of gate I, the interface gate, is to identify the interface of the inspected part. Gate A, B, C, and D are general purpose gates (see section 4.11.5 on page 287). Each gate has its own color to easy identification (see Figure 3-34 on page 117).

Figure 3-34 Gate colors

NOTE

Gate C and gate D are available only with a Focus LT acquisition unit.

3.10 UT Settings Dialog Box

Clicking the UT Settings button () on the **Main** and **TomoView Manager** toolbars toggles the visibility of the **UT Settings** dialog box. The **UT Settings** dialog box contains eight basic tabs and two additional tabs: **General**, **Gate**, **TCG**, **Digitizer**, **Pulser/Receiver**, **Probe**, **Alarms**, and **I/O** (see Figure 2-22 on page 70). When you use a phased array unit such as the Focus LT, this dialog box also includes two additional tabs: **Transmitter** and **Receiver**. For more details on the different tabs of the UT Settings dialog box, see section 3.10 on page 117.

This section describes the different tabs available in the UT Settings dialog box.

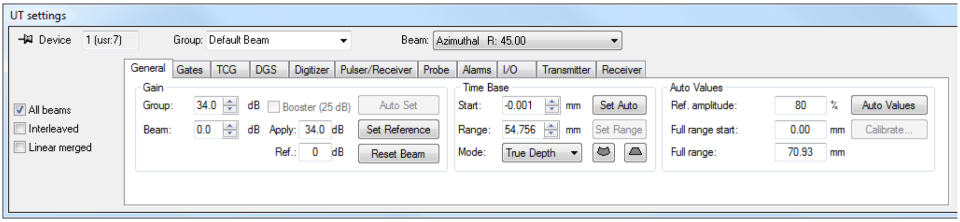


Figure 3-35 The UT Settings dialog box

All beams

When this checkbox is activated, modifying a parameter causes all focal laws to be affected by the modification.
If this checkbox is not selected, modifying a parameter causes only the active focal law to be affected by the modification.

Interleaved

Used to change the firing order of the different focal laws (see section 4.14 on page 289 for more information on using this function).

Linear merged

Selecting this check box activates the linear merged mode, which performs the dynamic merging of the individual firings. (For proper data display, this mode should only be used with 0° linear focal laws).

3.10.1 General Tab

The **General** tab (see Figure 3-36 on page 118) contains options that are used to configure the basic ultrasonic parameters.

Description of the General tab

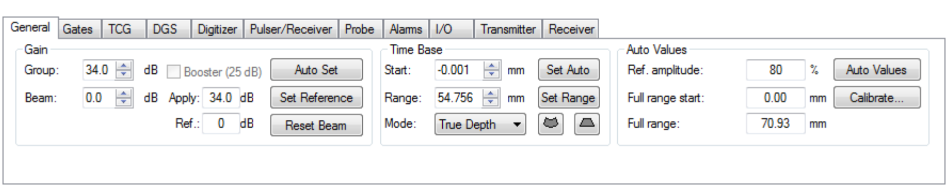


Figure 3-36 The General tab

The **General** tab is subdivided into the **Gain, Time base** and **Auto Values** group boxes.

Gain group box

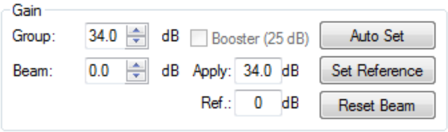


Figure 3-37 The Gain group box

Group

Sets the gain value in decibels (dB) for the receiver of the active group. The group gain is added to the beam gain (when a phased array unit is connected to the system). See Figure 3-37 on page 119.

When the TCG (time-corrected gain) function is used, the gain programmed in the TCG curve (see section 3.10.3 on page 127) is added to this gain value to produce the total input gain.

Beam

Sets the gain value in decibels (dB) for the active beam's receiver.

Apply

This box displays the applied gain, considering the sum of the group and focal law gain.

Ref.

This box displays the reference gain that you have set by using the **Set Reference** button.

Auto Set

This button automatically sets the amplitude of the echo between the cursors to the value specified in the **Ref. amplitude** box.

Set Reference

This button transfers the **Apply** value into the **Ref.** box.

Reset Beam

This button resets the gain of the active beam to 0 dB.

Time base group box

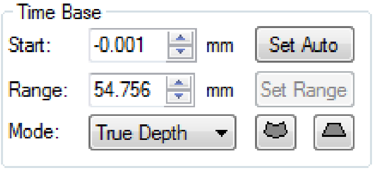


Figure 3-38 The Time base group box

Start

Sets the delay in the material at which the acquisition starts. This value can be displayed in distance or time units, according to the unit chosen in the **Mode** drop-down list box. See Figure 3-38 on page 120.

Range

Sets the acquisition range in the material (according to the unit set in the **Mode** drop-down list box).

Note: The maximum range value is determined by the maximum number of samples allowed for an A-scan.

Mode

Selects the unit mode used to set the time base:

- **Time**: Time base values are expressed in ultrasonic time-of-flight units: **µs** (microseconds).
- **Full path**: Time base values are expressed in distance units: **in.** (inches) or **mm** (millimeters). The value entered is equal to the time of flight multiplied by the ultrasonic velocity in the material.
- **Half path**: Time base values will be expressed in distance units: **in.** (inches) or **mm** (millimeters). The values are equal to half the total distance traveled by the ultrasonic wave (**Full path** divided by 2).
- **TOFD**: Time base values will be expressed in distance units, in function of the TOFD calibration: **in.** (inches) or **mm** (millimeters).
- **True depth**: Time base values will be expressed in distance units: **in.** (inches) or **mm** (millimeters). The calculated true depth is a function of the beam angle in the material.

This button appears in a phased array setup when **True depth** is selected in the **Mode** drop-down list box. Clicking this button takes the true depth **Start** and **Range** values of the *active focal law* and applies the corresponding half path **Start** and **Range** values to all focal laws in the active group.

This button appears in a phased array setup when **True depth** is selected in the **Mode** drop-down list box. Clicking this button takes the true depth **Start** and **Range** values of the active focal law and applies them to *focal laws in the active group*.

Set Auto

This button sets the ultrasonic scale (**Start** and **Range** of the **Time base** group box) according to the values specified in the **Full Range Start** and **Full Range** boxes of the **Auto Values** dialog box.

Set Range

This button sets the ultrasonic scale (**Start** and **Range** of the **Time base** group box) according to the positions of the reference and measurement cursors in an online A-scan view.

Auto Values group box

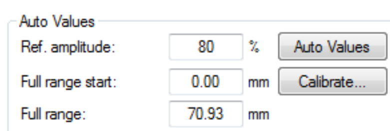


Figure 3-39 The Auto Values group box

Ref. amplitude

Indicates the reference amplitude that has been specified in the **Auto Values** dialog box (opened with the **Auto Values** button). See Figure 3-39 on page 121.

Full range start

Indicates the full range start on the ultrasonic axis, according to the value specified in the **Auto Values** dialog box.

Full range

Indicates the distance on the ultrasonic axis corresponding to the full range, according to the value specified in the **Auto Values** dialog box.

Auto Values

This button opens the **Auto Values** dialog box, used to define the automatic values. See Figure 3-40 on page 122.

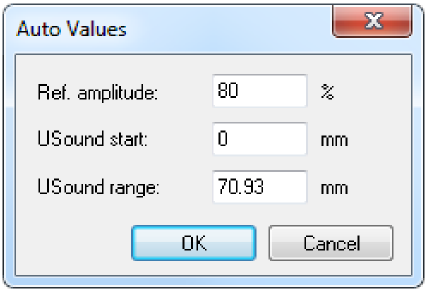


Figure 3-40 The Auto Values dialog box

In this dialog box, enter the Ref. **Amplitude, Ultrasound start** and **Ultrasound range** values that should be programmed when the **Set Auto** button is clicked. Click **Ok** to accept values, or **Cancel** to return to previous values.

Calibrate

This button opens a dialog box, used to calibrate the ultrasonic velocity or delay according to the reference cursor position. See Figure 3-41 on page 123.

To use this function, select a side view (corrected for the angle) and then place the cursors on two reflectors at known positions and click the **Calibrate** button. In the calibration dialog box that pops up, enter the know depths of the two selected reflectors and the values shown in the Results section of this dialog box will be calculated automatically. To accept these values, click **Ok**. TomoView will program the calculated values.

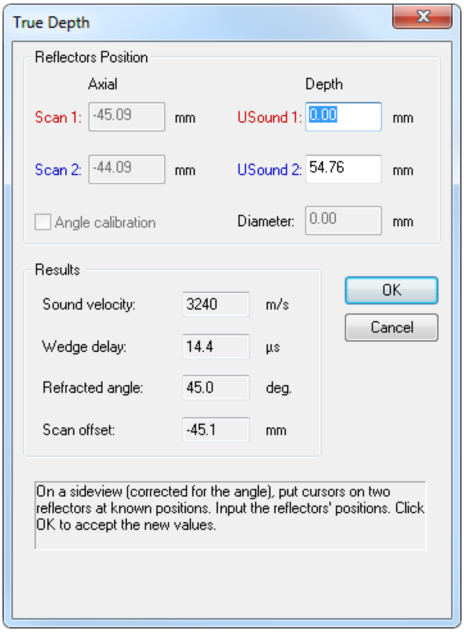


Figure 3-41 The Calibrate dialog box in True Depth mode

3.10.2 Gate Tab

The **Gate** tab (see Figure 3-42 on page 124) contains options that are used to define the detection gate parameters. Detection gates are used to extract information from the A-scan and form the C-scan data groups.

Whenever a gate is enabled, it is represented on the corresponding A-scan view by a horizontal line with small vertical lines on both ends. In addition, each time a detection gate is defined and enabled, a new data group is added in the left section of the **Contents** dialog box.

You can define four detection gates for each channel, in addition to the synchronization gate. Gate I is the synchronization gate, and gates A, B, C, and D are the acquisition gates. The A-scan signal that is viewed and acquired can be synchronized on pulse or on echo in the main gate. Each gate is defined by a starting position, a gate length, a threshold level, and an alarm level. Detection occurs when a portion of the signal exceeds the threshold level during the time interval specified by the starting position and gate length.

The setting of the gates can be done either by filling the **Start** and **Length** boxes for each gate, or by positioning both cursors (reference and measurement) on the A-scan and clicking the set button of the gate, or interactively. The basic procedure to follow for the definition of the gates is described later in this section. For the procedure to define gates see section 4.11 on page 282.

Description of the Gates tab

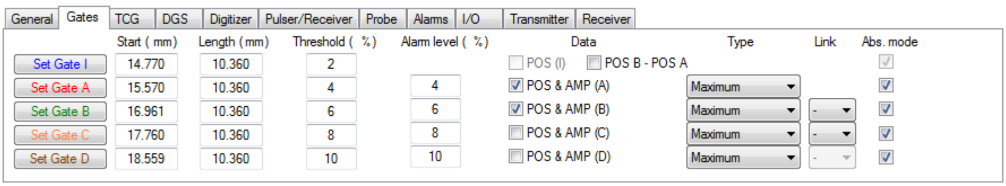


Figure 3-42 The Gates tab

The **Gates** tab contains the following group boxes:

Set Gate buttons

Sets the start position, the length, and the threshold of synchronization of the selected gate according to the position of the reference cursor and the measurement cursor in time, and the reference cursor in level.

Start

Sets the start position of the corresponding gate. The position is expressed in the unit chosen in the **Mode** drop-down list box of the **General** tab.

Length

Sets the length of the corresponding gate. The length is expressed in the unit chosen in the **Mode** drop-down list box of the **General** tab.

Threshold

Sets the threshold level of the corresponding gate. The level is expressed as a percentage (%) of the full screen height.

Alarm level

Sets the alarm level of the corresponding gate. The level is expressed as a percentage (%) of the full screen height.

POS and AMP checkboxes

When enabled, the position and amplitude data of the corresponding gate will be recorded.

POS B - POS A

When enabled, a differential-position data group in which the position of a signal above gate A threshold is subtracted from the position of a signal above gate B threshold will be recorded.

Type

This drop-down combo box allows the user to select one of the following detection types according to the detection gate:

Maximum

The position and amplitude data of the detected maximum signal peak inside the gate will be recorded (respectively P1 and A1 in Figure 3-43 on page 125).

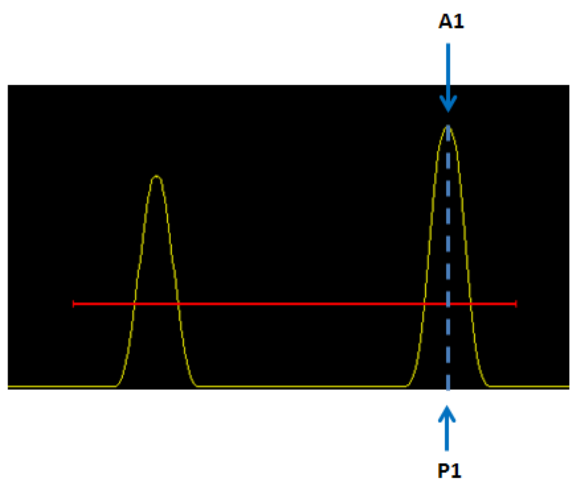


Figure 3-43 Maximum gate detection

Crossing

The position data of the first peak crossing the gate and the amplitude data of the maximum detected signal peak inside the gate will be recorded. (respectively P1 and A1 in Figure 3-44 on page 126).

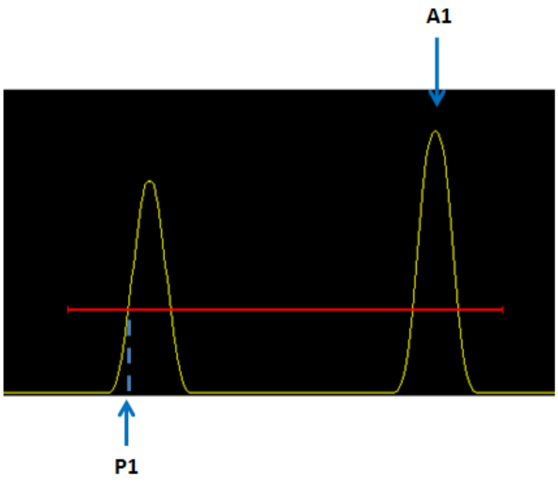


Figure 3-44 Crossing gate detection

First Peak Maximum

The position and amplitude data of the first signal peak detected inside the gate will be recorded (respectively P1 and A1 in Figure 3-45 on page 126).

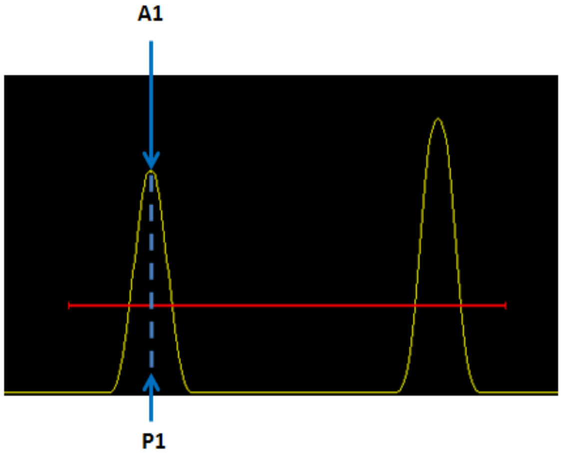


Figure 3-45 First peak maximum gate detection

First Peak Crossing

The position data of the first peak crossing the gate and the amplitude data of the first detected signal peak inside the gate will be recorded (respectively P1 and A1 in Figure 3-46 on page 127).

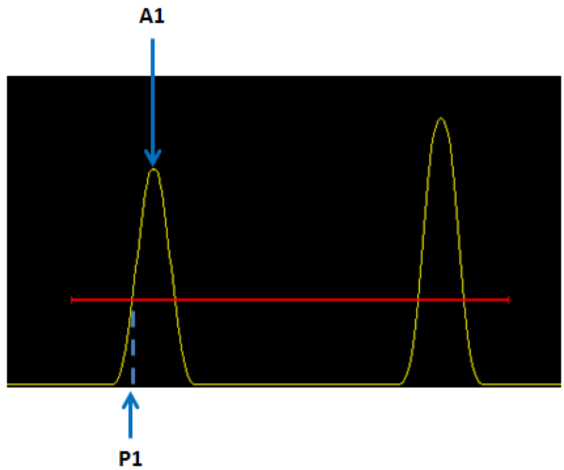


Figure 3-46 First peak maximum gate detection

Link

This drop-down combo box can be used to synchronize the start of the corresponding gate with respect to the previous gate.

3.10.3 TCG Tab

The **TCG** tab (see Figure 3-47 on page 128) contains the options that can be used to set the time-corrected gain (TCG) for the current group. The TCG function operates by modifying the receiver gain during data acquisition to compensate for the attenuation of the ultrasonic wave in the material. The TCG curve defines the gain values that are added to the group gain.

Description or the TCG tab

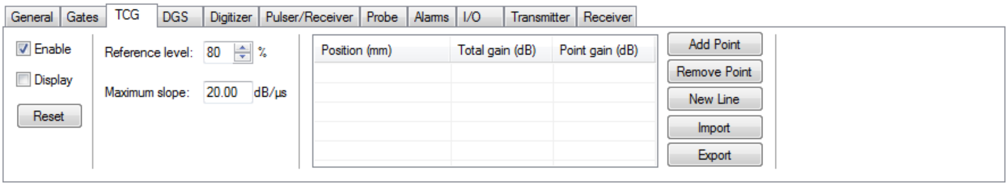


Figure 3-47 The TCG tab

The **TCG** tab contains the following group boxes:

Enable

This check box alternately activates and deactivates the TCG function according to the values set on the **TCG** tab.

Display

This check box alternately displays and hides the TCG curve in an area covering 25 % of the A-scan view.

Reset

This button resets the TCG curve, by clearing all currently defined points.

Ref. level

Sets the wanted reference level, in percent, for the next point.

Maximum slope

Displays the maximum slope between two TCG points.

Position

Indicates the position of the TCG points on the ultrasonic axis. These points are represented by red dots when the TCG curve is displayed at the top of the corresponding A-scan view. The position is expressed in the unit chosen in the **Units** box of the **Units** tab of the **Pane Properties** dialog box.

Gain

Indicates the gain level for the TCG points.

Add Point

This button adds a new point to the TCG curve which will be placed in a position corresponding to the maximum amplitude that is read between the reference and measurement cursors. The gain associated with the TCG point will correspond to the theoretical gain required to reach the reference level.

Remove Point

This button removes the selected point from the grid. The TCG curve is then recalculated according to the remaining points. The gain and slope parameters are updated for the current position.

New Line

This button adds a new point to the TCG curve, following the rightmost point.

Import/Export

These buttons allow the user to export the TCG curve applied to the current group or import a TCG curve from a .csv file. For more information on the TCG curve import/export function, see section 4.7.4 on page 260.

3.10.4 DGS tab

The **DGS** tab contains the parameters that can be used to configure a DGS curve. The distance gain size (DGS) is method for sizing indications based on a calculated DGS curve for a given transducer, material, and a known reflector size. For more information on how to use the DGS, see section 4.9 on page 267.

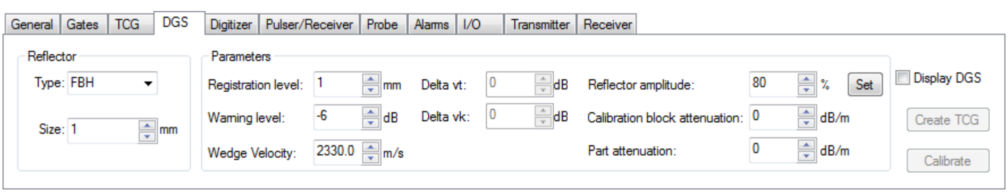


Figure 3-48 The DGS tab

3.10.5 Digitizer Tab

The **Digitizer** tab (see Figure 3-49 on page 130) contains options that are used to set the basic digitizer and data-acquisition parameters. The **Digitizer** tab is subdivided into the **Digitizing**, **PRF**, **Data Sample Size**, **Data**, and **Multipeak** group boxes.

Description of the Digitizer tab

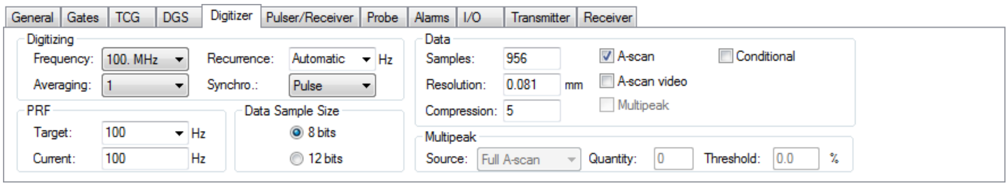


Figure 3-49 The Digitizer tab

The **Digitizer** tab (see Figure 3-50 on page 130) contains the following:

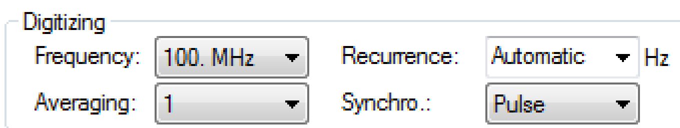


Figure 3-50 The Digitizing group box

Frequency

Allows the user to select the digitizer frequency, specified in megahertz (MHz).

Recurrence

The recurrence is the firing recurrence frequency (pulse repetition frequency, or PRF) which is defined independently for every group. The effective recurrence is the number of ultrasonic pulses generated per second.

Averaging

Indicates the number of A-scans acquired for each A-scan that is returned. The returned A-scan is obtained by calculating, for each sample, the average value over all the acquired A-scans.

The averaging can be used to reduce the effect of noise; however, using a great number of acquisitions for the averaging will decrease the maximum acquisition rate.

Synchro

Selects the synchronization mode:

- **Pulse**: the acquisition is synchronized on the ultrasonic firing pulse.
- **Echo**: the acquisition is synchronized on the echo crossing gate 1.
- **External**: the acquisition is synchronized by an external signal.

The **PRF** group box (see Figure 3-51 on page 131) contains the following:

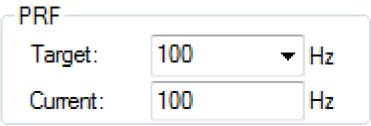


Figure 3-51 The PRF group box

Target

This box can be used to select whether or not the PRF should be set to the maximum possible value in hertz, or to a custom defined value which TomoView will try to match as closely as possible.

Current

Indicates the current PRF value in hertz.

Data Sample Size group box (see Figure 3-52 on page 131):

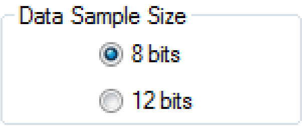


Figure 3-52 The Data Sample Size group box

The selected data sample size indicates the resolution of the amplitude digitalization. This resolution will influence the size of an elementary data sample and therefore the size of the resulting data file.

The **Data** group box (see Figure 3-53 on page 132) contains the following choices:

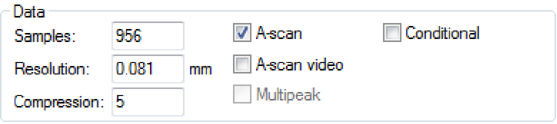


Figure 3-53 The Data group box

Samples

Indicates the number of samples per data acquisition, that is, the number of points that compose the A-scan. This number is directly related to the **Range** (**General** tab) and **Digitizing frequency** values. If the **Units** parameter is in time (microseconds), multiplying the **Range** value (in seconds) by the **Digitizing frequency** value (in hertz) will give the number of points. This has an effect on the **Compression**.

Resolution

Indicates the digitizer resolution in the UT scale unit.

Compression

This box is used to set the digitizer compression ratio. You can reduce the sample quantity, and thus the data file size, by keeping the position of the first sample with the maximum amplitude (see example shown in Figure 3-54 on page 133). This box is only available for certain acquisition units.

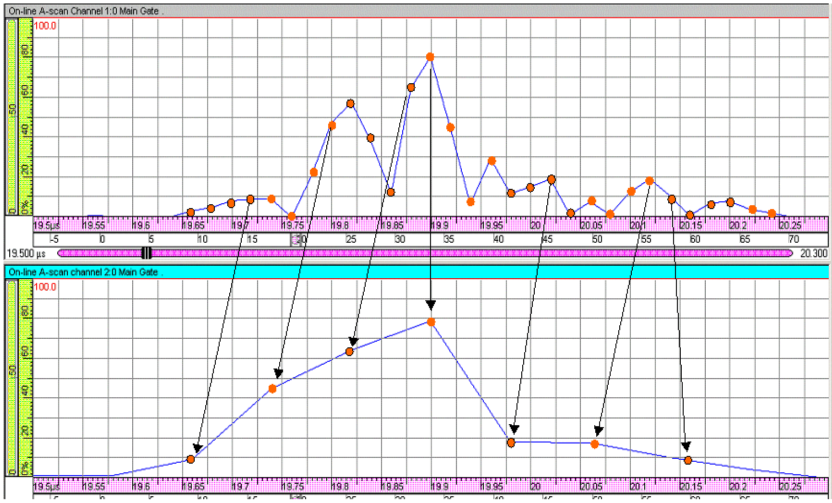


Figure 3-54 Example of a compression ratio of 4

A-scan

Creates the A-scan data group.

A-scan video

Creates the video A-scan data group. (This data cannot be acquired.)

Multipeak

Creates a compressed A-scan showing only peak data.

Conditional

Selecting this option will allow the A-scan to be recorded only when an alarm was triggered. For more details about this function, see section 4.15 on page 292.

The **Multipeak** group box (see Figure 3-55 on page 133) has the following choices:

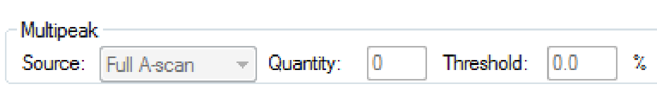


Figure 3-55 The Multipeak group box

Source

Selects the peak detection source (this box is available only if the **Peak** check box is selected).

Quantity

Sets the number of peaks used to produce the **A-scan** peak data group. This box is available only if the **Peak** check box is selected.

Threshold

Sets the level for the peak detection.

3.10.6 Pulser/Receiver Tab

The **Pulser/Receiver** tab (see Figure 3-56 on page 134) contains options that you can use to configure the pulser and receiver used for the inspection.

Description of the Pulser/Receiver tab

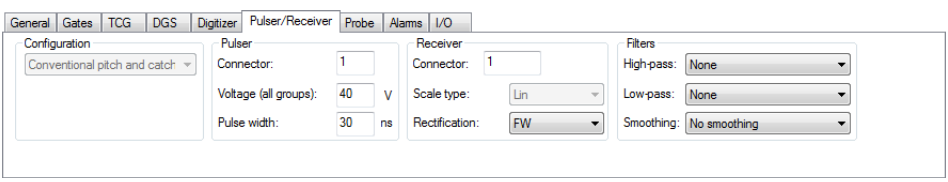


Figure 3-56 The Pulser/Receiver tab

The **Pulser/Receiver** tab is subdivided into the following sections: **Configuration**, **Pulser**, **Receiver,** and **Filter** group boxes. See Figure 3-56 on page 134.

Configuration group box

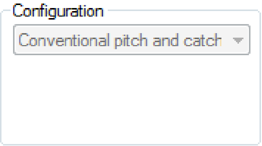


Figure 3-57 The Configuration group box

Configuration

Indicates the current group configuration from the following choices: **Conventional pitch-and-catch, Phased array pulse-echo, Phased array pitch-and-catch, Conventional pulse-echo**, and **TOFD**.

Pulser group box

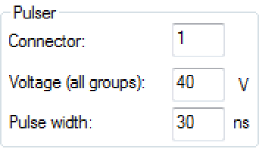


Figure 3-58 The Pulser group box

Connector

For a Conventional UT group, this box allows you to select the connector that will be used for the pulser. For a Phased array group, it indicates the number of the first element used in emission.

Voltage (all groups)

Sets the pulse voltage of the pulser, which is the same for all groups, in volts (V).

Pulse width

Sets the pulse width of the pulser in nanoseconds.

Receiver group box

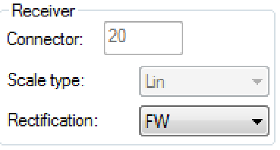


Figure 3-59 The Receiver group box

Connector

For a Conventional UT group, this box allows you to select the connector that will be used for the receiver. For a Phased array group, it indicates the number of the first element used in reception.

Scale Type

Indicates the receiver type between **LIN** (linear receiver), **LOG** (logarithmic receiver), and **LOGDAC** (receiver with a logarithmic DAC function).

Rectification

Allows you to select the rectification type between **RF** (the A-scan signal is displayed without any rectification), **HW+** (only the positive part of the signal is kept), **HW-** (only the negative part of the signal is kept), and **FW** (the negative part of the signal is brought back to the positive part).

Filters group box

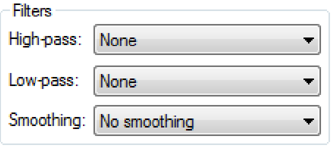


Figure 3-60 The Filters group box

High-pass

Selects the frequency of the high-pass filter used to filter the group signal.

Low-pass

Selects the frequency of the low-pass filter used to filter the group signal.

Smoothing

Selects the video filtering of the rectified signal in order to smooth the curve. The smoothing function creates an envelope on the rectified signal to eliminate the effects of the original RF signal (alternating positive and negative amplitudes).

3.10.7 Probe Tab

The **Probe** tab (see Figure 3-61 on page 137) contains options that are used to set the parameters of the probes used for the inspection.

Description of the Probe tab

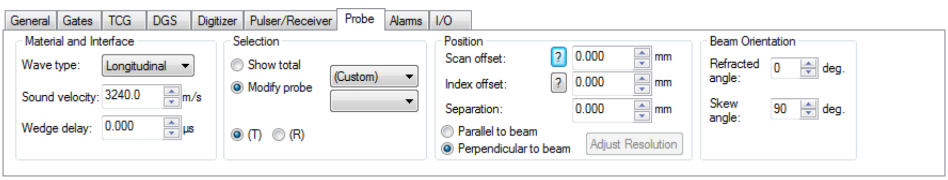


Figure 3-61 The Probe tab

The **Probe** tab is subdivided into **Material and Interface, Selection, Position**, and **Beam Orientation** group boxes.

Material and interface group box

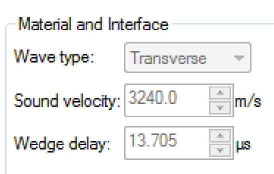


Figure 3-62 The Material and Interface group box

Wave type

Allows you to select the wave type: **Longitudinal** or **Transverse**.

Sound velocity

Sets the sound velocity in the inspected material for each wave type. This value must be determined with precision as it will be used to convert time units into distance units.

Wedge delay

Sets the total wedge delay of the probe. The wedge delay is calculated as follows:

$$\text{Wedge delay} = \frac{\text{Ultrasound path in wedge}}{\text{Sound velocity in wedge}}$$

Selection group box

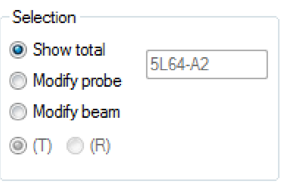


Figure 3-63 The Selection group box

Show total

Disables the probe and law parameters modification.

Modify probe

Allows you to modify the probe parameters.

Modify beam

Allows you to modify the beam parameters.

T/R radio buttons

When a **Pulse-Echo** group is selected, the **T** button transfers the pulse-receiver probe parameters. When a **Pitch-and-catch**, or a **TOFD** group is selected, the **T** button transfers the pulser probe parameters and the **R** button transfers the receiver probe parameters.

Position group box

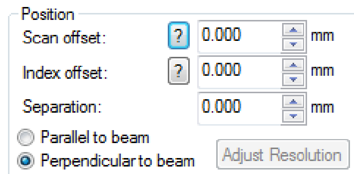


Figure 3-64 The Position group box

Scan offset

The scan offset represents the distance between the beam direction and the scan origin, which can be positive or negative (click on the ? button to view a graphical representation of this offset). This box sets the scan offset of the probe or the beam depending on whether the **Modify probe** or **Modify beam** option was chosen in the **Selection** group box. See Figure 3-64 on page 139.

Index offset

The index offset represents the distance between the beam exit point and the index origin, which can be positive or negative (click on the ? button to view a graphical representation of this offset). This box sets the scan offset of the probe or the beam depending on whether the **Modify probe** or **Modify beam** option was chosen in the **Selection** group box.

Separation

Specifies the distance between the probes (available only with an active TOFD group).

Parallel/perpendicular to beam radio buttons

Specifies the beam orientation as parallel or perpendicular to the scan direction (available only with an active TOFD group).

Adjust Resolution

This button opens the dialog box shown in Figure 3-65 on page 140 which can be used to adjust the data group resolution.

(1) **Min.**

Sets the minimal offset difference, ensuring that there will be no data fusion among the data groups, although acquisition gaps may occur.

(2) **Max.**

Sets the minimal offset difference, ensuring that there will be no acquisition gap among the data groups, although data fusion may occur.

(3) **Average**

Sets an average of the offset difference, therefore providing a good compromise between acquisition gaps and data fusion among data groups.

(4) **User**

Sets a user-defined offset difference, allowing you to select a data group resolution between the minimum and maximum offset differences.

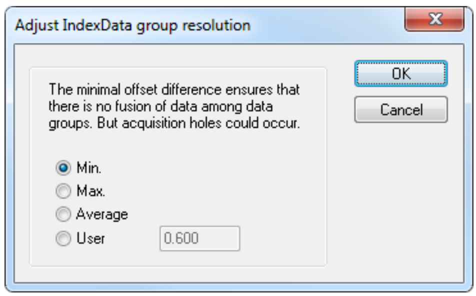


Figure 3-65 The Adjust Scan data group resolution dialog box

Beam orientation group box

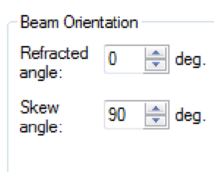


Figure 3-66 The Beam orientation group box

Refracted angle

Defines the refracted angle of the probe beam.

The refracted angle (β) (see Figure 3-67 on page 141) is calculated from the probe incidence angle (α), sound velocity in wedge, and sound velocity in material according to the following formula (Snell's law):

$$\frac{\sin\alpha}{\sin\beta} = \frac{\text{Sound velocity in wedge}}{\text{Sound velocity in material}}$$

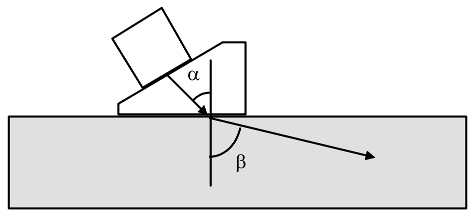


Figure 3-67 Refracted angle

Skew angle

Sets the skew angle of the probe, which is defined as the angle between the primary axis of the probe and the scan axis. For more details on the probe orientation convention, see section 3.5 on page 89.

3.10.8 Alarms Tab

The **Alarms** tab (see Figure 3-68 on page 141) contains options that are used to define the alarms. For more information on defining the alarms, see section 4.12 on page 288.

Description of the Alarms tab

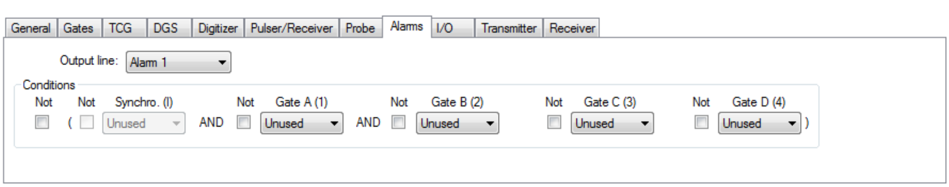


Figure 3-68 The Alarms tab

The **Alarms** tab contains the following group boxes:

Output line

Selects the alarm output line to be used when the alarm condition is met.

Conditions group box

Not

When selected, these check boxes add the operator NOT to the next condition to the right.

Synchro

This box is used to select the logical operator relative to the synchronization gate (**Gate I**): **Unused** and **Detect**.

Gates A, B, C, and D

These boxes are used to select the conditions relative to gates A, B, C, and D (**Unused** and **Detect**).

NOTE

Alarm conditions are defined independently for each channel.

3.10.9 I/O Tab

The **I/O** tab (see Figure 3-69 on page 142) contains options that are used to enable and disable the analog input and output lines. Description of the I/O tab

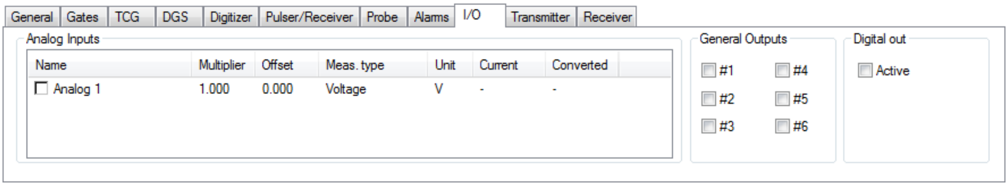


Figure 3-69 The I/O tab

The **I/O** tab is subdivided into the following group boxes: **Analog Inputs, General Outputs**, and **Digital out**.

Analog inputs group box

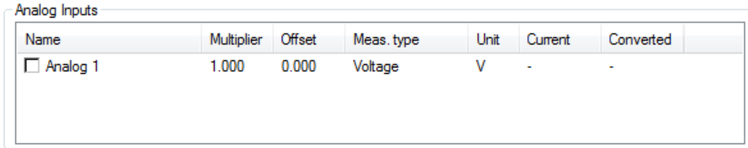


Figure 3-70 The Analog inputs group box

Name

Selecting these check boxes enables the corresponding analog input line. The name of the analog input line can be edited by double-clicking the current name. See Figure 3-70 on page 143.

Multiplier

Defines the multiplier value to convert the current analog input value (in volts) to a meaningful value.

Offset

Defines the offset value to convert the current analog input value (in volts) to a meaningful value. The offset value is added after the **Multiplier** value.

Measure type

Defines the type of measurement made with the associated analog input line.

Unit

Defines the unit of the converted value of the associated analog input line.

Current

Displays the current value (in volts) of the associated analog input line.

Converted

Displays the converted value of the associated current value of the analog input line.

General outputs group box

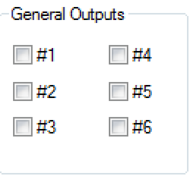


Figure 3-71 The General outputs group box

General outputs

Selecting the check boxes enables the corresponding general output lines (only available for the MS5800 acquisition unit). See Figure 3-71 on page 144.

Digital out group box

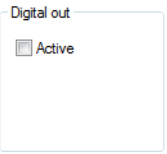


Figure 3-72 The Digital out group box

Active

Selecting this checkbox enables the **External trig** (Trig out) function, which sends an external signal to the Digital Output No. 2 (DOUT2) each time an individual focal law is fired.

NOTE

The Digital Output No. 1 (DOUT1) is connected to the **External Pace** (Pace out) function, which sends an external signal before each set of focal laws is fired. This function is always activated, whether or not the **Active** checkbox is enabled. For example, if firing on encoders, then each time the encoder changes position, the firing sequence will be triggered and a signal sent to DOUT1.

3.10.10 Transmitter Tab

The **Transmitter** tab (see Figure 3-73 on page 145) is used to specify the parameters of each element of a focal law in transmission.

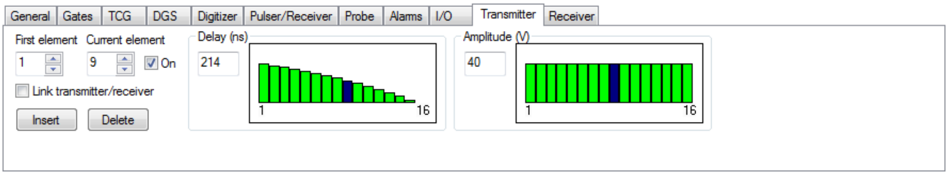


Figure 3-73 The Transmitter tab

The **Transmitter** tab contains the following group boxes:

First element

Specifies the first active element of the transmitter in the considered focal law.

Current element

Selects an element of the probe as the current element. You can also select an element by clicking the corresponding bar on the **Delay** or **Amplitude** graphs.

On

Selecting or clearing this check box respectively turns on or off the element in transmission.

NOTE

Holding down the SHIFT key while clearing the **On** check box turns off all the elements in transmission at once.

Link transmitter/receiver checkbox

When this check box is selected, modifying the **First Element** value for the transmitter will also set the same value for the receiver.

Insert

Used to insert an element to be used as a transmitter in the active focal law. The new element is inserted before the currently selected element.

Delete

Deletes the currently selected transmitter element from the active focal law.

Delay (ns)

The **Delay** box is used to specify the transmission delay applied to the current element in nanoseconds (ns).

The bar graph represents the transmission delay law for the different elements. The blue bar indicates the element being modified, that is, of which the parameters are displayed. The green bars indicate the delays of the other active elements. The inactive elements are left blank.

Amplitude (V)

The **Amplitude** box is used to specify the pulse amplitude applied to the element in volts (V).

3.10.11 Receiver Tab

The **Receiver** tab (see Figure 3-74 on page 146) is used to specify the parameters of each element of a focal law in reception.

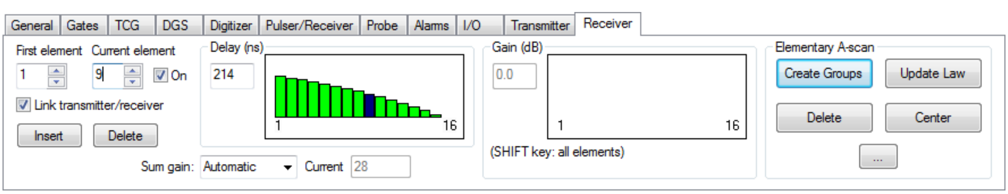


Figure 3-74 The Receiver tab

The **Receiver** tab contains the following:

First element

Specifies the first active element of the receiver in the considered focal law.

Current element

Selects an element of the probe as the current element. You can also select an element by clicking the corresponding bar on the **Delay** or **Gain** graphs.

On

Selecting or clearing this check box respectively turns on or off the element in reception.

NOTE

Holding down the SHIFT key while clearing the **On** check box turns off all the elements in reception at once.

Link transmitter/receiver checkbox

When this check box is selected, modifying the **First Element** value for the receiver will also set the same value for the transmitter.

Insert

Used to insert an element to be used as a receiver in the active focal law. The new element is inserted before the currently selected element.

Delete

Deletes the currently selected receiver element from the active focal law.

Sum gain

This combo box allows you to select whether the **Sum** gain should be automatically set or be manually defined.

Current

Indicates the current hardware **Sum** gain value.

Delay (ns)

The text box is used to specify the reception delay applied to the element in nanoseconds (ns).

The bar graph represents the reception delay law for the different elements. The blue bar indicates the element being modified, that is, of which the parameters are displayed. The green bars indicate the delays of the other active elements. The inactive elements are left blank.

Gain (dB)

The text box is used to specify the gain applied to the element in decibels (dB).

The bar graph represents the amplitude law in reception for the different elements. The blue bar indicates the element being modified, that is, of which the parameters are displayed. The green bars indicate the amplitudes of the other active elements. The inactive elements are left blank.

3.11 Scan and Mechanical Settings Dialog Bar

The following section describes the **Scan and Mechanical Settings**.

3.11.1 Description of the Sequence tab

Clicking the Scan and Mechanical Settings button ( ) on the Main and TomoView Manager toolbars toggles the visibility of the Scan and Mechanical Settings dialog box. The Scan and Mechanical Settings dialog bar contains four tabs: **Scan**, **Scan Controls**, **Encoders**, and **Options** (see Figure 3-75 on page 148). When a MCDU-02 is detected, this dialog box also includes two additional tabs: **MCDU Control** and **MCDU I/O**. See section 3.12 on page 162.

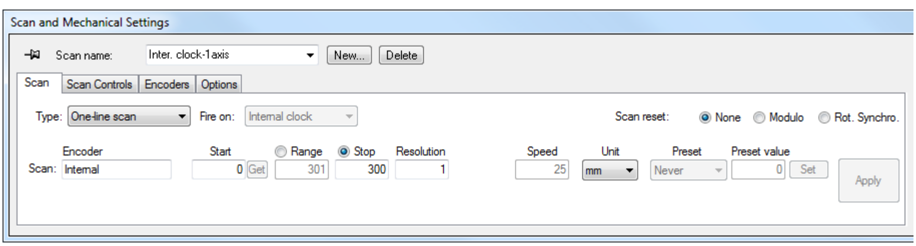


Figure 3-75 The Sequence tab for Bidirectional sequence

Type

This list box is allows you to select the sequence type to be used for the inspection from the following choices: **One-line scan**, **Free running**, **Bidirectional**, **Unidirectional**, **Helicoidal**, **Angular**, and **Custom**.

These inspection types are described in sections 5.1.1 on page 295 to 5.1.6 on page 304.

Fire on

This list box is used to select the trigger signal used to fire the ultrasonic pulses from **Encoder**, **External signal**, **Internal clock**, and **Absolute** position.

Scan reset

Allows you to select the Scan axis reset options between the following options:

None

The Scan axis is never reset.

Modulo

The Scan axis is reset when the Scan encoder reaches the **Stop** position.

Rot. Synchro

The Scan axis is reset when an external signal is received on the corresponding instrument I/O connector (also known as *top tour*).

Index preset

Allows you to select the Index axis preset options from the following options:

None

No preset at acquisition end.

At acquisition end

When the acquisition is stopped, the **Start** value for the index axis is set to the current index axis encoder position.

Angle

If an **Angular** sequence type is chosen, the angle between the Mechanical axis and the Scan axis can be defined.

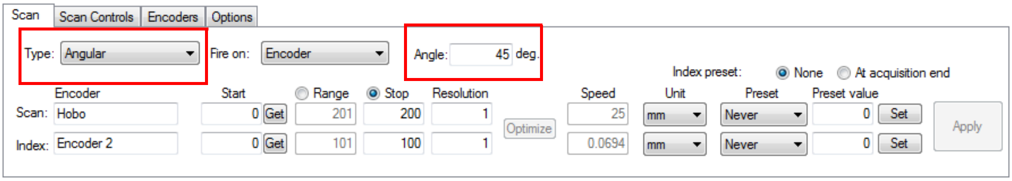


Figure 3-76 The Scan tab with Angular sequence type

Deg./Index

If a **Helicoidal** sequence type is chosen, this box indicates the number of degrees of rotation for every index position.

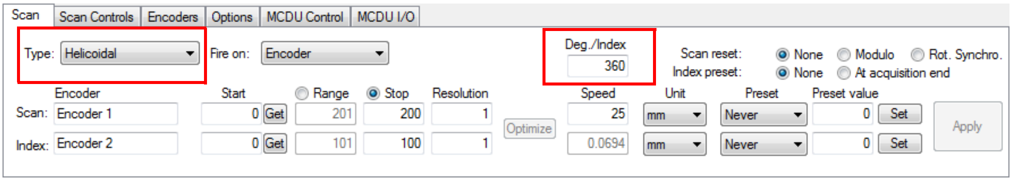


Figure 3-77 The Scan tab with Helicoidal sequence type

Encoder

These boxes indicate the name of the encoder associated with each axis. The encoder names can be changed in the **Encoder** tab.

Start

These boxes are used to set the position on the scan and index axes where the inspection sequence is to start. These values can be positive or negative.

Get

These buttons set the start value to the current encoder position for the associated axis.

Range

If the **Range** radio button has been selected, these boxes are used to set the range on the scan and index axes.

Stop

If the **Stop** radio button has been selected, these boxes are used to set the stop position on the scan and index axes.

NOTE

In the case of an encoder sequence, all acquisitions for which the position is outside the inspection surface defined by the **Start** values and **Stop** values are ignored.

Resolution

These boxes are used to set the acquisition resolution for the scan and index axes. The resolution of the scan axis determines the spacing between the acquisitions. The resolution of the index axis, in the case of a surface scan, determines the separation between the scan lines.

Optimize

This button is enabled when, in the **UT Settings** dialog bar, the **Linear merged** mode option is selected. Depending on the selected skew angle, the optimal resolution for the relevant axis is automatically calculated (0° and 180° = scan axis resolution, 90° and 270° = index axis resolution).

Speed

In slave mode, this parameter has no implication.

In master mode, these boxes are used to set the speed that the motors should follow during inspection.

Whenever the motor movement for the Scan axis controls the data acquisition, the Scan axis Speed defines the required acquisition rate:

$$\text{Acquisition rate} \geq \frac{\text{Scan axis speed}}{\text{Scan axis resolution}}$$

Unit

These boxes are used to select the unit of the corresponding axis in **mm**, **in.**, or **deg (°)**

Preset

This combo box allows you to select when the corresponding encoder is to be reset during the sequence:

Never

The corresponding encoder is never reset.

On start

The corresponding encoder is reset when the acquisition is started.

External

The corresponding encoder is reset when an external signal is received on the corresponding instrument's I/O connector.

Preset value

These text boxes specify the encoder positions to be used with the **Preset** option or the **Set** button.

Set

These buttons set the encoder positions to the **Preset value** positions for the associated axis.

Apply

This button applies the current tab settings to the inspection sequence that is selected in the **Scan name** list box.

3.11.2 Scan Controls Tab

The **Scan Controls** tab contains options and indicators related to the acquisition sequence (see Figure 3-78 on page 152)

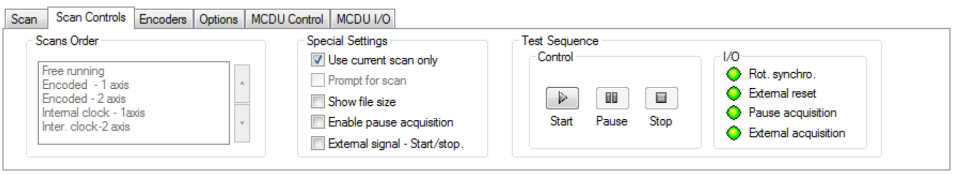


Figure 3-78 The Scan Controls tab

The **Scan Con**trols tab is subdivided into **Scans Order, Special Settings, Test Sequence**, and **I/O** group boxes.

Scans order group box

This group box lists the inspection sequences and their order of execution, and can be modified if the **Use current scan only** checkbox is not activated in the **Special Settings** group box.

You can modify the sequence order by moving the selected inspection sequence up and down using the  or the  buttons.

Use current scan only

When this check box is selected, only the selected inspection sequence is performed during the acquisition. The current sequence can be selected in the **Scan name** list box.

Prompt for scan

When this check box is selected, the **Inspection Parameters for Scan** dialog box is displayed at the acquisition start (see Figure 3-79 on page 153) which you can use to skip a scan as well as to check and modify the sequence parameters if needed.

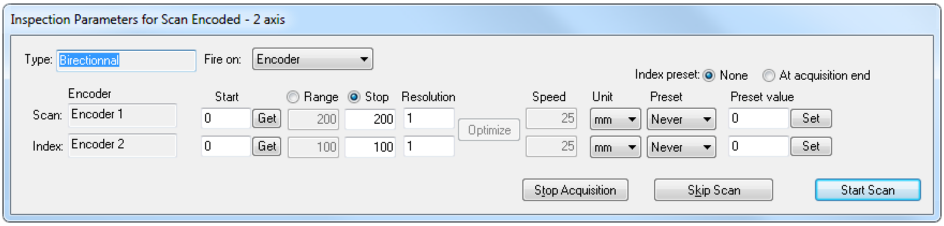


Figure 3-79 The Inspection Parameters for Sequence dialog box

The **Inspection Parameters for Sequence** dialog box contains the same options as the **Sequence** tab, except that the sequence type cannot be saved (see section 3.11.1 on page 148 for more details).

Stop Acquisition

Click this button to stop the inspection.

Skip Scan

Click this button to skip the current sequence and present the next sequence as defined in the **Sequences order** group box of the **Sequence Controls** tab. If the current sequence is the only one in the **Sequences order** group box, or the last one, the acquisition process is stopped.

Start Scan

Click this button to start the inspection.

Show file size

When this check box is selected, a message box appears at the acquisition start to indicate the estimated data file size that is required according to your acquisition parameters. See Figure 3-80 on page 153.

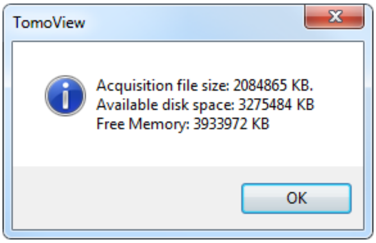


Figure 3-80 The File Size dialog box

Enable pause acquisition

By selecting this check, the inspection sequence can be paused during the acquisition by an external signal.

External signal - Start/stop

Activates the digital input number 4 (DIN4) pin, for the start/stop of...

Test sequence group box

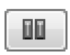
This box groups command buttons used when an MCDU-02 unit is controlled by TomoView (master mode only).

Control section

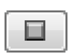
Start

The  button starts the inspection and can be used to resume the sequence if the pause button is activated.

Pause

The  button pauses the inspection.

Stop

The  button stops the inspection.

NOTE

If the **Use current scan only** checkbox is not selected in the **Special settings** combo box, and the inspection is stopped using the **Stop** button inside the **Test Sequence** combo box, after the data file acquired with the first sequence has been saved, TomoView will automatically return to inspection mode with the second sequence in the **Sequences** order list. However, if the inspection is stopped with the  button on the **TomoView Manager** toolbar, all the unfinished sequences are discarded.

I/O section

The indicators in this section give the user the current status of **Rot. synchro, External reset, Pause acquisition** and **External acquisition** input/output signals.

3.11.3 Encoders Tab

The **Encoders** tab contains options related to the configuration of the encoders (see Figure 3-81 on page 155).

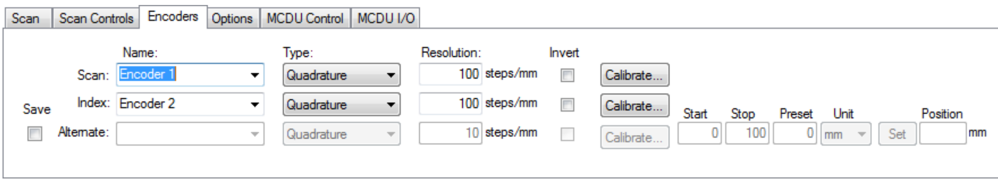


Figure 3-81 The Encoders tab

The **Encoders** tab contains the following options:

Save

When selected, this check box enables the system to record in a C-scan the position of a third encoder (alternate).

Name

These boxes are used to select the encoder associated with the **Scan** and the **Index** axes, and the **Alternate** encoder.

You can edit the default encoder names but not when **Internal** is selected.

Type

These boxes are used to select the encoder type from **Pulse** + (positive pulse) **Pulse** - (negative pulse), **Quadrature,** and **Clock Dir**. (clock direction).

Resolution

This box is used to set the encoder resolution, that is, the number of encoder steps needed to move 1 mm (or 1 in.) in linear units, or 1° in angular units.

Invert

When one of these check boxes is selected, the sign of the corresponding encoder data is automatically inverted.

Calibrate

This button displays the **Calibration of Encoder** dialog box, used to calibrate the encoder resolution for the corresponding axis (see Figure 3-82 on page 156).

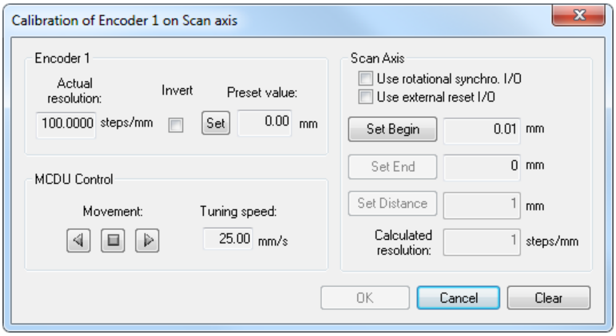


Figure 3-82 The Calibration of Encoder dialog box

The **Calibration of Encoder** dialog box is subdivided into **Encoder, MCDU Control**, and **Scan Axis** group boxes.

Actual resolution

This box indicates the actual encoder resolution set in the **Resolution** box of the **Encoders** tab of the **Scan and Mechanical Settings** dialog bar.

Invert

This check box, when selected, inverts the count direction of the encoder.

Set

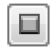
This button is used to set the encoder position to the value indicated in the **Preset value** box.

Preset value

This box displays the value entered in the **Preset value** box on the **Sequence** tab.

Figure 3-83 The MCDU dialog box

Movement

These buttons are disabled in slave mode. In master mode, these buttons are used to manually control, with the mouse, the scanner movement on each axis.The  button moves the scanner backward, the  button moves the scanner forward, the  button stops the movement along the corresponding axis. See Figure 3-83 on page 156.

Tuning speed

This box indicates the maximum speed the scanner can reach when you are using the **Movement** buttons. This value is set on the **MCDU Control** tab.

Scan axis

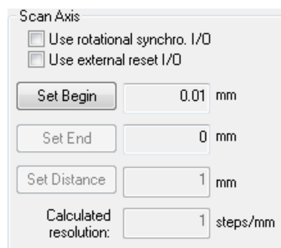


Figure 3-84 The Scan axis group box

Use rotational synchro I/O

Select this check box to use an external rotation synchronization input/output signal to set the beginning and end of motion used for the calibration. See Figure 3-84 on page 157.

Use external reset I/O

Select this check box to use an external reset input/output signal to set the beginning and end of motion used for the calibration.

Set Begin

This button sets the current position as the beginning of the motion used for the calibration.

Set End

This button sets the current position as the end of the motion used for the calibration.

Set Distance (box)

This box indicates the distance covered by the motion on the axis to perform the calibration. When the motion is done, enter the appropriate distance value.

Set Distance (button)

This button is used to calculate the encoder resolution so that the actual position matches the expected distance you specify in the text box.

NOTE

The units for the scan and index axes are the same as the ones used on the **Encoders** tab, which are defined on the **Scan** tab.

Calculated resolution

This box gives the encoder resolution calculated with the calibration.

OK

This button applies the calculated resolution in the **Resolution** box of the **Encoders** tab of the **Inspection Sequences and Mechanical Settings** dialog bar, and closes the dialog box.

Cancel

This button closes the dialog box without applying the calculated resolution.

Clear

This button resets the parameters of the dialog box to the default values.

Start

This text box indicates the start limit that defines the color palette associated with the third (**Alternate**) encoder position. In the alternate encoder C-scan view, all encoder values below the start value then have the same color.

Stop

This text box indicates the stop limit that defines the color palette associated with the third (**Alternate**) encoder position. In the alternate encoder C-scan view, all encoder values below the stop value then have the same color.

Preset

This text box specifies the encoder position to be used with the **Set** button.

Unit

This text box is used to select the unit of the corresponding alternate encoder. The following units are available: mm, in., or degree.

Set

This button sets the alternate encoder position to the preset value position.

Position

This box indicates the current position of the alternate encoder.

3.11.4 Options Tab

Using the **Options** tab (see Figure 3-85 on page 159), you can set the automatic file saving options that are used when performing acquisition of ultrasonic data with the considered inspection sequences.

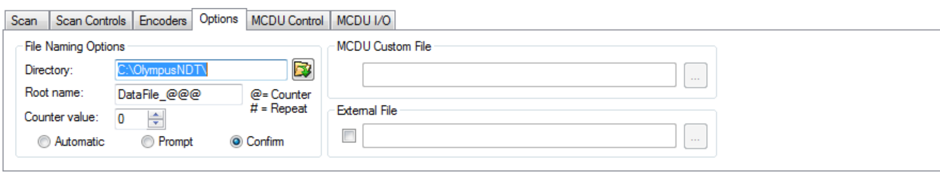


Figure 3-85 The Options tab

The **Options** tab is subdivided into **File Naming Options, MCDU Custom File** and **External File** group boxes. See Figure 3-86 on page 159.

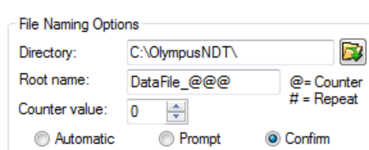


Figure 3-86 The File Naming Options group box

Directory

Using this text box, you can specify the directory, with its complete path, that is then used for the automatic file naming. You can enter the directory name either by typing it directly into the text box, or by clicking and selecting the directory in the **Browse for Folder** dialog box that appears.

Root name

Using this text box, you can specify the root name that is to be used for the automatic file naming.

- The @ character inserts a counter, which will be automatically incremented in the file names. For example, typing **test@** will produce test0.rdt, test1.rdt, test2.rdt, and so on.
- The # character adds the desired number of digits for repeats. For example, typing **test@##** will produce test000.rdt, test100.rdt, test200.rdt, and so on.
- If a file already exists (for example, test000.rdt), then the new file will be saved with the following name: test001.rdt (first repeat of the test000.rdt file name).

Counter value

This box sets the start value of the counter that is inserted in the file name with the @ character.

Automatic

Selecting this option button activates the automatic file naming that does not require your confirmation.

Prompt

Selecting this option button deactivates the automatic file naming. With this option, the **Save As** dialog box appears at the end of the inspection sequence, prompting you to enter the file name you want.

Confirm

Selecting this option button activates the automatic file naming that requires your confirmation. With this option, the **Save As** dialog box appears at the end of the inspection sequence, prompting you to confirm the file name defined in the **Root name** box.

Figure 3-87 The MCDU Custom File group box

MCDU Custom File

When working in master mode, if the **Custom Scan** was selected in the **Sequence** tab, this button opens a dialog box to modify the selected .gal file. This button is disabled in slave mode. See Figure 3-87 on page 160.

Figure 3-88 The External File group box

External File group box

Selecting this check box specifies the automatic file naming parameters (**Root name**, **Directory**) by an external text file. See Figure 3-88 on page 161. The external .txt file must have the following syntax:

[ACQUISITION]

FILENAME="Root name"

PATH="Directory"

See Figure 3-89 on page 161.

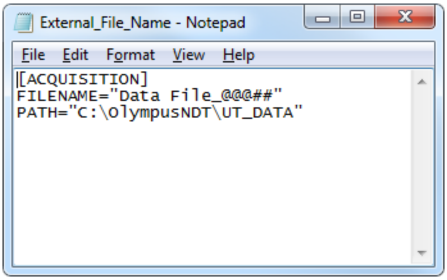


Figure 3-89 Example of an External File in a Notepad window

3.12 Scanner Parameters (Master Mode Only)

This section describes the options required to set the scanner parameters with TomoView. You must set the scanner parameters at least once for each application performed with a given scanner, but in most cases this should require only minor modifications.

The **MCDU Control** tab of the **Scan and Mechanical Settings** dialog bar contains the options used to display and manually control the position of the motors or encoders required in a sequence. This tab and its options are available in the TomoView window when an MCDU-02 unit is detected.

3.12.1 MCDU Control Tab

The **MCDU Control** tab contains the options to set the scanner parameters and manually control the position of the motors or encoders required in a sequence.

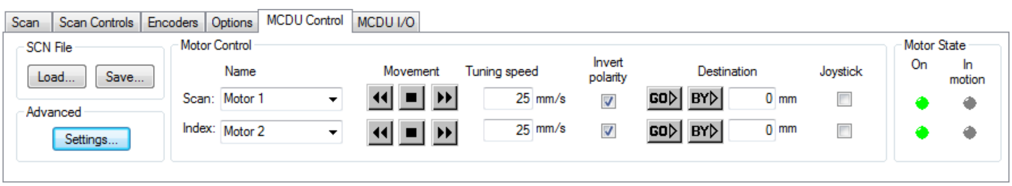


Figure 3-90 The MCDU Control tab

The **MCDU Control** tab is subdivided into **SCN File, Advanced, Motor Control**, and **Motor State** group boxes.

The **SCN File** text box:

Figure 3-91 The SCN File group box

Scn file: Load

This button opens the **Load Scanner** dialog box, used to select and load a scanner file (.scn). **MCDU Control** tab settings are updated according to the selected scanner file.

Scn file: Save

This button opens the **Save Scanner** dialog box, used to name and save the current **MCDU Control** tab settings in a scanner file (.scn).

The **Advanced** text box:

Settings button

This button opens the **Advanced MCDU Control** dialog box, used to set the parameters for each axis. See Figure 3-92 on page 163.

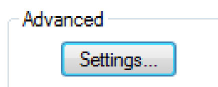


Figure 3-92 The Advanced group box

The **Advanced MCDU Control** dialog box (see Figure 3-95 on page 166) provides parameters related to servomotors. You can open the **Advanced MCDU Control** dialog box by clicking **Settings** in the **MCDU Control** tab.

The MCDU-02 servo can control two servomotors. The information on the position of the servomotor is provided by an encoder that must be associated with its movement.

Motor Control group box

This group box sets the MCDU control parameters, used when performing motion in setup mode. See Figure 3-93 on page 163.

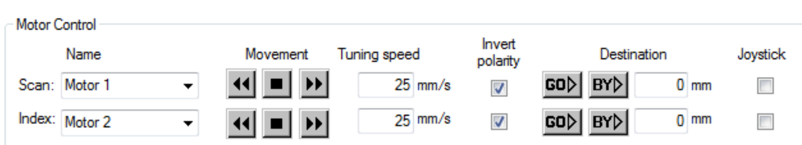
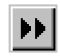
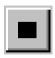


Figure 3-93 The Motor Control group box

Name

These list boxes are used to select the motor to be assigned to each axis. The name of the motor can be edited.

Movement

These buttons are used to manually control the scanner movement on each axis and are used to move the scanner in different directions. The  button moves the scanner backward, the  button moves the scanner forward, the  button stops the scanner along the corresponding axis.

Tuning speed

These boxes are used to set the maximum speed the scanner can reach, when you are using the **Movement** buttons in setup mode.

Invert polarity

Selecting these check boxes inverts the motor polarity of the corresponding axis.

This button moves the scanner to the position specified in the **Destination** box of the corresponding axis.

This button moves the scanner by a distance corresponding to the distance specified in the **Destination** box of the corresponding axis.

Destination

These boxes are used to enter the position where the scanner moves on the corresponding axis when you click **Go**.

Joystick

When selected, you can use these check boxes to enable an external joystick to control the scanner movement on each axis.

Motor state group box

The indicators of this group box give the motor status of each axis. See Figure 3-94 on page 165. The different indicators have the following meanings:

On:

When this indicator is green, the motor of the corresponding axis is turned on.

In motion:

When this indicator is green, the motor is in motion on the corresponding axis. When the indicator is red, a motion error occurred with the motor of the corresponding axis.

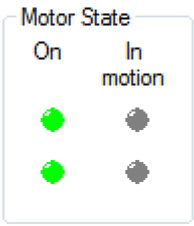


Figure 3-94 The Motor State group box

NOTE

Servomotors must be used in a closed loop by using encoder feedback. The first encoder is automatically assigned to the first motor, and the second encoder to the second motor.

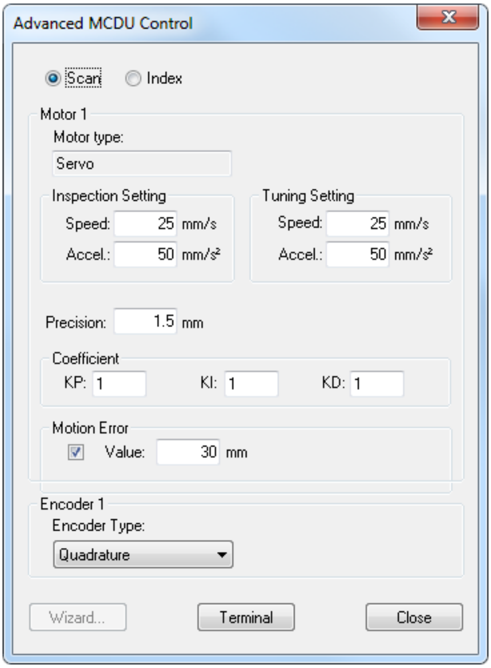


Figure 3-95 The Advanced MCDU Control dialog box for servomotors

The **Advanced MCDU Control** dialog box for servomotors contains the following options:

Scan, Index

These option buttons are used to select the axis for which you want to set the parameters: scan axis or index axis.

Motor type

This box indicates the motor type used for the selected axis:

Servo: The MCDU-02 servo can support two servomotors.

Inspection setting: Speed

This box is used to set the maximum speed the motor can reach when executing the inspection sequence.

Inspection setting: Accel.

This box is used to set the motor acceleration when executing the inspection sequence. The motor accelerates at the rate indicated until it reaches the maximum speed specified. The motor decelerates at the same rate when stopped.

Tuning setting: Speed

This box is used to set the maximum speed the motor can reach during system tuning.

Tuning setting: Accel.

This box is used to set the motor acceleration during system tuning. The motor accelerates at the rate indicated until it reaches the maximum speed specified. The motor decelerates at the same rate when stopped.

Precision

This box indicates the precision, in distance units, required when position feedback is provided by an encoder.

Coefficient: KP

This text box is used to set the proportional gain of the system compensation PID filter. Proportional control is a pure gain adjustment acting on the error signal to provide the driving input to the process. The KP term in the PID-controller is used to adjust the speed of the system.

Coefficient: KI

This box is used to set the integrator of the system compensation PID filter. Integral control is used to provide the required accuracy for the control system.

Coefficient: KD

This box is used to set the derivative term of the system compensation PID filter. Derivative action is normally introduced to increase the damping in the system. In effect, it applies the “brakes” in an attempt to prevent overshoot (or undershoot).

NOTE

The above parameters must be set with particular attention. You must modify these parameters when manually tuning PID parameters for servomotors.

Motion error

The check box, when selected, enables a protection function that stops the servomotors when a too large difference occurs between the target position and the actual position. Using the **Value** text box, you can enter a specific tolerance value for the motion error protection, expressed according to the currently specified unit.

Encoder type

This box is used to select the encoder type from the following: **Quadrature, Reversed Quadrature, Clock Dir**., and **Reversed Clock Dir.**

Wizard

This option has not yet been implemented in TomoView software.

Terminal

This button opens the **MCDU-02 Terminal** window, which experts can use to enter low-level GALIL commands in order to program the system. It should be used only by an MCDU-02 expert or by a user following precise instructions given by the Evident NDT technical support personnel.

Close

This button confirms your changes and closes the dialog box.

3.12.2 MCDU I/O Tab

The MCDU I/O tab contains the options for setting the MCDU-02 inputs and outputs. See Figure 3-96 on page 168.

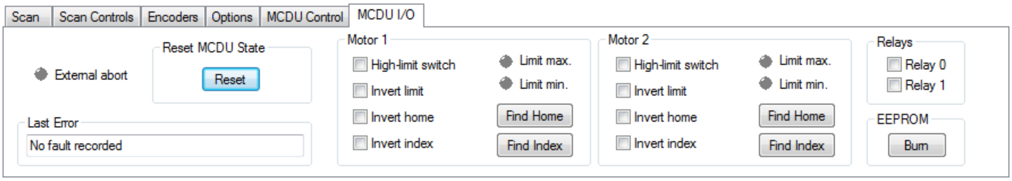


Figure 3-96 The MCDU I/O tab

NOTE

You must set the input/output parameters at least once for each different scanner. As you cannot modify the input/output parameters of the MCDU-02 unit, you need a PC and the TomoView software to modify and save the I/O settings in the MCDU-02 permanent memory. If you do not set the I/O parameters for your scanner, then the MCDU-02 unit uses the last settings that were saved or the scanner factory settings.

The **MCDU I/O** tab is subdivided into the **Reset MCDU State, Last Error, Motor 1, Motor 2, Relays**, and **EEPROM** group boxes.

External abort

This indicator, when turned on in red, indicates that the STOP emergency button is pressed on the MCDU-02.

Reset MCDU state group box

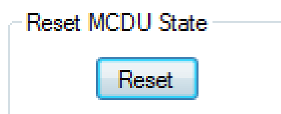


Figure 3-97 The MCDU State group box

The **MCDU State group box** has the following check boxes. See Figure 3-97 on page 169.

Reset

This button resets the state of the MCDU-02: motion error, voltage applied to the motors, relays, etc. This command, however, does not reset the PID parameters or the encoder type. See Figure 3-97 on page 169.

Last error group box

Figure 3-98 The Last Error group box

The **Last Error group box** has the following check boxes. See Figure 3-98 on page 169.

This box displays the last system error that occurred. See Figure 3-98 on page 169.

Motor 1, and **Motor 2 group boxes**

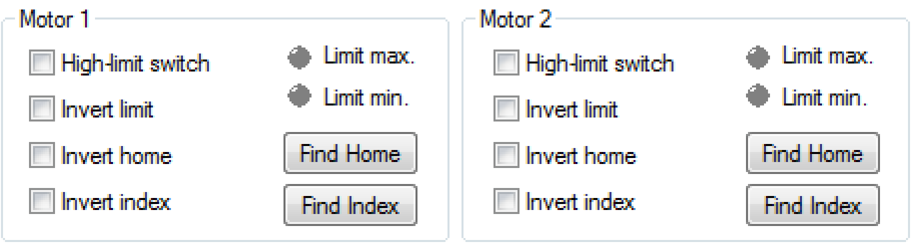


Figure 3-99 The Motor 1 and Motor 2 group box

The check boxes of these two group boxes (see Figure 3-99 on page 170) can be selected or cleared to enable or disable the four options described as follows according to the motor chosen.

High limit switch

When selected, this check box inverts the polarity of the corresponding motor limit switch (if applicable).

Invert limit

When selected, this check box inverts the position of the corresponding motor limit switch (if applicable).

Invert home

When selected, this check box inverts the corresponding motor home signal (if applicable).

Invert index

When selected, this check box inverts the corresponding motor index signal (if applicable).

Find Home

In the case of custom applications, this button moves the scanner in such a way as to find the home signal.

Find Index

In the case of custom applications, this button moves the scanner in such a way as to find the index signal.

Relays group box

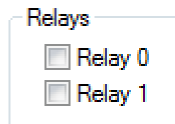


Figure 3-100 The Relays group box

The **Relays group box** has the following check boxes. See Figure 3-100 on page 171.

Relay 0 and **Relay 1**

In the case of custom applications, these check boxes are used to control relays in the MCDU-02 unit. See Figure 3-100 on page 171.

EEPROM group box

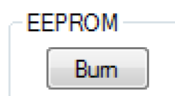


Figure 3-101 The EEPROM group box

The **EEPROM group box** has the following check boxes. See Figure 3-101 on page 171.

Burn

This button is used to save the current specific I/O settings (encoder type, motor polarity, etc.) in the MCDU-02 EEPROM permanent memory. The saved configuration is then used by default at MCDU-02 startup.

IMPORTANT

If you do not click the **Burn** button, some settings could be lost when you quit the TomoView software.

3.13 View and Pane Properties Dialog Bar

Clicking the **View and Pane Properties** button  on the **Main** and **TomoView Manager** toolbars toggles the visibility of the **View and Pane Properties** dialog bar. The **Information**, **Display**, **Echo Dynamics**, **Overlay**, **Palette**, **Data Source**, **Parameters**, **Units**, **View Linking**, **Rebounds**, and **Strip** tabs are available in the Pane Properties dialog box, depending on the data type in the active view. When the tab parameters are not applicable to a selected pane, the tab does not appear in the View and Pane Properties dialog bar. See Figure 3-102 on page 172.

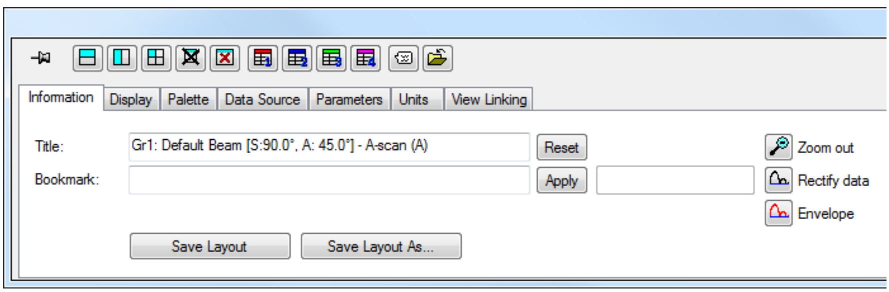


Figure 3-102 The View and Pane Properties dialog box

NOTE

The **View and Pane Properties** dialog box is not available in the TomoView **Lite Aero and TomoView Weld** versions.

3.13.1 Information Tab

The **Information** tab (see Figure 3-103 on page 173) contains commands that can be applied to the active view or layout.

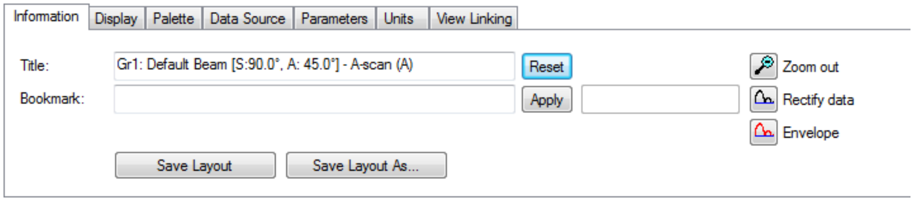


Figure 3-103 The Information tab

The **Information** tab (see Figure 3-103 on page 173) contains the following group boxes:

Title

This box identifies the title of the active view and can be used to change titles.

Bookmark

This box can be used to define the text to be displayed inside a pane when a view isn't available on the current data file.

Reset

Clicking on the Reset button resets the **Title** to it original value, and also removes any defined **Bookmark**.

Apply

When the bookmark contains code for advanced layout association, clicking this button applies the bookmark code. This button is disabled by default for standard users.

**Zoom Out**

Zooms out the active view in order to show all data.

**Rectify Data**

Applies a software rectification to the signal data.

**Envelope**

Alternately activates and deactivates the envelope mode for the active A-scan view.

Save Layout

Saves the modifications made to the current layout.

Save Layout As

Opens the **Save Current Layout As** dialog box, used to save the current layout under the number and name of your choice.

3.13.2 Display Tab

The **Display** tab contains various options to modify the active view display and will change depending on the active view type. See Figure 3-104 on page 174.

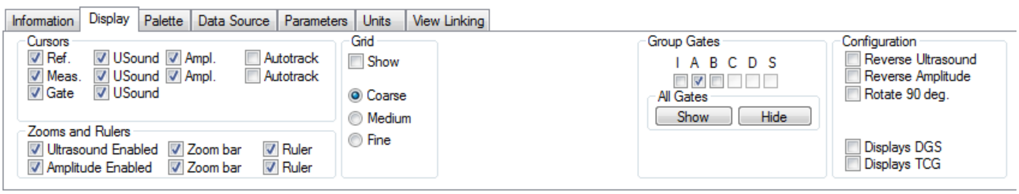


Figure 3-104 The Display tab for an A-scan view

The **Display** tab is subdivided into **Cursors**, **Zooms and Rulers**, **Grid**, **Group Gates**, and **Configuration** group boxes.

Cursors group box

This group box encloses check boxes used to select display options related to the cursors. See Figure 3-105 on page 174.

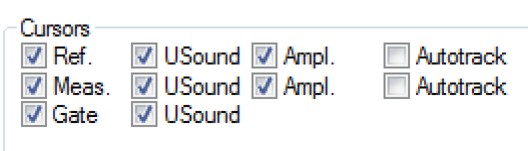


Figure 3-105 The Cursors group box

Ref., Meas, and Gate

Selecting one of these check boxes adds the **Reference, Measurement**, or **Gate** cursors to the view. Depending on the active view, these cursors can include:

Scan

Cursor for the scan axis.

Index

Cursor for the index axis.

USound

Cursor for the ultrasonic axis.

Ampl.

Cursor for the amplitude axis.

Autotrack

Autotracking measurement cursor that displays the amplitude of the ultrasonic cursor position.

TOFD

Hyperbolic cursor for time-of-flight diffraction.

Polar coordinates

Polar cursors (available only when Polar view is active).

Meas.

Adds the measurement cursors to the view (see previous **Ref.** check box).

Gate

Adds the gate cursors to the view (see previous **Ref.** check box). The gate cursors are black cursors showing the position of the gate selector.

Zooms and Rulers group box

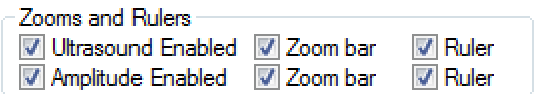


Figure 3-106 Zooms and Rulers group box

The **Zooms and Rulers group box** has the following check boxes. See Figure 3-106 on page 175.

USound Enabled/Amplitude Enabled

This check box enables the zoom bar on the **Ultrasound/Amplitude** axis.
Zoom bar

Adds a zoom bar to the corresponding axis in the active view.

Ruler

Adds a ruler to the index corresponding axis in the active view.

Grid group box

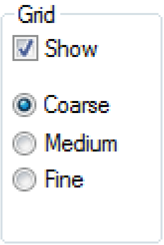


Figure 3-107 The Grid group box

The **Grid group box** has the following check box. See Figure 3-107 on page 176.

The **Show** button enables the Grid on the active A-scan view, and the **Coarse**, **Medium**, and **Fine** options are used to customize the grid density.

Polar View Image Processing group box

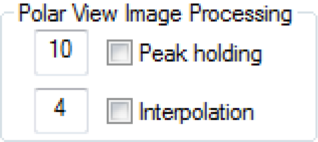


Figure 3-108 Polar View Image Processing group box

Depending on the inspection sequence resolutions on the scan or index axis, there might be positions where no data is shown due to the limited size of the color-coded A-scan line. You can use the **Peak holding** or **Interpolation** image processing algorithms to fill these gaps. See Figure 3-108 on page 176.

Peak holding

This algorithm looks for a maximum in amplitude and spreads this value over the selected number of samples (must be less than or equal to 50). This algorithm is optimized for polar views constructed from C-scan data.

Interpolation

This algorithm spreads a sample over the selected number of samples (must be less than or equal to 50) to its left and right, until recorded data of a higher amplitude sample is detected. This algorithm is optimized for polar views constructed from A-scan data.

Auto-Scroll Zoom group box

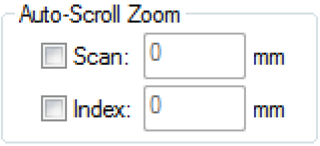


Figure 3-109 The Auto-Scroll Zoom group box

The **Auto-Scroll Zoom group box** has the following check boxes. See Figure 3-109 on page 177.

Scan/Index

This activates the Auto-Scroll Zoom function for the selected axis, which dynamically zooms in on the active view within the specified range.

Smooth

Adds a display smoothing function to the active view.

Configuration group box

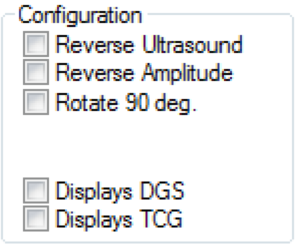


Figure 3-110 The Configuration group box

The **Configuration group box** has the following check boxes. See Figure 3-110 on page 178.

Reverse USound

Reverses the data along the ultrasonic axis (horizontal or vertical flip).

Reverse amplitude

In an A-scan view selecting this check box reverses the data along the amplitude axis (vertical flip).

Rotate 90 deg

Rotates the data so that the two axes interchange.

Display DGS

Enables the distance-gain-size (DGS)

Display TCG

Enables the time-corrected gain (TCG) display on the active A-scan view.

Reverse Horizontal/Vertical

Reverses the horizontal/vertical axis on an active Polar view.

Show part

Enables the display of the defined part on an active Polar view.

Gate selector

Adds the gate selector to the active view.

Keep 1:1 ratio

Displays the view in a way that keeps the sample ratio of both displayed axes.

Reverse scan

Reverses the data along the scan axis (horizontal or vertical flip).

Reverse index

Reverses the data along the index axis (horizontal or vertical flip).

Reverse amplitude

In an A-scan view selecting this check box reverses the data along the amplitude axis (vertical flip).

Group Gates group box

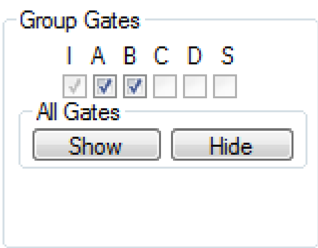


Figure 3-111 Group Gates group box

The **Group Gates group box** has the following. See Figure 3-111 on page 179.

I, A, B, C, D, and S

These check boxes enable the display of the corresponding gates on the active A-scan.

All Gates

Show All / Hide All

Enables or disables the display of gates on the active A-scan view.

The **Display** tab for a Polar view contains additional options that are specific to this view (see Figure 3-112 on page 180).

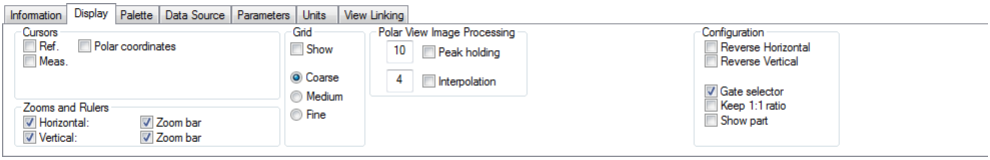


Figure 3-112 The Display tab for a Polar View

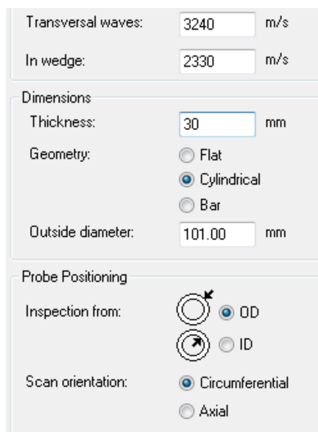


Figure 3-113 Part Definition dialog box

The **Part Definition** dialog box is subdivided into the **Material Velocity, Dimensions**, and **Probe Positioning** group boxes.

Material Velocity group box

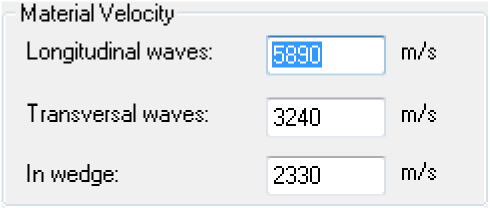


Figure 3-114 The Material Velocity group box

The **Material Velocity group box** contains the following. See Figure 3-114 on page 181.

Longitudinal/transversal waves

Sets the velocity of the longitudinal and transversal waves inside the part material.

In Wedge

Sets the velocity inside the wedge.

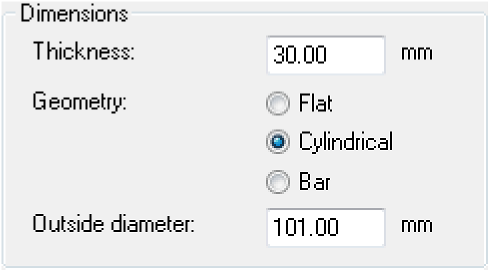


Figure 3-115 The Dimensions group box

The **Dimensions group box** contains the following. See Figure 3-115 on page 181.

Thickness

Defines the thickness for flat and cylindrical part types.

Geometry

Option buttons used to select the geometry of the inspected part from **Flat**, **Cylindrical,** and **Bar**.

Probe Positioning group box

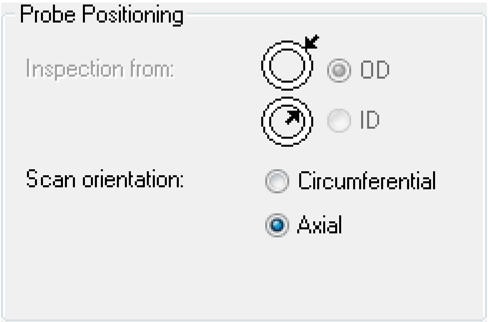


Figure 3-116 The Probe Positioning group box

The **Probe Positioning group box** contains the following. See Figure 3-116 on page 182.

Inspection from

Defines if the probe is positioned on the outside diameter (**OD**) or inside diameter (**ID)** for the inspection of cylindrical parts.

Scan orientation

Defines if the orientation of the scan axis is **Circumferential** or **Axial** during the inspection.

NOTE

If the part type is defined as cylindrical or bar, the polar view type is added to the **Contents** dialog box when performing the analysis. See Figure 3-117 on page 183.

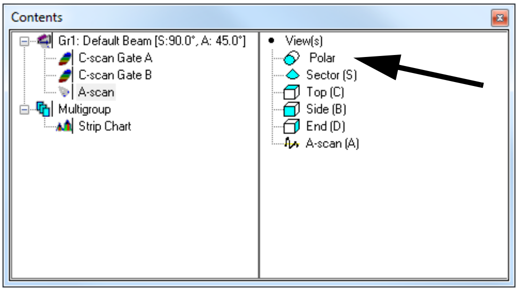


Figure 3-117 Contents dialog box with a Polar view

3.13.3 Echo Dynamics Tab

The echo dynamics is a 2-D curve resulting from the projection of the data under a cursor or between cursors. The **Echo Dynamics** tab (see Figure 3-118 on page 183) is used to set the echo-dynamic curve parameters.

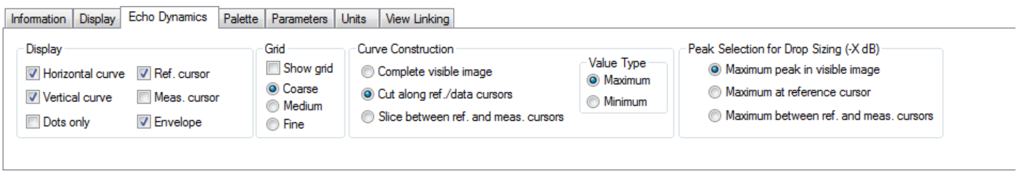


Figure 3-118 The Echo Dynamics tab

The **Echo Dynamics** tab is subdivided into the **Display, Grid, Curve Construction,** and **Peak Selection for Drop Sizing** (-X dB) group boxes.

Display group box

This group box encloses the display options of the echo-dynamic curves. See Figure 3-119 on page 184.

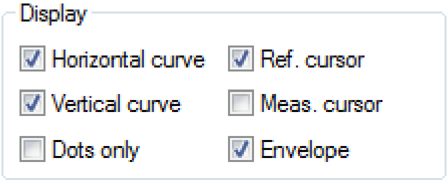


Figure 3-119 The Display group box

Horizontal/vertical curve

When this check box is selected, a horizontal echo-dynamic curve is displayed at the top or right side of the active view.

Dots only

When this check box is selected, echo-dynamic curves are displayed with dots.

Ref./Meas. cursor

Enables the display of the reference/measurement cursors on the echo-dynamic curves.

Envelope

When this check box is selected, the echo-dynamic curves are displayed as an envelope of all the curves between the cursors or the complete visible image.

Grid group box

Figure 3-120 The Grid group box

The **Grid group box** contains the following. See Figure 3-120 on page 184.

The **Show grid** enables the Grid on the echo dynamic curves of the active view. The **Coarse, Medium**, and **Fine** options can be used to customize the grid density.

Curve Construction group box

The **Curve Construction group box** contains the following. See Figure 3-121 on page 185.

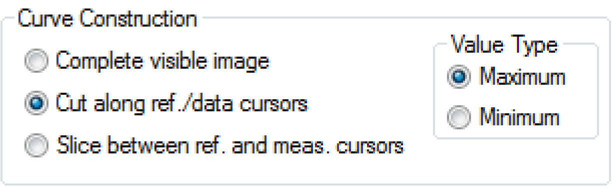


Figure 3-121 The Curve Construction group box

This group box encloses option buttons that are used to select one of the three curve construction types.

Complete visible image

With this option, the echo-dynamic curve is constructed with the complete visible image of the inspected part.

Cut along reference cursors

With this option, the echo-dynamic curve is constructed with the cross section that is specified by the reference cursor.

Slice between Ref. and Meas. cursors

With this option, the echo-dynamic curve is constructed with the slice that is specified by the reference and measurement cursors.

Value type

Select to build the **Echo Dynamics** curve using the **Maximum** or the **Minimum** value of the current area.

Peak Selection for Drop Sizing group box (–X dB)

The **Peak Selection for Drop Sizing (-X dB) group box** contains the following. See Figure 3-122 on page 186

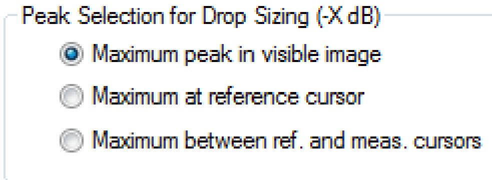


Figure 3-122 The Peak Selection for Drop Sizing (-X dB) group box

This group box allows you to select the option that will be used for the –X dB drop sizing measurements on echo-dynamic curves from **Maximum peak in visible image, Maximum at reference cursor,** and **Maximum between ref. and meas. cursors**.

3.13.4 Overlay Tab

The **Overlay** tab (see Figure 3-123 on page 186) is used to set the display of overlay drawings on the active volume-corrected view.

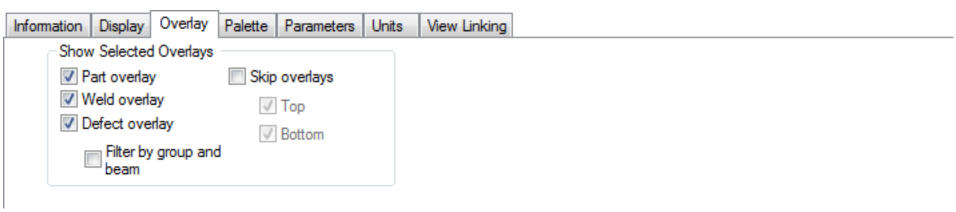


Figure 3-123 The Overlay tab

Show Selected Overlays group box

The S**how Selected Overlays group box** contains the following. See Figure 3-124 on page 187.

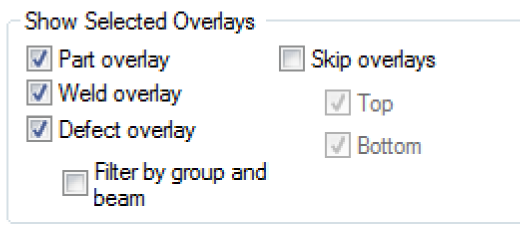


Figure 3-124 The Show Selected Overlays group box

Enables the display of the selected overlay on the active volume corrected view.

Part/Weld/Defect Overlays

Enables the display of the selected overlay on the active volume corrected view.

Filter by group and beam

Selecting this check box will only display the defects created on the current group for the active view.

Skip Overlays

Enables the display of the skips on the current view for the corrected volume and Sector (S) views. Deselecting the **Top** or **Bottom** check boxes will remove the corresponding skips from the active view.

3.13.5 Palette Tab

The **Palette** tab (see Figure 3-125 on page 187) is used to set the color palette of the active view.

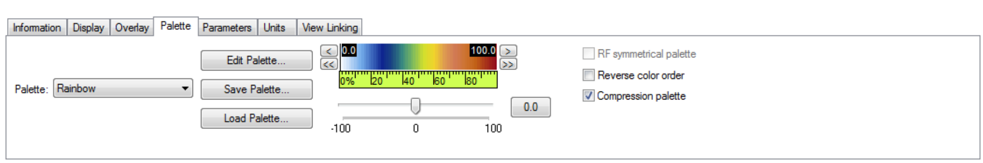


Figure 3-125 The Palette tab

The **Palette** tab is subdivided into **Palette**, **Edit Palette**, **Save Palette**, **Load Palette**, the color palette, the cursors, left and right arrow buttons, the software gain slider, **RF symmetrical**, **Reverse color order**, and **Compression palette**.

Palette

This box is used to select a predefined color palette from a list. A zone displays the selected color palette.

Edit Palette

Opens the **Palette Editor** dialog box, used to define color palettes.

Save Palette

Opens a standard **Save As** dialog box, used to name and save a color palette (.col file). The saved color palette replaces the selected palette in the **Palette** box.

Load Palette

Opens a standard **Open** dialog box, used to select and load a color palette (.col file). The loaded color palette replaces the currently selected palette in the **Palette** box.

Color palette

Displays the current color palette for the active view. See Figure 3-126 on page 188.

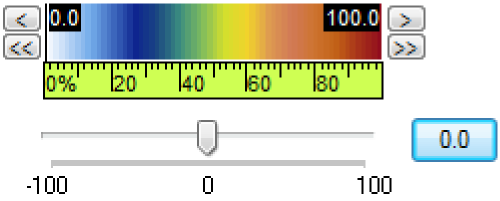


Figure 3-126 The Color Palette dialog box

Double-clicking between the cursors divides the current palette range by two and double-clicking outside the cursors restores the full palette range default.

Double-clicking outside the cursors restores the default full palette range, which goes from –100 % to +100 %.

Left and right arrow buttons

Clicking the left and right arrow buttons, respectively, shifts the color palette range by a fine increment to the left and to the right.

Left and right double-arrow buttons

Clicking the left and right double-arrow buttons, respectively, shifts the color palette range on the left and on the right by a coarse increment.

Color palette left/right cursors

Dragging the left/right cursor sets the lower limit of the color palette range as a percentage (linear data), or in decibels (logarithmic data).

Dragging the color palette itself sets the lower and upper limits of the color palette range simultaneously.

Software gain slider

Dragging the slider, or using the arrow keys, sets the software gain, which ranges from –100 dB to +100 dB. Clicking on the gain slider moves the slider by 6 dB or 6 % increments, depending on the case, simultaneously.

The software gain you set appears in a box to the right of the software gain slider. Clicking this box restores the default software gain, which is 0 dB or 0 %.

NOTE

The software gain slider is used to set a software-calculated amplitude gain, which leaves the acquisition data unchanged.

RF symmetrical palette

Selecting this check box makes the selected color palette symmetrical relative to the zero value.

Reverse color order

Selecting this check box reverses the color order in the color palette.

Compression palette

When enabled, the full color palette range between the left and right color palette.

3.13.6 Data Source Tab

Using the **Data Source** tab (see example in Figure 3-127 on page 190), you can set the data source used to compute the projection displayed in a view. The contents of the Data Source changes depending on the current type of view selected.

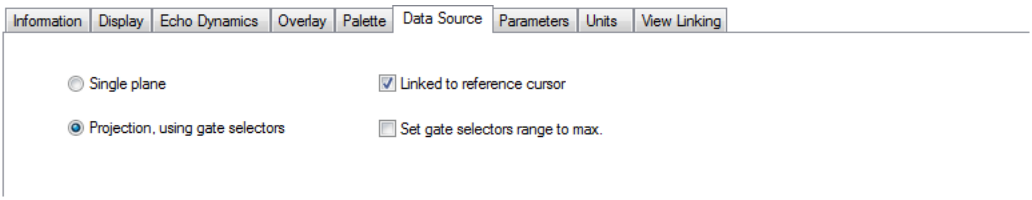


Figure 3-127 The Data Source tab

The **Data Source** tab may contain the following group box options:

Single plane

In this mode, the view displays a single plane crossing the inspection volume. A single gate selector selects the cross section to display in the inspected part.

Projection, using gate selectors

In this mode, TomoView uses all data contained between two gate selectors to generate the displayed image. When many samples of data must use the same display pixel, the highest value is used.

Link to reference cursor

This check box, when selected, links the gate selector position to the corresponding reference and measurement cursors. Dragging these cursors then causes gate selectors of other views to react accordingly.

Set gate selectors range to max.

When this check box is selected, the gate selectors will be set at lower and higher values to display the volume of data. When this check box is selected, it is not possible to adjust the gate selector values at the bottom of the pane.

Scan source

Using this list box, you can select the cursor (**Reference cursor**, **Measure cursor** or **Data cursor**) that is to be used to select the A-scan along the scan axis.

Index source

Using this list box, you can select the cursor (**Reference cursor**, **Measure cursor** or **Data cursor**) that is to be used to select the A-scan along the index axis.

3.13.7 Parameters Tab

The **Parameters** tab (see Figure 3-128 on page 191) is used to set the parameters that link the UT data volume to the actual physical volume of the specimen. These parameters, available only in inspection mode, are used to make adjustments to the ultrasonic settings and are saved in an .A01 file.

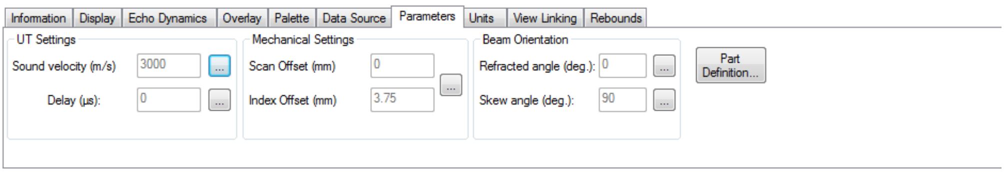


Figure 3-128 The Parameters tab

The **Parameters** tab is subdivided into **UT Settings, Mechanical Settings**, and **Beam Orientation**.

UT Settings group box

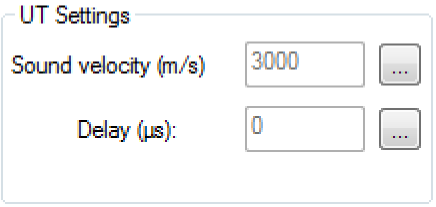


Figure 3-129 UT Settings group box

Sound velocity (m/s)

This box displays the sound velocity in the inspected material. The sound velocity is dependent of the wave type and material type used. Clicking the

button opens the **Sound Velocity** dialog box (see Figure 3-129 on page 191) in which you can set the sound velocity parameters.

Sound Velocity dialog box

Using the **Sound Velocity** dialog box (see Figure 3-130 on page 192), you can set the sound velocity parameters.

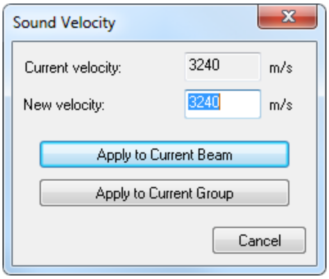


Figure 3-130 The Sound Velocity dialog box

The **Sound Velocity** dialog box contains the following:

Current velocity

Displays the velocity of the current wave mode, either in meters per second (m/s) or in inches per microsecond (**in./µs**).

New velocity

Using this box, you can set the sound velocity for the active mode, either in meters per second (**m/s**) or in inches per microsecond (**in./µs**). The value is limited to the range between 500 m/s and 20,000 m/s or the corresponding values in **in./µs**.

Apply to Current Beam/Apply to Current Group

Applies the velocities defined in the New velocity box to either the current beam or current Group.

Delay (µs)

This box displays the sum of all time delays preceding the first A-scan sample, in microseconds. The **Delay** dialog box (see Figure 3-131 on page 193) is used to set the probe, focal law, and channel delay parameters.

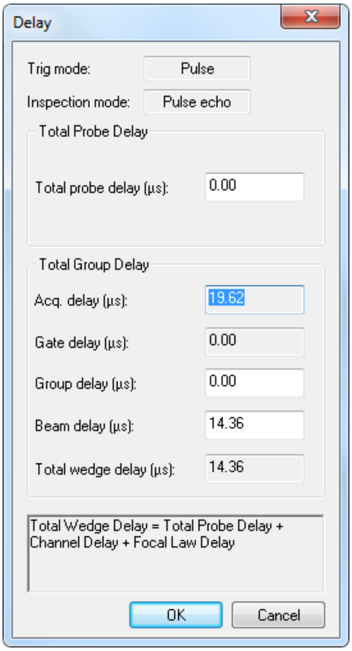


Figure 3-131 The Delay dialog box

The **Delay** dialog box contains the following:

Total probe delay (µs)

Defines the total probe delay of the current Group.

Trig mode/Inspection mode

Indicates the current Trigger and Inspection modes.

Acq./Gate/Total wedge delay (µs)

Displays the acquisition/gate/total wedge delays in microseconds. the total wedge delay is calculated using the following formula:

Total wedge delay = Total probe delay + channel delay + focal law delay + gate delay

Mechanical Settings group box

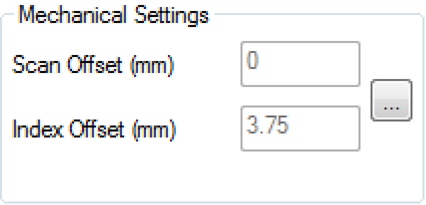


Figure 3-132 The Mechanical Settings group box

Scan axis offset (mm) and **Index axis offset (mm)**

These two boxes display the sum of the Scanner, Probe, and Beam offsets of the emergence point of a beam (based on the values entered in the **UT Settings** dialog bar), relative to the scan and index axes: scanner offsets, probe offsets, beam offsets. The **Scan axis offset** and **Index axis offset** parameters set the absolute position of the first data point relative to the scan and index axes (top view), that is, the exact place where the first data point was acquired from the part. Clicking on the [...] button will open the **Mechanical Settings** dialog box (see Figure 3-132 on page 194) which is used to set the offset parameters.

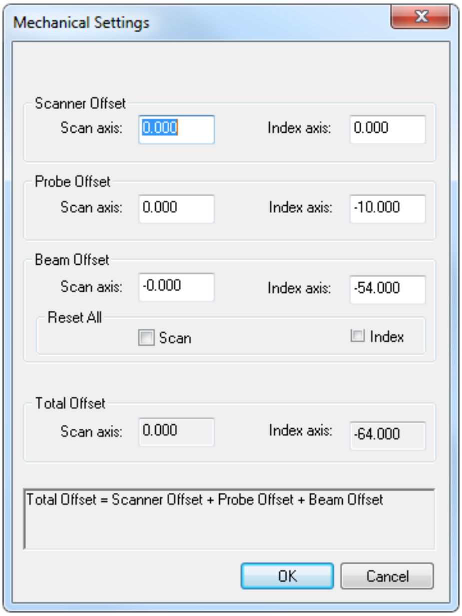


Figure 3-133 The Mechanical Settings dialog box

The **Mechanical Settings** dialog box contains the following:

Scanner offset

This group box encloses the boxes used to set the scanner offset for the scan and index axes. These offsets will be applied to all groups.

Probe offset

This group box encloses the boxes used to set the probe offset for the scan and index axes. These offsets will be applied only yo the active group.

Beam offset

This group box encloses the boxes used to set the beam offset for the scan and index axes. These offsets will be applied only to the active beam. The **Reset All** buttons resets the corresponding value, and makes it unavailable for further modifications.

Total offset

This group box displays the total offset for the scan and index axes, which is calculated using the values entered in the dialog box, according to the following formula:

Total offset = Scanner offset + Probe offset + Beam offset

Beam Orientation group box

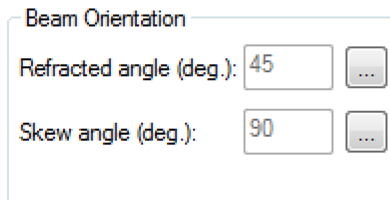


Figure 3-134 The Beam Orientation group box

Refracted angle (deg)

This box displays the refracted angle in degrees, relative to the normal of the specimen surface, from –90° to +90°. Clicking opens the Refracted angle (deg.) dialog box. (See Figure 3-134 on page 196.)

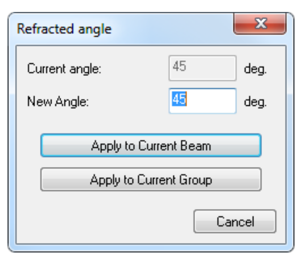


Figure 3-135 The Refracted Angle dialog box

Current angle

Displays the current refracted angle in degrees.

New angle

Resets the refracted angle in degrees.

Apply the Current Beam/Apply the Current Group

Applies the refracted angle defined in the **New Angle** box to either the current beam or the current Group.

Skew angle (deg)

This box displays the skew angle in degrees, relative to the scan axis, from 0° to 360°. Clicking  displays the **Skew angle** dialog box (see Figure 3-136 on page 197)

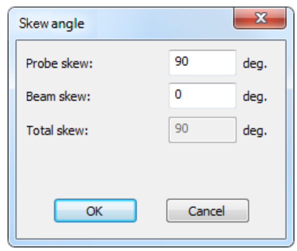


Figure 3-136 The Skew angle dialog box

Probe skew/Bean skew

Sets the probe and beam skews in degrees.

Total skew

Displays the total skew angle calculated according to the following formula:

$$\text{Total skew} = \text{Probe skew} + \text{Beam skew}$$

Part Definition

This button opens the **Part Definition** dialog box.

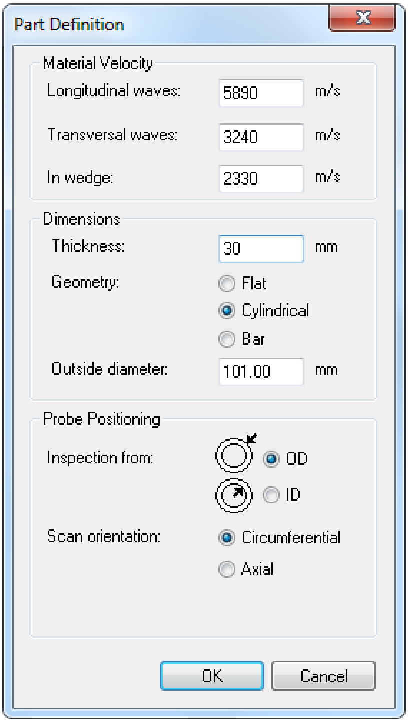


Figure 3-137 The Part Definition dialog box

The **Part Definition** dialog box (see Figure 3-137 on page 198) is subdivided into the **Material Velocity, Dimensions,** and Probe **Positioning** group boxes.

Material velocity group box

The **Material Velocity group box** contains the following. See Figure 3-138 on page 199.

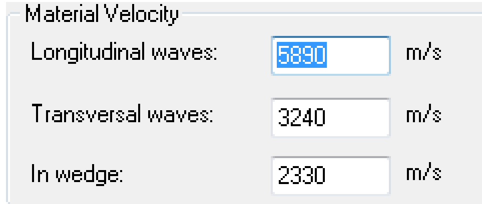


Figure 3-138 The Material Velocity group box

Longitudinal/Transversal waves

Sets the velocity of the longitudinal and transversal waves inside the part material.

In Wedge

Sets the velocity inside the wedge.

Dimensions group box

The **Dimensions group box** contains the following. See Figure 3-139 on page 199.

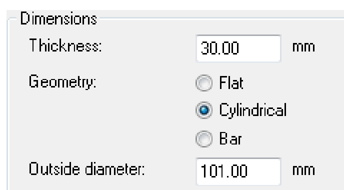


Figure 3-139 The Dimensions group box

Thickness

Defines the thickness for flat and cylindrical part types.

Geometry

Option buttons used to select the geometry of the inspected part. Choose from **Flat, Cylindrical**, or **Bar**.

Probe Positioning group box

The **Probe Positioning group box** contains the following. See Figure 3-140 on page 200.

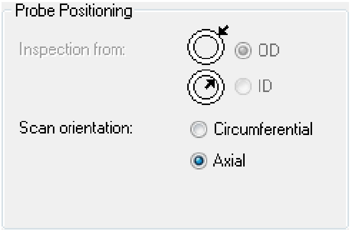


Figure 3-140 The Probe Positioning group box

Inspection from

Defines if the probe is positioned on the outside diameter **(OD)** or inside diameter **(ID)** for the inspection of cylindrical parts.

Scan orientation

Defines if the orientation of the scan axis is **Circumferential** or **Axial** during the inspection.

NOTE

If the part type is defined as cylindrical or bar, the polar view type is added to the **Contents** dialog box when performing the analysis. See Figure 3-141 on page 200.

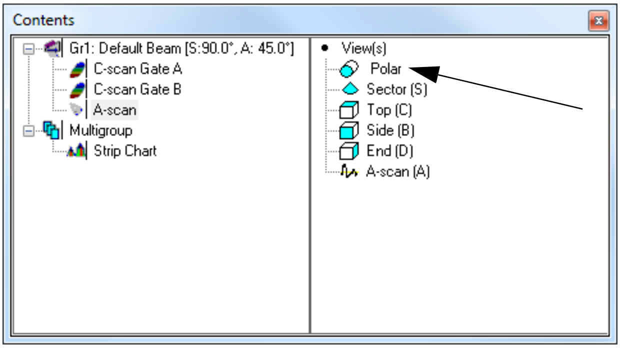


Figure 3-141 Contents dialog box with a Polar view

3.13.8 Units Tab

The **Units** tab (see Figure 3-142 on page 201) is used to set the units for the four data dimensions, for all views displaying the same data group as the active view.

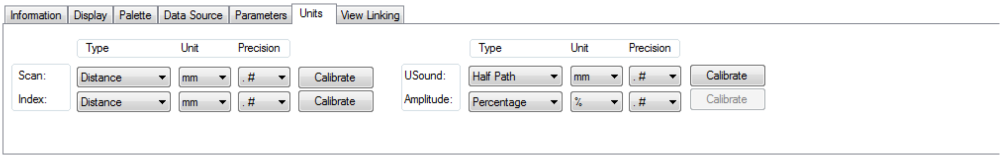


Figure 3-142 The Units tab

The **Units** tab contains the following:

Scan/Index/USound/Type

Set the scan/index/ultrasonic axis/amplitude units.

Type

This column contains the boxes used to select the unit type for each parameter. According to the dimension, the possible unit types are the following:

Scan: **Distance**, **Samples**, **Rotation**.

Index: **Distance**, **Samples**, **Rotation**.

USound: **True Depth**, **Half Path**, **Full Path**, **Time**, **Time/2**, **Samples**, **TOFD**.

Amplitude: **Samples**, **Percentage**, **Decibels**.

Unit

This column contains the boxes used to select the measurement unit for each parameter. According to the unit type, the possible units are the following:

Distance: **True Depth**, **Half Path**, **Full Path**: **m**, **mm**, **in.**, **mils**.

Samples: **smpl**.

Rotation: **deg**, **rad**.

Time, **Time/2**: **s**, **µs**.

TOFD: **m**, **mm**, **in.**, **mils**.

Percentage: **%**.

Decibels: **dB**.

Precision

This column contains the boxes used to select the measurement precision, that is:

.= 0 digits after the decimal

.#= 1 digit after the decimal

.##= 2 digits after the decimal

.###= 3 digits after the decimal

Calibrate

Clicking on the Calibrate buttons opens the **Mechanical Axis, Time/Full Path** and **Time/Half Path, TOFD, True Depth** or **Translation/Rotation** calibration dialog boxes. See the following for a description of these dialog boxes.

Mechanical Axis Calibration dialog box

The **Mechanical Axis Calibration** dialog box (see Figure 3-143 on page 202) is used to calibrate the distance value on the scan or index axes.

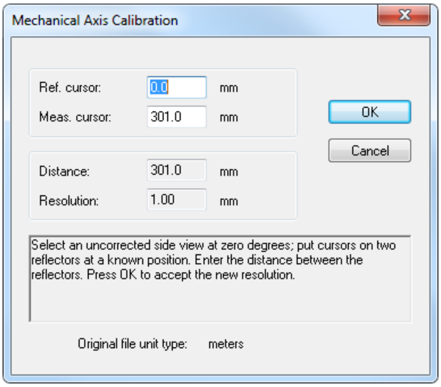


Figure 3-143 The Mechanical Axis Calibration dialog box

The **Mechanical Axis Calibration** dialog box contains the following:

Reference cursor/Measurement cursor

These boxes are used to set the reference and measurement cursor positions.

Distance/Resolution

These boxes indicate the distance between reference and measurement cursor positions along with the measurement resolution.

Original file unit type

This line displays the original unit used in the data file.

Time / Full Path and Time / Half Path dialog boxes

The **Time / Full Path** and **Time / Half Path** dialog boxes (see Figure 3-144 on page 203) are used to calibrate the time value relative to the full path value and the half path value.

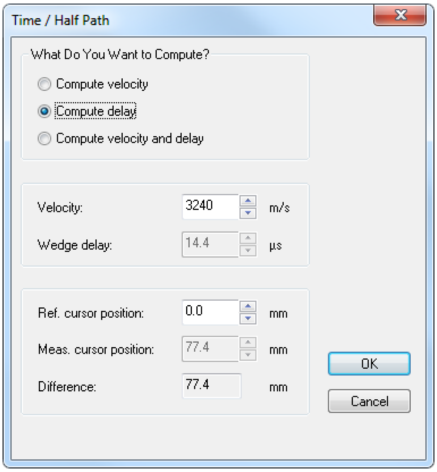


Figure 3-144 The Time / Half Path dialog box

The **Time / Full Path** and the **Time / Half Path** dialog boxes contain the following:

What do you want to compute?

This group box allows you to select which parameter should be calculated.

Velocity/Wedge Delay

This box indicates the sound velocity and wedge delay inside the material.

Ref. cursor position/Meas. cursor position

These boxes indicate the current reference and measurement cursor positions.

Difference

This box indicates the difference between the reference and measurement cursor positions.

TOFD dialog box

The **TOFD** dialog box (see Figure 3-145 on page 204) is used to calibrate the ultrasonic axis when you use the TOFD method.

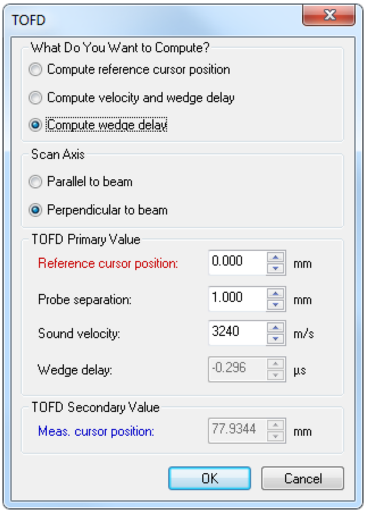


Figure 3-145 The TOFD dialog box

The **TOFD** dialog box contains the following:

What do you want to compute?

This group box encloses option buttons used to select the parameter that is to be calculated from other parameters:

Compute reference cursor position

This option is used to adapt the TOFD ruler, setting it to its zero position at the reference cursor position. Position the reference cursor on the first echo signal and then click **Ok**.

Compute velocity and wedge delay

This option is used to calculate both the material sound velocity and the probe wedge delay by using two reference signals, typically the lateral wave and the backwall echo.

Compute wedge delay

This option is used to calculate the probe wedge delay by using a fixed value of the sound velocity and one reference signal, typically the lateral wave or the backwall echo.

Scan axis

This group box encloses option buttons that can be used to select if the scan axis orientation relative to the ultrasonic beam projection should be defined as **Parallel to beam** or **Perpendicular to beam**.

Reference cursor position

This box defines the real (true depth) reference cursor position in millimeters (mm) or inches (in.).

Probe separation

This box defines the distance between the probes in millimeters (mm) or inches (in.).

Sound velocity

This box defines the material sound velocity in meters per second (m/s) or inches per microsecond (in./µs).

Wedge delay

This box defines the probe wedge delay in microseconds (µs).

NOTE

For details on using the calibration function with the **TOFD** dialog box, see section "Performing TOFD Calibration in Analysis Mode" on page 355.

True Depth dialog box

The **True Depth** dialog box (see Figure 3-146 on page 206) is used to calibrate the ultrasonic axis in true depth units.

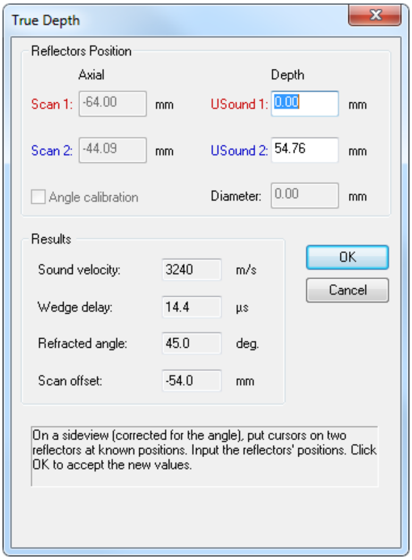


Figure 3-146 The True Depth dialog box

The **True Depth** dialog box contains text boxes (on white background) for the values that can be changed, or display boxes (appearing dimmed) for the values that are calculated from other values. The dialog box contains the following:

Reflectors position

This group box encloses parameters used to calibrate the ultrasonic axis.

Scan 1/Scan 2

These boxes define the real scan axis position of the reference and measurement cursor positions in millimeters (mm) or in inches (in.).

USound 1/USound 2

These boxes defines the real (true depth) reference cursor position in millimeters (mm) or in inches (in.).

Diameter

This box defines the diameter of the hole used for calibration.

Angle calibration

This check box, when selected, enables the calibration of the refracted angle.

Results

The **Sound velocity, Wedge delay, Refracted angle,** and **Scan offset** values displayed in this section give the values calculated from the defined parameters.

Translation / Rotation dialog box

The **Translation / Rotation** dialog box (see Figure 3-147 on page 207) is used to calibrate the translation value relative to the rotation value.

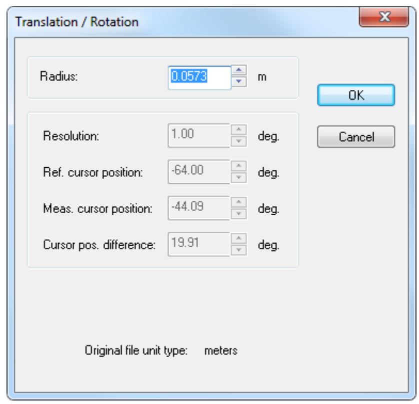


Figure 3-147 The Translation / Rotation dialog box

The **Translation / Rotation** dialog box contains the following:

Radius

This box indicates the radius of the rotary motion.

Resolution

This box indicates the measurement resolution.

Ref. cursor/Meas. cursor

These boxes indicate the reference and measurement cursor positions.

Cursor pos. difference

This box indicates the difference between reference and measurement cursor positions.

Original file unit type

This line displays the original unit used in the data file.

3.13.9 View Linking Tab

The **View Linking** tab (see Figure 3-148 on page 208) is used to set linking cursors types, rulers, palette ranges, and other settings for the active view. The particular linking types set for the active view override the default links set in the **Linking** tab of the **Preferences** dialog box.

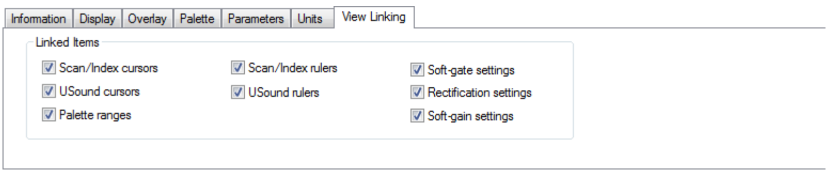


Figure 3-148 The View Linking tab

The **View Linking** tab contains the following group box:

Linked Items

This group box encloses the various objects and settings that can be linked to the graphical tools of the other views. Selecting a check box activates the linking of the corresponding object or setting.

3.13.10 Rebounds Tab

The **Rebounds** tab (see Figure 3-149 on page 208) is used to set the skip processing parameters.

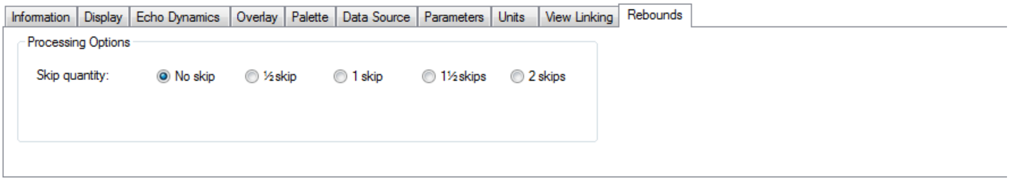


Figure 3-149 The Rebounds tab

Description of the Rebounds tab

The **Rebounds** tab contains the following group boxes:

Processing options

This group box encloses options that are selected for the skip processing.

Skip quantity

Using these option buttons, you can select the number of skips used in the processing from the following: **No skip**, **½ skip**, **1 skip**, **1½ skips**, **2 skips**.

3.13.11 Strip Tab

The **Strip** tab (Figure 3-150 on page 209) is used to set options and parameters related to the strip views. The **Strip** tab is only available when the active view is a strip view.

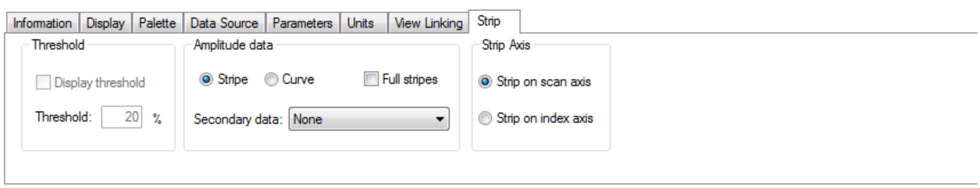


Figure 3-150 The Strip tab

The **Strip** tab is subdivided into **Threshold, Amplitude/Position data**, and **Strip Axis** group boxes.

Threshold group box

This group box encloses the threshold options of the strip views.

- **Display threshold**

  When this check box is selected, the **Threshold** value is displayed as a black line in the active strip view. When the **Secondary data** is displayed, if the secondary data is below the **Threshold** value at a defined position, the associated stripe of the strip view becomes green. If the secondary data is above the **Threshold** value at a defined position, the associated stripe of the strip view becomes red.

- **Threshold**

  Indicates the threshold level in amplitude or as percentage of gate length.

Amplitude/Position data group box

This group box encloses the amplitude/position data options for the strip views.

- **Stripe**

  When this option is selected, the data is represented by a colored stripe. The color and the length of the stripe are indicative of the amplitude or the position of the indication within the associated gate.

- **Curve**

  When this option is selected, the data is represented by a curve.

- **Full stripes**

  When this check box is selected, the data is represented by a colored stripe with its length at full-view height. The color of the stripe is indicative of the amplitude or position of the indication within the associated gate.

- **Secondary data**

  Using this list box, you can choose secondary data to be displayed in the active strip view. Depending on the number of gates, the data type recorded, and the active strip view, the **Secondary data** and the different gate's amplitude and position data.

Strip axis group box

This group box allows you to select whether the strip view should be constructed along the scan or index axis. (available in analysis mode only).

3.14 Preferences Dialog Box

You can customize various TomoView parameters using the **Preferences** dialog box. You access the **Preferences** dialog box by selecting **File > Preferences** on the menu. See Figure 3-151 on page 211.

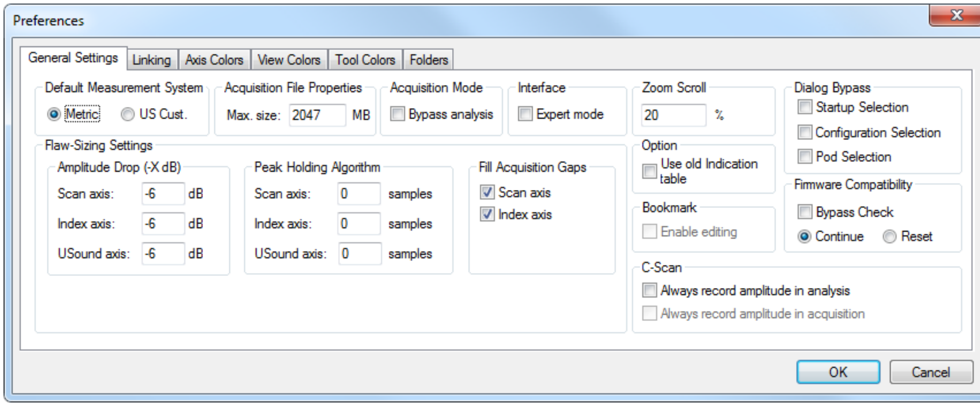


Figure 3-151 The Preferences dialog box

NOTE

The **Preferences** dialog box is not available in the TomoView Lite and TomoViewer editions.

3.14.1 General Settings Tab

The **General Settings** tab of the **Preferences** dialog box (see Figure 3-152 on page 211), contains the following group boxes.

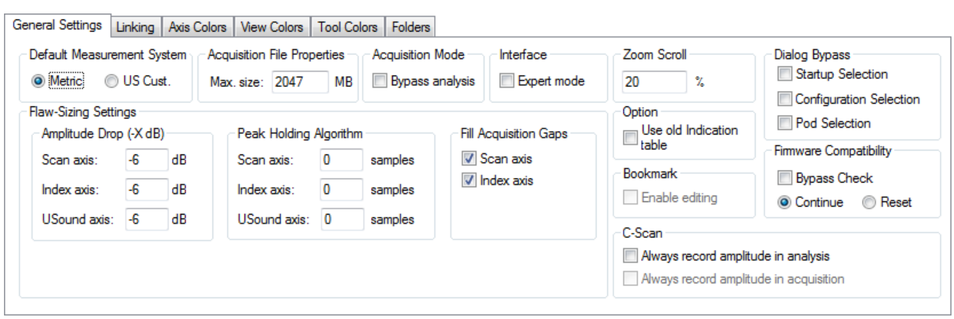


Figure 3-152 The General Settings tab of the Preferences dialog box

Default Measurement System

Select **Metric** or **US Cust.** to respectively use the metric or the US customary measurement systems for all TomoView parameters and views. You can configure units in more detail for each view from the **Properties** dialog box, in the **Units** tab.

Acquisition File Properties

Use **Max Size** to set the maximum size of the acquisition file. The maximum size of a data file that can be recorded with TomoView is 2 GB.

Acquisition Mode

Select the **Bypass analysis** checkbox to bypass the online analysis mode so that you immediately go to the setup mode at the end of the acquisition.

Interface

Select the **Expert mode** checkbox to activate advanced features (see section 3.16 on page 220 for details).

Zoom Scroll

You can scroll the vertical zoom bars on the selected view using the wheel of the mouse. Holding the CTRL key while moving the wheel scrolls the horizontal zoom bar. The axis range percentage value indicates the magnitude of the zoom bar displacement for one mouse wheel tick.

Dialog Bypass

Select the check box for the dialog boxes (**Startup Selection**, **Configuration Selection**, or **Pod Selection**) that you want to bypass at startup. These options may be useful when you always use the same setup and want to avoid acknowledging the default setup, or when you remotely control TomoView and you cannot respond to the dialog boxes.

Amplitude drop

Set the amplitude drop negative value for the three axes to be used for flaw sizing with the Zone tool.

Peak Holding Algorithm

Set the number of samples to be used in the peak holding algorithm for the three axes for flaw sizing with the Zone tool. A setting of *n* samples means that the algorithm ignores an amplitude drop below –*x* dB along the axis if it is smaller than *n* samples.

Fill Acquisition Gaps

Select the check box to activate the function that fills data acquisition gaps when performing amplitude-drop sizing measurements for each of the two axes.

Option

Select the **Use old Indication table** check box when you wish to rather use the legacy Indication table available in TomoView prior to version 2.9. You need to restart TomoView to see the effect after changing this option.

Bookmark

The **Enable edition** check box is disabled because its use is reserved for Evident NDT.

Firmware Compatibility

At startup, TomoView validates its compatibility with the acquisition unit firmware. TomoView displays a dialog box when an incompatibility is detected (see Figure 3-153 on page 213).

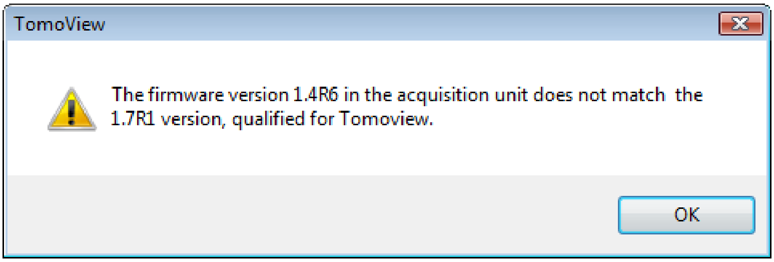


Figure 3-153 Example of dialog box for an incompatible OmniScan firmware program

Select the **Bypass Check** check box when you wish not to display the dialog box. This option may be useful when you remotely control TomoView and no one is present in front of the TomoView computer to respond to the dialog box.

For connection to acquisition units other than an OmniScan, when the **Bypass Check** check box is selected, choose one of the following option to determine what TomoView does when a firmware incompatibility is detected:

Continue

TomoView continues to use the acquisition unit with the current firmware.

Reset

TomoView uploads the compatible firmware to the acquisition unit through the Bootp server.

C-Scan

On an amplitude C-scan, TomoView normally does not show signal amplitudes that are below the gate level, replacing these values by **no detection** codes.

Select the **Always record amplitude in analysis** check box when you want to configure TomoView to show this signal in Analysis mode, after moving the gate. Select **Always record amplitude in acquisition** if you want the amplitude data to be shown whether or not the signal has crossed the gate (therefore ignoring the **no detection** codes).

3.14.2 Linking Tab

The **Linking** tab of the **Preferences** dialog box (see Figure 3-154 on page 215), accessible from the **File > Preferences** menu selection, contains parameters to customize the linking features in TomoView.

When an element is linked, changing it in one context (view, group, gate) automatically changes it identically in the other linked contexts. For example, with the **Scan cursors** set to **Fully linked**, moving the reference cursor on the scan axis in one view, moves it to the same position in all the other views. The linking feature can save you a large number of tedious adjustments when you work with multiple views, multiple groups, and multiple gates. You can set the cursor and ruler linking behavior differently for uncorrected and corrected views.

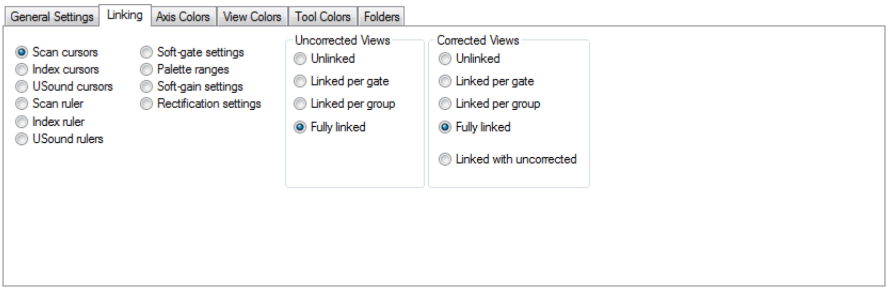


Figure 3-154 The Linking tab of the Preferences dialog box

TIP

You can also configure the links for individual views from the **View Linking** tab in the **View Properties** dialog box (see section 3.13.9 on page 208 for details).

The available linking behaviors with other views are:

Unlinked

The selected item is not linked.

Linked per gate

The selected item is linked per gate.

Linked per group

The selected item is linked per group.

Fully linked

The selected item is linked for all gates and all groups.

Linked with uncorrected

The selected item is linked as defined under **Uncorrected Views**.

NOTE

TomoView classifies an A-scan as an uncorrected view, even when the ultrasound axis is set to true depth mode.

3.14.3 Colors Tabs

The **Axis Colors**, **View Colors**, and **Tool Colors** tabs of the **Preferences** dialog box contain buttons that you can use to view or select the color of various TomoView items. You can access dialog box by selecting **File > Preferences** on the menu.

The **Axis Colors** tab (see Figure 3-155 on page 216) displays the colors of the different axes for uncorrected and corrected views. These colors are shown only for reference purposes. You cannot change these colors.

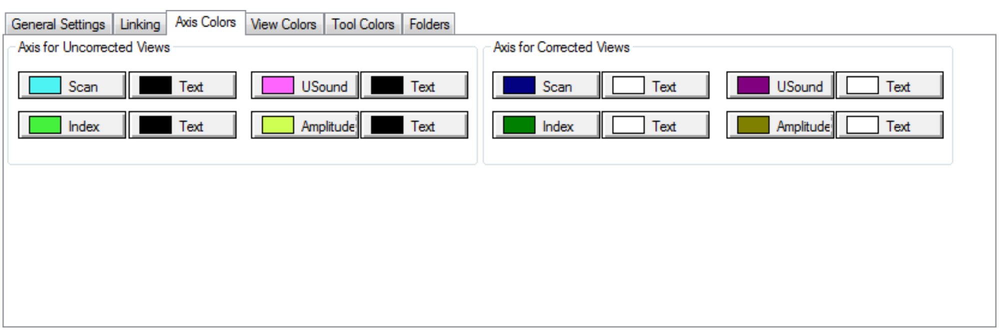


Figure 3-155 The Axis Colors tab of the Preferences dialog box

The **View Colors** tab (see Figure 3-156 on page 217) is used to set the colors of the background, the grid, the curve, the autotracking measurement cursor, and the DGS curves in the views.

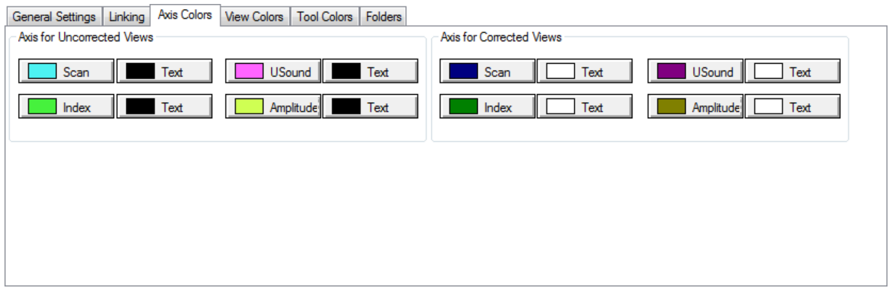


Figure 3-156 The View Colors tab of the Preferences dialog box

The **Tools Colors** tab (see Figure 3-157 on page 217) is used to set the color of the four information groups and of the overlays.

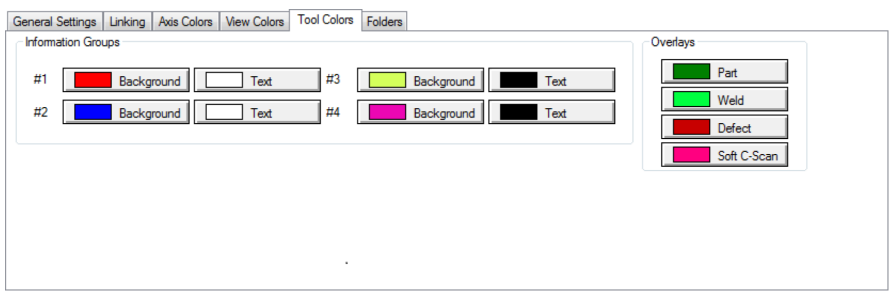


Figure 3-157 The Tool Colors tab of the Preferences dialog box

Clicking any of the previously mentioned buttons opens a standard **Color** dialog box, in which you select a predefined color or define a new custom color (for details, see section 10.2 on page 449).

3.14.4 Folders Tab

You can customize the default folders for the various data file types. On the menu, select **File > Preferences**, and then select the **Folders** tab (see Figure 3-158 on page 218).

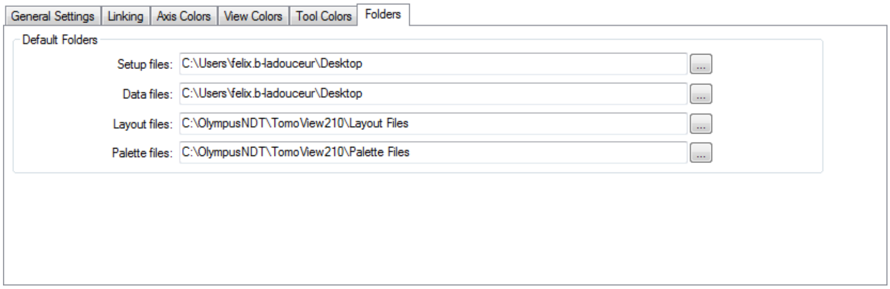


Figure 3-158 The Folders tab of the Preferences dialog box

3.15 About the Advanced Calculator

The Advanced Calculator is proprietary Evident software installed on the TomoView. Start the Advanced Calculator from the **TomoView Manager** dialog box by clicking the  button. From TomoView, use the Advanced Calculator to specify the probe and wedge used in the inspection, the geometry and material of the inspected part, and the beam configuration (see Figure 3-159 on page 219). The Advanced Calculator calculates the beams and returns the information to TomoView.

Refer to *Advanced Calculator User's Manual* (DMTA080-01EN) for more information.

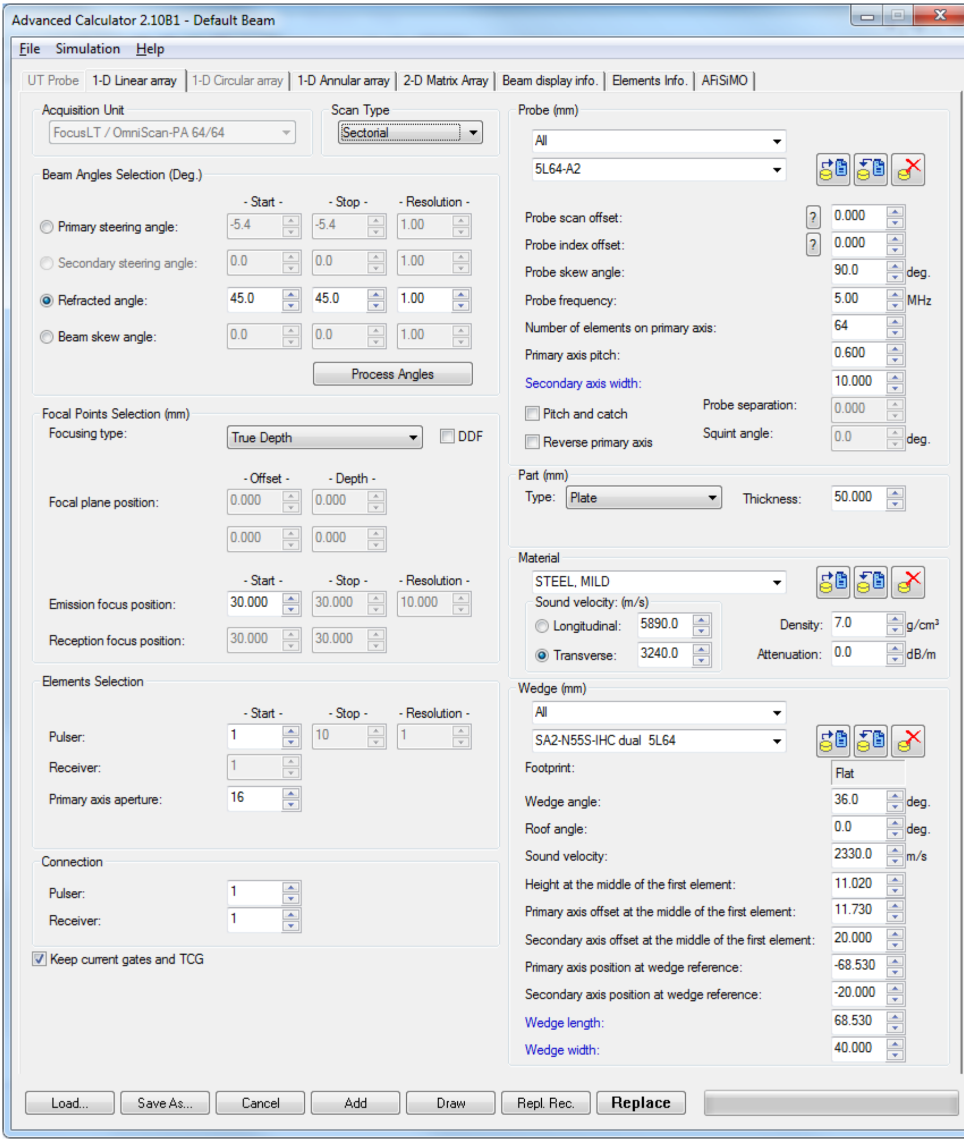


Figure 3-159 The Advanced Calculator dialog box

3.16 About the Expert Mode

TomoView includes an expert mode in which a larger number of readings are available. The categories of readings available in the **Information Groups** dialog box when the expert mode is active are shown in Figure 3-160 on page 220.

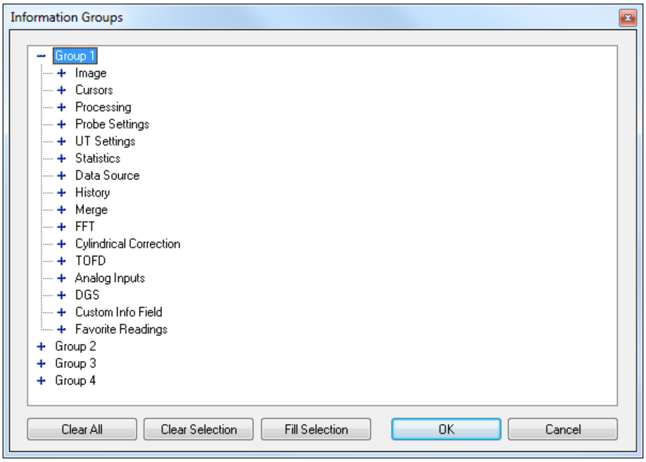


Figure 3-160 The categories of readings in the expert mode

To activate/deactivate the expert mode

1. On the menu, select **File > Preferences**.
2. In the **Preferences** dialog box, select the **General Settings** tab.
3. Under **Interface**, select or clear the **Expert mode** check box, respectively, to activate or deactivate the expert mode.

3.17 About the Master and Slave Modes

TomoView provides the software components necessary to control motorized or manual inspection sequences and is available in either **Master** or **Slave** mode.

Master Mode

In the *master* operation mode, the MCDU-02 system controls the motors of the mechanical system. The MCDU-02 supports input from two digital-position encoders (refer to the *MCDU-02 User's Manual*, part number DUMG017B for specifications and connection methods for these encoders). In the *master* mode, TomoView supports both the single-axis and dual-axis MCDU-02 unit scan capabilities. With TomoView, you can use the motor control drive unit with Evident NDT scanners and ultrasonic acquisition systems, as well as third-party scanners.

Slave Mode

In the *slave* operation mode, the mechanical system is not controlled by TomoView. Instead, the mechanical system is controlled by an external control unit, or is moved manually.

In order for TomoView to start in *master* mode, an MCDU-02 (Motor Control Drive Unit) must be selected along with the acquisition unit at startup (see Figure 3-161 on page 221). TomoView will operate in master mode if a MCDU-02 (Motor Control Drive Unit) is selected at startup. Otherwise, it startup in *slave* mode.

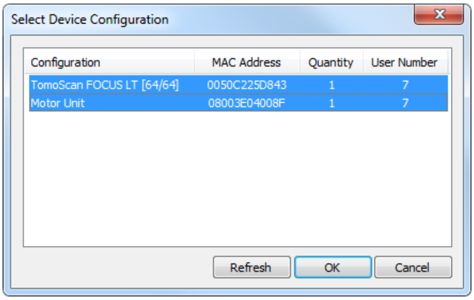


Figure 3-161 The Select Device Configuration dialog box

3.18 File Formats

TomoView can produce, opens, and imports various data types stored in various file formats listed in Table 11 on page 222 and Table 12 on page 222.

Table 11 File formats supported by TomoView

File type	Extension	File content
Hardware setup	.acq	Setup of the acquisition unit
Data	.rdt	Data file in RDTIFF (Evident proprietary file format) format. Also contains display setup information and acquisition setup information.
	.opd	OmniScan phased array (PA) data file imported into TomoView using the OmniScan File Converter.
	.oud	OmniScan UT data file imported into TomoView using the OmniScan File Converter.
Display setup	.rst	Display setup: layouts and system preferences
Scanner settings	.scn	Scanner setting PID parameters
Part	.rsp	Part definition parameters
Report	.r01	Report information including indication zone data
Calculator setup	.xcal	Advanced Calculator extended setup file
	.law	Calculated beam parameters readable by the OmniScan
	.pac	Calculated beam parameters readable by TomoView
Attributes	.A01	Attributes: data modified in analysis and associated with the .rdt file

Table 12 Legacy file formats supported by TomoView

File type	Extension	File content
Data	.dat	Data file in Tomoscan format
Hardware Setup	.srd	Legacy TomoView setup file for the µTomoscan and TomoScan FOCUS acquisition units
Calculator setup	.cal	Advanced Calculator setup file

The hierarchical content of the file formats generated by TomoView is illustrated in Figure 3-162 on page 223.

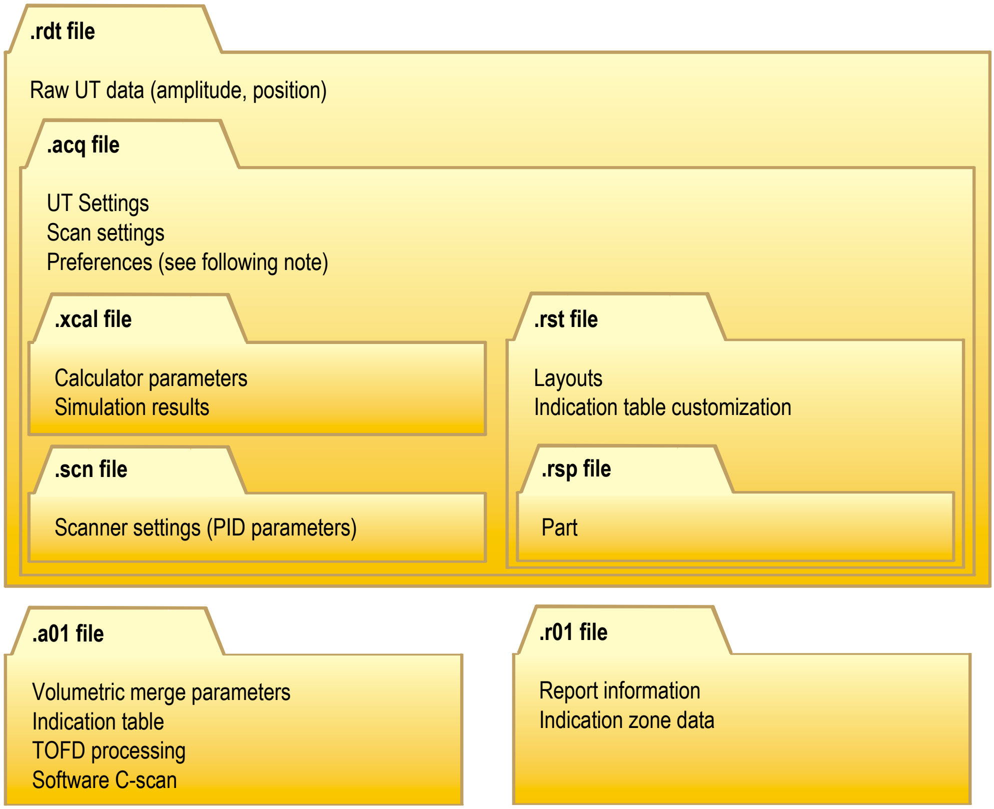


Figure 3-162 TomoView file organization

NOTE

The preferences data saved in the .rst file include view linking options, axis colors, view colors, tool colors, and the measurement system. By default, the contents of the .rst file is included in the .acq file.

NOTE

You can customize the default folders for the various data file types (see Section 3.14.4 on page 217 for details).

Part Two: Working with TomoView

4. Setup Creation

A TomoView configuration can be saved in the **.acq** (acquisition, also referred to as the setup) file, which can be recalled at any time. This section describes the basic steps for creating a setup file using TomoView, starting from the default configuration.

4.1 Working with Setup Files

The following section describes how to create the setup files.

4.1.1 Loading an .acq Setup File

The TomoView configuration can be stored in the .acq file. An .rdt (data) file can also be recalled to load the .acq configuration which was used at the time when the data was recorded. You can reload the parameters into the ultrasonic acquisition unit to perform acquisition with the same parameters or to display the parameters that were used for acquisition.

To load a TomoView acquisition file

1. From the document Toolbar, click  to open a setup file.
   The **Open** dialog box appears (see Figure 4-1 on page 228).

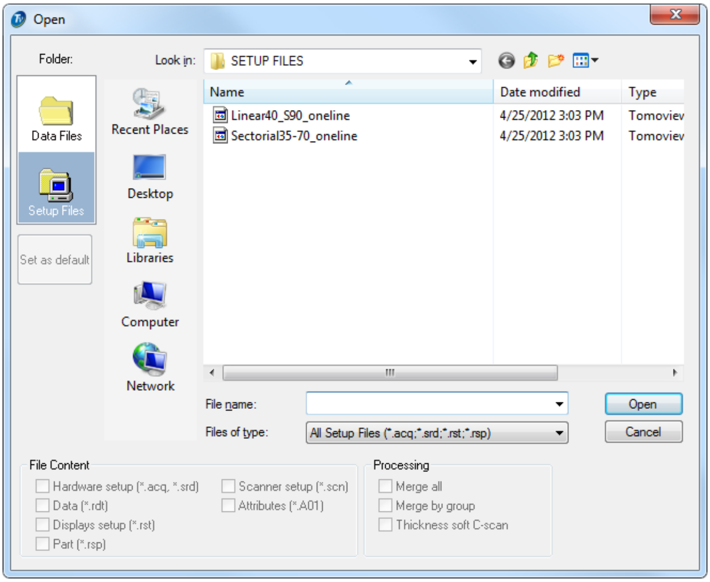


Figure 4-1 The Open dialog box

2. In the **Open** dialog box that appears (see Figure 4-1 on page 228):
   a) Select the configuration type from **Data File** or **Setup File**.
   b) From the **File Content** section, select the file to be loaded and choose the elements to be loaded.
   c) Click **Open**.

4.1.2 Loading an SRD Setup File

TomoView can load .srd files, along with the .acq files. The .srd files were created using older software versions.

To open an .srd setup file

1. From the toolbar, select **File > Import SRD file.**
2. When the Open dialog box appears, select the .srd file to load and click **Open.** (See Figure 4-2 on page 229.)

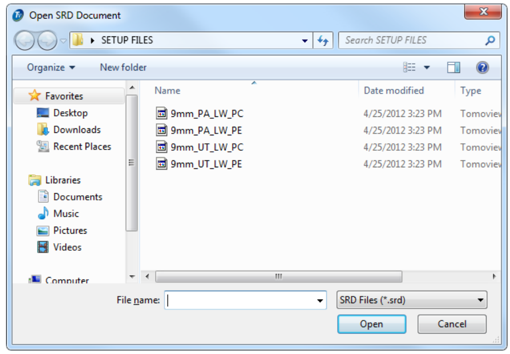


Figure 4-2 The Open SRD Document dialog box

3. When the setup is loaded:
   a) Configure the **Display** layout.
   b) Configure the **Scan and Mechanical Settings** parameters.
   c) Click **Open**.
4. Save the setup in an .acq file using the standard save option.

4.1.3 Saving an .acq Setup File

To save a setup

1. From the **Document** toolbar, click  to save a setup file.
2. In the **Save As** dialog box that appears (see Figure 4-3 on page 230):
   a) For the **.acq** file to be saved, enter the **File Name**.
   b) In the **File Content section,** select the elements to be saved.
   c) To make this setup file the new default configuration, click on **Save** to save the .acq file, or click **Save as default**.

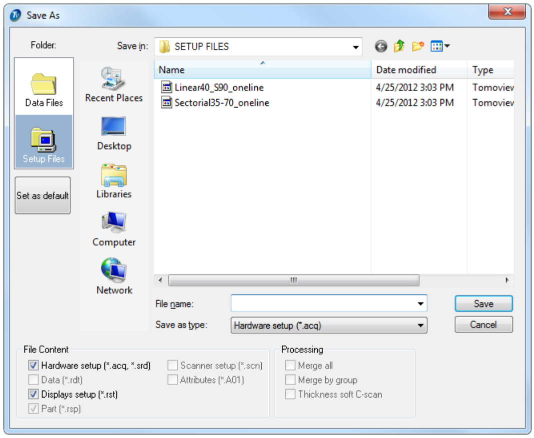


Figure 4-3 The Save As dialog box

4.1.4 Default Setup Files

TomoView comes with a number of default setup files. Each default setup file contains parameters appropriately configured for a particular common application. Loading an appropriate default setup file for your application is an easy way to discover TomoView advanced features.

The default setup files are available in the default setup file folder ([Installation Folder]\EvidentNDT\TomoView210\Setup Files).

4.2 Working with Groups

In TomoView, a *group* is a defined configuration of parameters to generate one or more ultrasonic beams using a conventional or phased array probe. A group can pulse and receive on the same probe, or you can use two different probes, one to pulse, and the other to receive, and a probe can be used by more than one group. The following sections describe how to create and configure phased array and conventional groups (see section 3.2 on page 85 for more information on groups).

4.2.1 Adding and Configuring a Phased Array Group

The following procedure describes how to create a phased array group in TomoView. Although some of the steps may not be mandatory for all inspection types, it provides an overview of the power and flexibility of TomoView.

To add and configure a phased array pulse-echo group

1. In the **TomoView Manager**, click  to add a group (see Figure 4-4 on page 231).

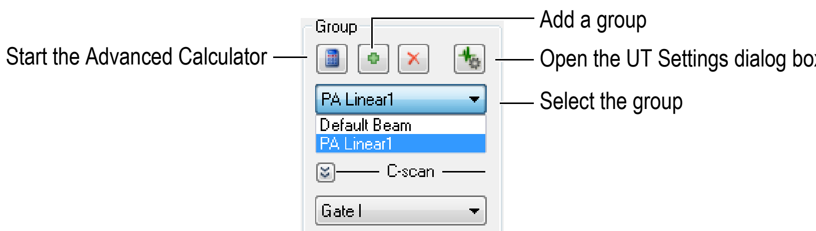


Figure 4-4 Selecting the newly created group

2. In the **Group Creation Wizard** dialog box that appears (see Figure 4-5 on page 232):
   a) Select **Phased array** for the type of group to create.
   b) Enter a name for the new group in the **Specify name** text box (for example: Angle Beam).
   c) Click **Finish**.

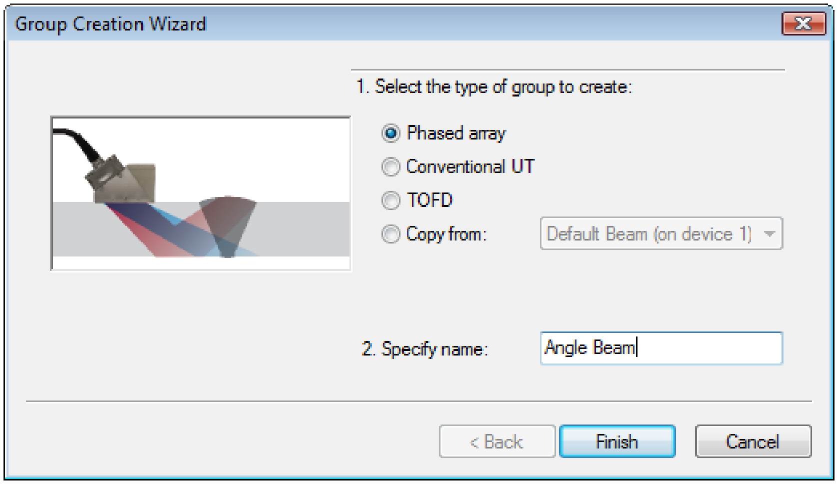


Figure 4-5 The Group Creation Wizard dialog box

3. In the **TomoView Manager**, in the **Group** area (see Figure 4-4 on page 231):
   a) Select the newly created group.
   b) Click  to open the Advanced Calculator.
   The Advanced Calculator is where you specify the probe and wedge used for the inspection, the geometry and material of the inspected part, and the beam configuration. See section 3.3 on page 87 for details.
4. In the Advanced Calculator, perform the following tasks:
   a) For an angle beam phased array inspection, click the **1-D Linear Array** tab (see Figure 4-6 on page 233).

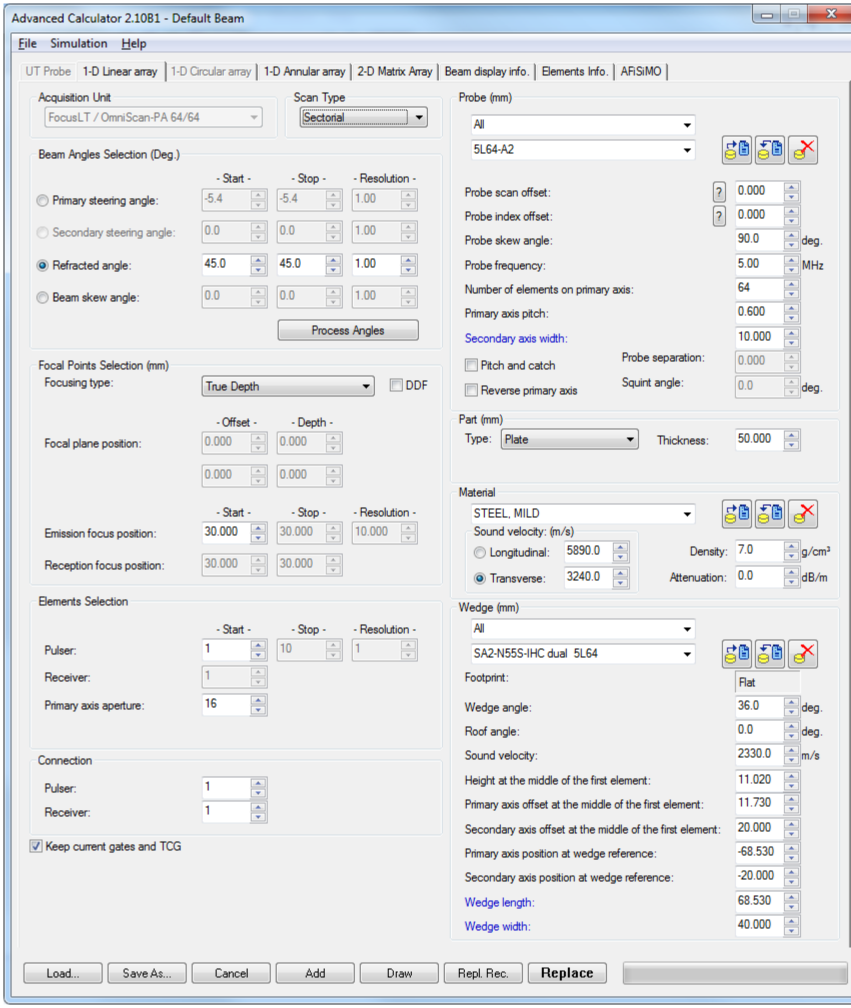


Figure 4-6 The Advanced Calculator dialog box

b) In the **Probe** group box, select the type of probe (**Angle Beam**) and the probe model (see Figure 4-7 on page 234).

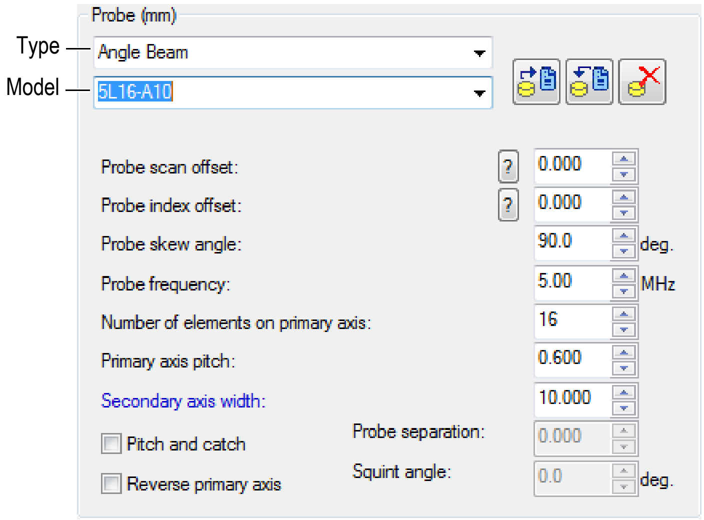


Figure 4-7 Selecting the probe

c) In the **Part** group box, select the inspected part geometry type and enter its thickness (see Figure 4-8 on page 234).

Figure 4-8 Specifying the inspected part geometry

d) In the **Material** group box, select the inspected part material and the **Longitudinal** or **Transverse** ultrasound wave type (see Figure 4-9 on page 235).

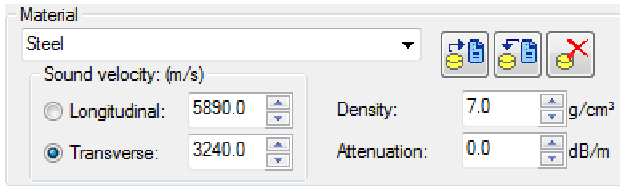


Figure 4-9 Specifying the inspected part material

e) In the **Wedge** group box, select the type of wedge and the wedge model (see Figure 4-10 on page 235).

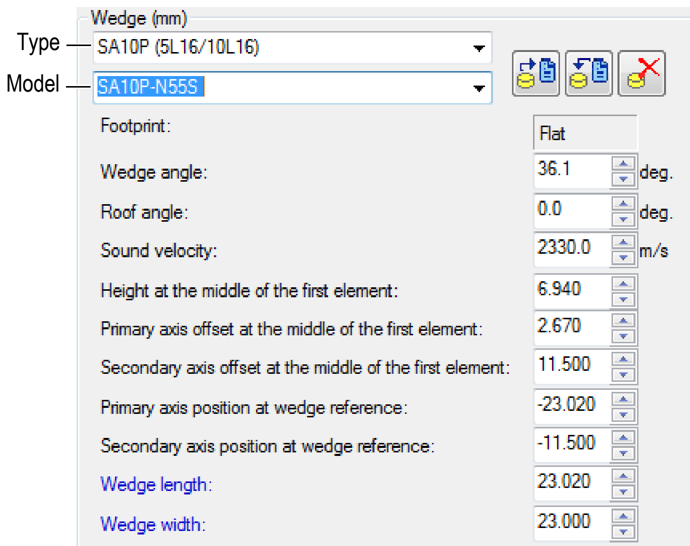


Figure 4-10 Specifying the wedge

f) In the **Scan Type** group box, for an angle beam inspection, select **Sectorial, Depth, or Static**. For more information on the differences between the Scan Types, refer to the *Advanced NDT Series* books available for free on our website.

g) In the **Beam Angles Selection** group box, select **Refracted angle**, and then specify the **Start** value along with the **Stop,** and **Resolution** values if available depending on the selected **Scan Type**.

h) In the **Focal Points Selection** group box, set the **Focusing type** to **True Depth**, **Half Path**, **Projection**, or **Focal Lane**, and if desired, select if you want to have Dynamic Depth Focusing (DDF) activated. Then define the **Focal plane position, Emission focus position**, and **Reception focus position** if available.

i) In the **Elements Selection** group box, select **Primary axis aperture**, and then enter the number of elements to use for each focal law. Then define the **Start, Stop**, and **Resolution** values if available.

j) In the Connection group box, set the Pulser and the Receiver values. This will define the element to use as the first element for the transmitter and receiver focal laws.

k) At the bottom of the window, click **Draw**.
TomoView prepares the beam display information

l) Wait for the progress bar to complete.

m) At the top of the screen, click the **Beam display info.** tab.

n) Use the four views and the parameters below the views to validate the calculated beams (see Figure 4-11 on page 237).

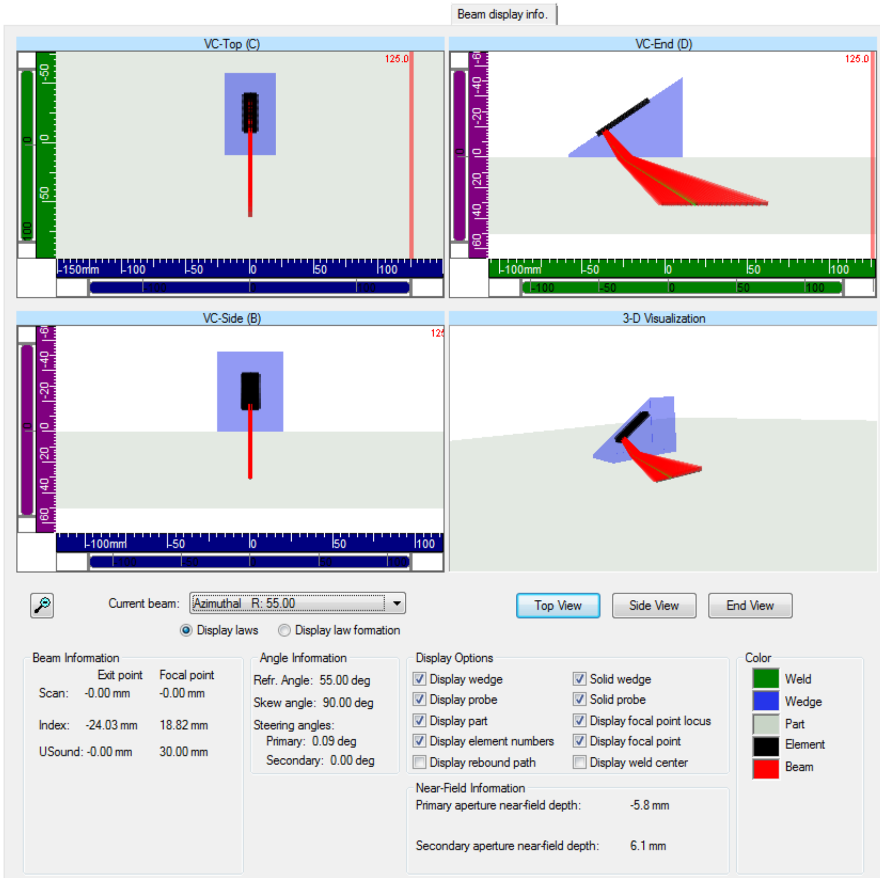


Figure 4-11 Example of the Beam Display Info. tab in the Advanced Calculator

o) Return to the **1-D Linear array** tab, and then if necessary, make further adjustment to the parameters.

p) At the bottom of the window, click **Replace** to calculate the configured focal laws and send the information back to TomoView.

To add and configure a phased array Pitch-and-Catch or Trough Transmission group

1. Follow steps 1 to 4.*e* of the procedure called “To add and configure a phased array pulse-echo group” on page 231.

2. Click  to open the **Advanced Calculator**:
   a) In the **Scan Type** group box, select **Static**.
   b) In the **Beam Angles Selection** group box, select **Refracted angle**, and then specify the **Start** value for the Pulser.
   c) In the **Elements Selection** group box, define the **Start** and **Primary axis aperture** values.
   d) At the bottom of the window, click **Replace** to calculate the configured focal laws and send the information back to TomoView.

3. Click  to open the **Advanced Calculator**:
   a) In the **Beam Angles Selection** group box, select **Refracted angle**, and then specify the **Start** value for the Receiver.
   b) In the Elements Selection group box, define the **Start** and **Primary axis aperture** values.
   c) In the **Probe** group box, define the **Probe skew angle** for the receiver, if necessary.
   d) At the bottom of the window, click **Replace Rec.** to calculate the configured reception focal laws and send the information back to TomoView.

4. Click  to open the **Advanced Calculator**:
   a) In the **Phased Array Parameters Source** dialog box (see Figure 4-12 on page 239), select the sector to be edited, then select **Emission** and click **Ok**.

      The **Advanced Calculator** will open with the corresponding Pulser configuration.
   b) In the **Advanced Calculator**, click on **Add**.

      This will add a new beam with a similar configuration to the one defined in step 2.

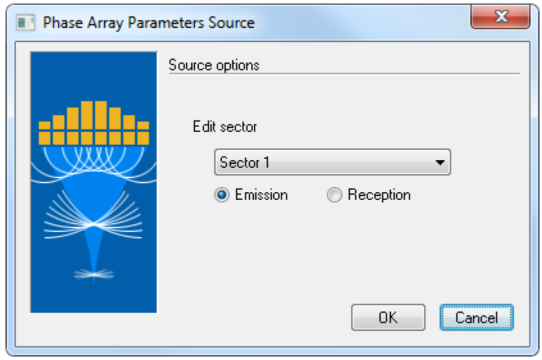


Figure 4-12 The Phased Array Parameters Source dialog box

5. Click  to open the **Advanced Calculator**:
   a) In the **Phased Array Parameters Sourc**e dialog box (see Figure 4-12 on page 239), select the sector to be edited and then select **Emission** and click **Ok**.
   b) Edit the different available parameters, if necessary.
   c) At the bottom of the window, click **Replace** to calculate the configured focal laws and send the information back to TomoView.

6. Click  to open the **Advanced Calculator**:
   a) In the **Phased Array Parameters Sourc**e dialog box (see Figure 4-12 on page 239), select the sector to be edited and then select **Reception** and click **Ok**.
   b) Edit the different available parameters, if necessary.
   c) At the bottom of the window, click **Replace Rec.** to calculate the configured reception focal laws and send the information back to TomoView.
7. Repeat steps 4 to 6 until all your beams are properly configured.

4.2.2 Adding and Configuring a Conventional UT Group

The following procedure describes how to create a conventional UT group in TomoView. The configuration of a conventional UT group is not performed from the Advanced Calculator as for the phased array groups. Therefore, the acoustic configuration and part definition will need to be performed separately.

To add and configure a conventional UT group

1. In the **TomoView Manager**, click  to add a group.
2. In the **Group Creation Wizard** dialog box that appears (see Figure 4-13 on page 240):
   a) Select **Conventional UT** for the type of group to create.
   b) Enter a name for the new group in the **Specify name** text box (for example, Conventional Beam).
   c) Click **Finish**.

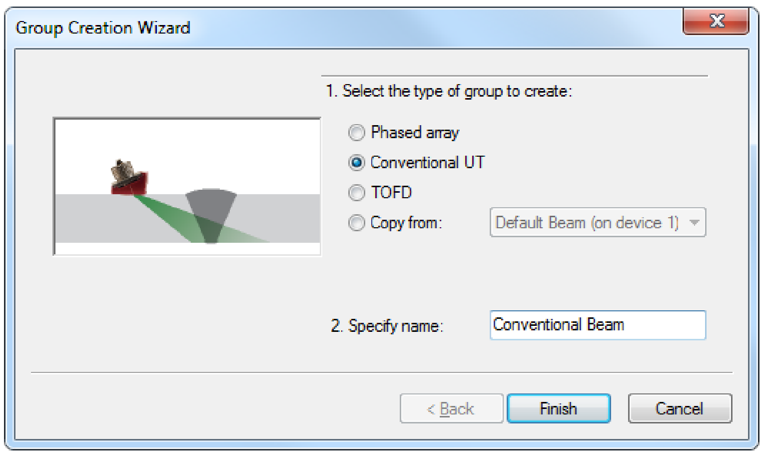


Figure 4-13 The Group Creation Wizard dialog box

3. On the menu, select **Part & Overlay > Part Definition**.
4. In the **Part Definition** dialog box (see Figure 4-14 on page 241):
   a) In the **Material Velocity** area, set the ultrasonic velocities for the **Longitudinal waves** and **Transversal waves** in the part.
   b) In the **Dimensions** area, set the values describing your part.
   c) For cylindrical parts, in the **Probe Positioning** area, select the appropriate **Inspection from** and **Scan orientation** values.
   d) Click **OK**.

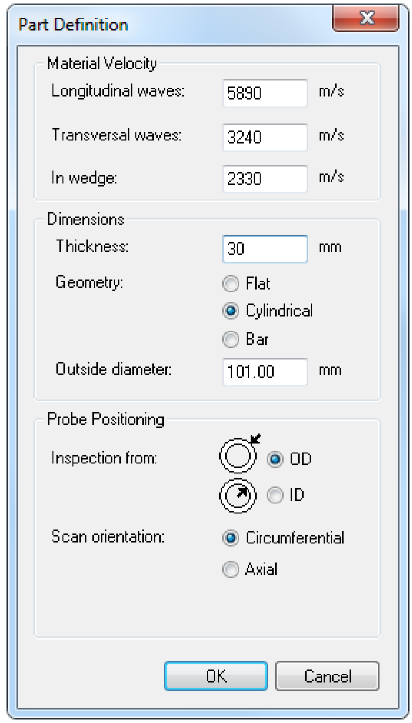


Figure 4-14 The Part Definition dialog box

5. In the **TomoView Manager**, in the **Group** area:
   - Select the newly created group (see Figure 4-15 on page 241).

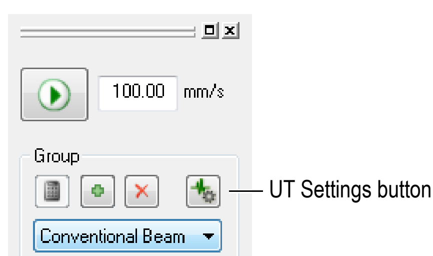


Figure 4-15 Selecting the UT group in the TomoView Manager

6. On the Component toolbar, click the UT Settings button ().
7. In the **UT Settings** dialog box:
   a) In the **Pulser/Receiver** tab, set the **connectors** for **Pulser** and **Receiver** along with the **Pulse width** value for your UT probe.
   Divide 500 by the probe frequency expressed in MHz to get the pulse width value expressed in ns (ex.: 500/10 MHz = 50 ns).
   b) In the Probe tab, under Selection, select Modify probe, and under Beam Orientation, enter the appropriate Refracted angle value.
   c) In the **Probe** tab, select **Modify probe** in the **Selection** group box.
   d) In the **Position** group box, define the **Scan offset** and **Index offset** values.
   e) In the **Beam Orientation** group box, define the **Skew angle** value.

4.2.3 Adding and Configuring a TOFD Group

The following procedure describes how to create a TOFD group in TomoView. Just as for a Conventional UT group, the TOFD configuration is not performed from the Advanced Calculator. Therefore, the acoustic configuration is performed separately.

To add and configure a TOFD group

1. In the **TomoView Manager**, click  to add a group.
2. In the Group Creation Wizard dialog box that appears (see Figure 4-16 on page 243).
   a) Select **TOFD** for the group type.
   b) Enter a name for the new group in the **Specify name** text box (for example, TOFD group).
   c) Click **Finish**.

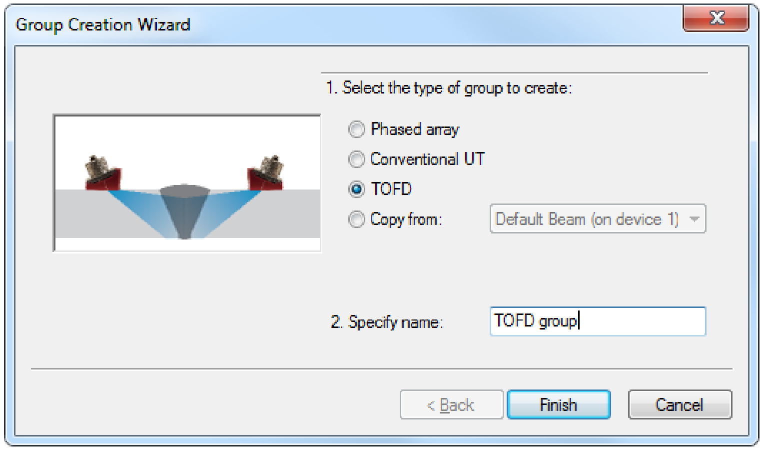


Figure 4-16 The Group Creation Wizard dialog box

3. In the **UT Settings > Pulser/Receiver** dialog box:
   a) Set **Connector** to the value corresponding to the Pulser connector number.
   b) Set **Receiver** to the value corresponding to the Receiver connector number.
   c) Set the **Voltage** to the highest value available.
   d) Set the **Pulse width** value for your UT probe.
   Divide the number 500 by the probe frequency (expressed in MHz) to obtain the pulse width value expressed in ns (e.g., 500/10 MHz = 50 ns).

4.2.4 Switching Between Groups

When you have more than one group in a configuration, you often need to switch the focus from one group to the other.

To switch between groups

1. On the menu, select **Tools > Change Group**.
2. In the **Change Group** dialog box, select the group to which you want to switch, and then click **OK**.
   OR
   On the menu, select **Tools > Previous Group** or press the F5 key to switch to the previous group.

OR

On the menu, select **Tools > Next Group** or press the F6 key to switch to the next group.

4.2.5 Renaming Groups

To rename a channel

1. Click  to open the **UT Settings** dialog bar.
2. In the **UT Settings** dialog bar, select the group you want to rename.
3. Click in the **Group** box, and then type the new group name.
4. Press **Enter**.

4.2.6 Deleting Groups

To delete a channel

In the **TomoView Manager** in the **Group** area:

1. Select the group you want to delete.
2. Click on the **Delete** button.

   A message box appears (see Figure 4-17 on page 244), asking you to confirm deletion of the channel.

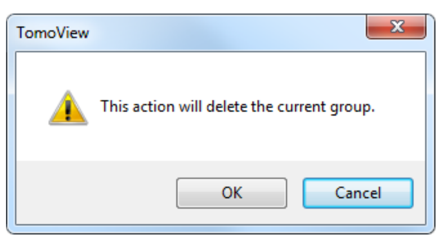


Figure 4-17 The message box for deleting a channel

3. Click **OK** to confirm the operation.

4.3 Calibrating a Phased Array Group

Before and after an inspection, you can perform an on-site calibration check to ensure the stability of the complete ultrasonic (UT) system. The phased array technique requires the calibration and verification of all the UT beams. The purpose of the calibration is to obtain a setup file that yields correct results regarding the position and amplitude of a known reflector in a calibration block.

The following sections present the calibration procedures to calibrate the **Beam Delays** and **Sensitivity** for phased array groups. For the phased array TCG calibration procedure, see section 4.7.3 on page 258.

4.3.1 Calibrating the Beam Delays

The purpose of the phased array beam delay calibration is to adjust the delay of each beam so that the indication of a known reflector appears at the correct depth for all beams. This procedure must be performed for each group.

To calibrate the beam delays

1. On the **Component** toolbar, click  (Phased Array Calibration).
2. In TomoView, in a Side (B), End (D), or Sector (S) view, position the **Reference** (red) and **Measurement** (blue) cursors respectively above and below the reference reflector of the calibration block (see Figure 4-18 on page 245).

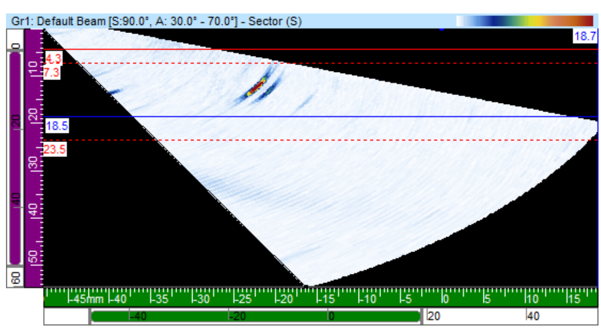


Figure 4-18 Example of cursors around an indication in a Sector (S) view

3. In the **Phased Array Calibration** dialog box:

a) Select the **Beam Delays** tab (see Figure 4-19 on page 246).
The graph presents the ultrasonic path between the **Reference** and **Measurement** cursors on the vertical axis and the beams on the horizontal axis.

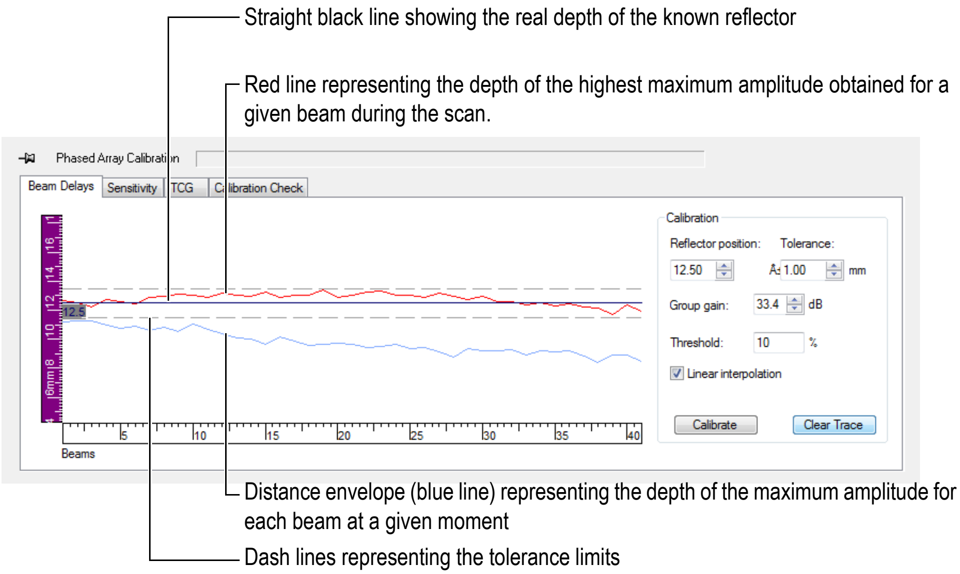


Figure 4-19 Curves for the beam delay calibration

b) Under **Calibration**, in the **Reflector position** parameter, enter the true depth of the known reflector used for this calibration.

c) If the reflector echo is too weak or too strong, adjust the **Group gain** parameter.

d) In the **Tolerance** parameter, enter the acceptable calibration tolerance.

e) Select the **Linear interpolation** check box to activate the averaging of the beam delays. The resulting line applies a global correction.

4. During the scan, the distance envelope (blue line) obtained for each beam is drawn, and a curve representing the position of the maximum amplitude is constructed (red line).
5. Place the probe on the calibration block and perform a first scan above the reference reflector.

IMPORTANT

After a scan, the maximum amplitude curve (red line) should already be close to the tolerance limits. When the maximum amplitude curve shape is significantly different, in the Advanced Calculator, revise the configuration of all parameters.

6. In the **Phased Array Calibration** dialog box, click **Clear Trace** to clear the data from the previous scan.
7. Perform a new scan above the reference reflector on the calibration block.
   In the graph, the red line might not always appear within the tolerance lines.
8. Click **Calibrate**.
   The graph content is cleared and TomoView calculates the wedge delay for each beam so that the reflector indication appears at the requested depth.
9. Perform a new scan above the reference reflector on the calibration block to validate the calibration by confirming that the red line appears within the tolerance lines.

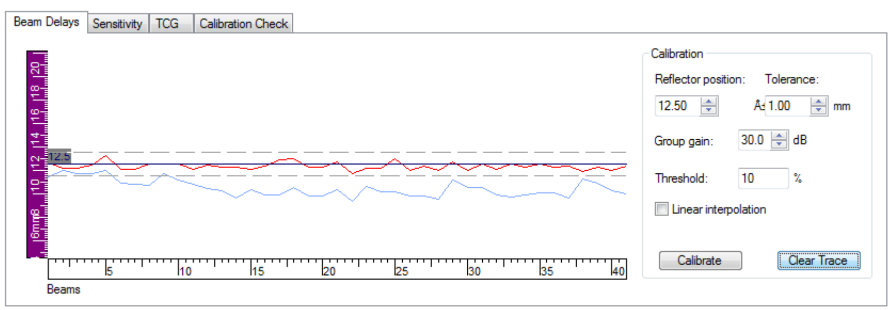


Figure 4-20 Red line appearing between tolerance lines after calibration

10. When the calibration is not successful, repeat steps 6 to 9.
11. If you need to reset the calibration:

    a) On the TomoView Manager, click the Advanced Calculator button ().

    b) In the **Advanced Calculator**, click **Replace**.

4.3.2 Calibrating the Sensitivity

The purpose of the phased array sensitivity calibration is to adjust the gain of each beam so that the amplitude of a known reflector appears at the same level for all beams.

To calibrate the sensitivity

1. On the **Component** toolbar, click  (Phased Array Calibration).
2. In the **Phased Array Calibration** dialog box, click the **Sensitivity** tab.
3. Couple the probe to the calibration block and perform a first scan on the calibration block above a reference reflector.
   During the scan, TomoView draws the distance envelope (blue line) obtained for each beam and constructs a curve (red line) representing the position of the maximum amplitude (see Figure 4-21 on page 248).

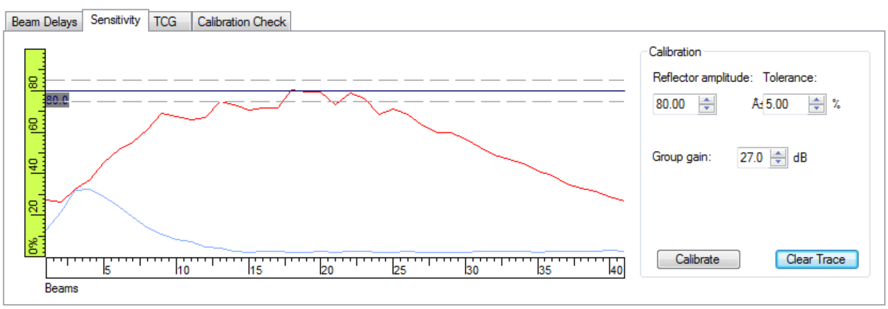


Figure 4-21 Sensitivity example after a first scan

4. In TomoView, in a Side (B), End (D), or, Sector (S) view, position the reference (red) and measurement (blue) cursors respectively above and below the reference reflector of the calibration block (see Figure 4-18 on page 245).
5. In the **Phased Array Calibration** dialog box:
   a) Set the **Reflector amplitude** parameter to the amplitude (%) at which you wish to see the reference reflector maximum amplitude appear.
   b) In the **Tolerance** parameter, enter the acceptable amplitude tolerance.

c) If the reflector echo is too weak or too strong, adjust the **Group gain** parameter.

d) Click **Clear Trace** to clear the data from the previous scan.

6. Perform a new scan above the reference reflector on the calibration block.
7. Click **Calibrate**.
   TomoView clears the graph content and calculates the gain for each beam so that the reflector echo amplitude appears at the requested level for all beams.
8. Perform a new scan above the reference reflector on the calibration block to validate the calibration by confirming that the red line appears within the tolerance lines (see Figure 4-22 on page 249).

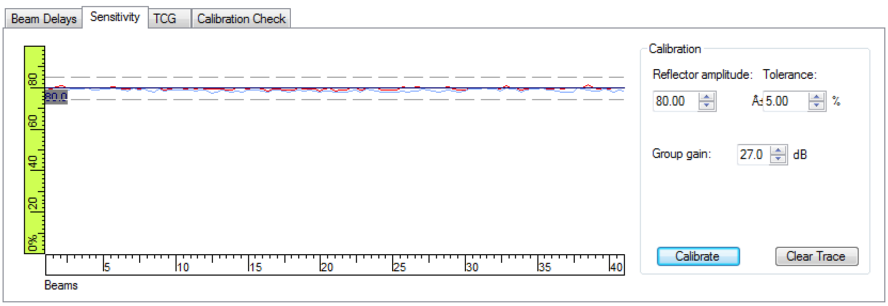


Figure 4-22 Curves for the sensitivity calibration

9. When the calibration is not successful, repeat steps 6 to 8.
10. If you want to review or reset the beam gains:

a) On the **Component** toolbar, click  (UT Settings).

b) In the **Beam** list above the tabs, select the desired beam.

c) In the **General** tab, under **Gain**, review or edit the **Beam** gain.

d) Click **Reset Beam** when you want to reset the sensitivity calibration.

4.4 Calibrating a Conventional UT Group

You can calibrate the ultrasonic velocity and the wedge delay of a conventional UT group using the parameters available from the **General** tab of the **UT Settings** dialog box (see Figure 4-23 on page 250).

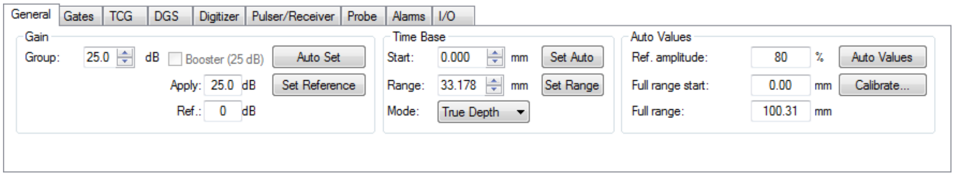


Figure 4-23 The General tab of the UT Settings dialog box for a UT group

- You need a calibration block with one known reflector to perform the delay calibration and a calibration block with two known reflectors to perform the velocity calibration.

To calibrate a conventional UT group

1. On the Component toolbar, click  (UT Settings).
2. In the **TomoView Manager**, in the **Group** area, select the Conventional UT group that you want to calibrate.
3. In the **UT Settings** dialog box, select the probe category and model in the **Selection** area.

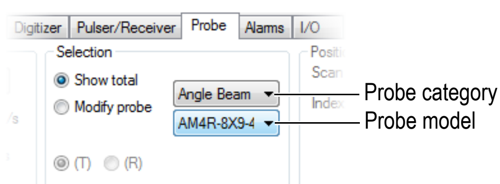


Figure 4-24 Selecting the probe in the UT Settings dialog box

4. Place the probe on the calibration block and position the probe over the two reflectors with known positions.
5. On an A-scan view:
   a) Select and configure the view to see the echoes of the two reflectors.
   b) Move the **Reference** cursor (red line) on the echo of the first reflector.
   c) Move the **Measurement** cursor on the echo of the second reflector.
6. In the **UT Settings** dialog box, in the **General** tab:

   a) Under **Time Base**, set **Mode** to **Half path**.
   b) Under **Auto Values**, click **Calibrate**.
7. In the **Time / Half Path** dialog box that appears (see Figure 4-25 on page 251):
   a) Under **What Do You want to Compute**, select **Compute Velocity and delay**.
   b) Set **Ref. cursor position** to the known position of the first reflector.
   c) Set **Meas. cursor position** to the known position of the second reflector.
   d) Click **OK**.
   TomoView calculates the ultrasonic velocity and wedge delay and sets the corresponding values in the **UT Settings** dialog box, in the **Probe** tab, under **Material and Interface**.

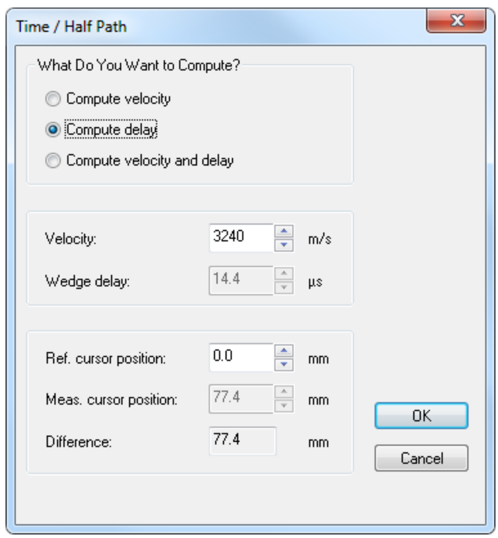
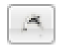


Figure 4-25 Calibrating with the Time / Half Path dialog box

4.5 Calibrating a TOFD Group in Setup Mode

Although TOFD groups are generally calibrated in Analysis mode, the following procedure explains how to calibrate in Setup mode. For more information on how to calibrate a TOFD group in Analysis mode, see section 7.6.2 on page 355.

To calibrate a TOFD group in setup mode

1. In the menu, select **View > Contents** and display the TOFD group A-scan and Side (B) views.
2. In the **UT Settings > General** dialog box, set **Mode** to **TOFD** in the **Time Base** group box.
3. In the **Component** toolbar, click  to open the **TOFD Manager** dialog box.
4. In the **TOFD Manager** dialog box, click **Calibration**.
5. In the **TOFD** dialog box:
   a) In the **What do you want compute?** group box, define what parameters you want to compute.
   b) In the **Scan axis** group box, define the scan axis as either **parallel**, or **perpendicular to beam**.
   c) In the **TOFD primary value** and **TOFD secondary value** group boxes, define the requested parameters.
   d) Click **Ok**.

NOTE

If you select the **Compute velocity and wedge delay** option button in the **What do you want compute?** group box of the **TOFD** dialog box, the measurement cursor has to be positioned on a second reference signal (for example, lateral wave).

The ultrasonic axis is now calibrated in setup mode.

4.6 Working with Overlay Drawings

An overlay is a graphical representation of weld geometry or of a part, which is superimposed over an ultrasonic data view. The overlay helps to visualize where defects are physically located relative to the weld or to the part geometry. TomoView provides predefined customizable overlays but you can also create weld and part overlays from drawing files (DXF).

For weld inspection applications, you can add a weld drawing as an overlay in the view:

To create a weld overlay

1. On the menu, select **Part & Overlay > Weld definition**.
   a) In the **Predefined Weld** dialog box, click the appropriate weld type (see Figure 4-26 on page 253).

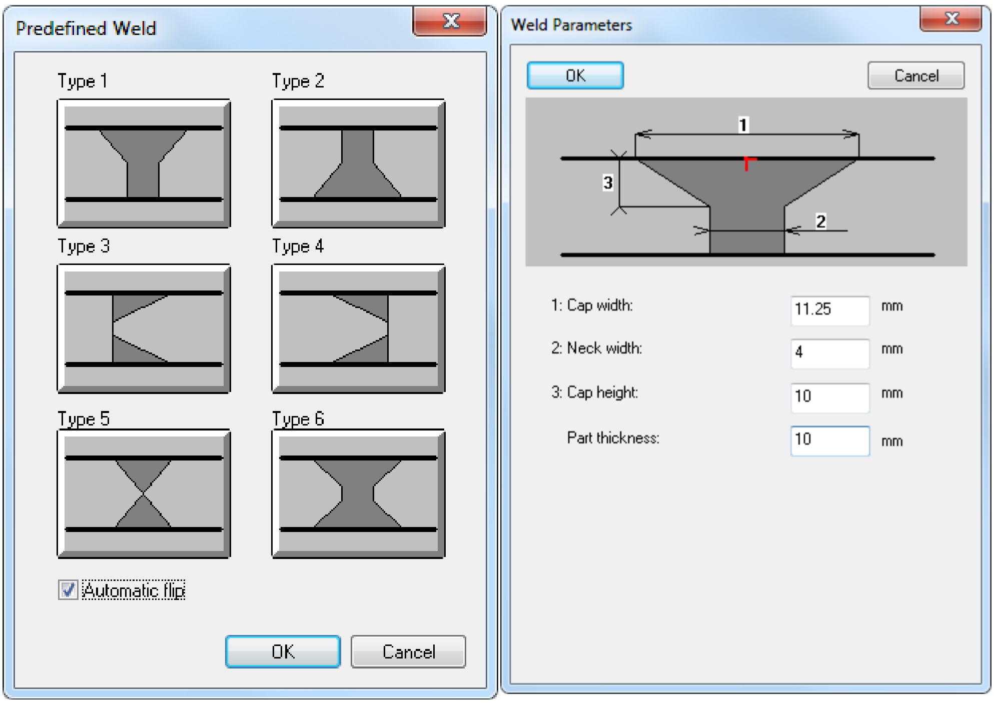


Figure 4-26 The Predefined Weld and Weld Parameters (for Type 1) dialog boxes

   b) In the **Weld Parameters** dialog box, type appropriate values to define your weld, and then click **OK**.
   c) Back in the **Predefined Weld** dialog box, select the **Automatic flip** check box, and then click **OK**.
   The weld overlay appears in the view (see Figure 4-27 on page 254).

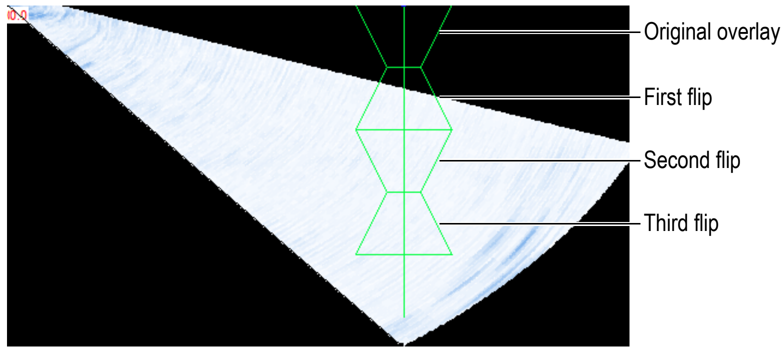


Figure 4-27 Example of a weld overlay with three flips

4.7 Working with the TCG Curve

The time-corrected gain (TCG) function operates by modifying the receiver gain during data acquisition to compensate for the attenuation of the ultrasonic wave in the material. The TCG curve defines gain values that are added to the group gain.

TomoView offers two methods to construct a TCG curve. For a UT conventional group (mono-element probe) or for a linear 0-degree phased array group, you can use the controls in the **TCG** tab of the **UT Settings** dialog box (see section 4.7.1 on page 254). For phased array group, use the Phased Array Calibration component (see section 4.7.3 on page 258).

In order to build a TCG curve, a calibration block is required with the same size reflectors at different depths. You can then display the TCG curve as a red line at the top of the corresponding A-scan view (see section 4.7.2 on page 257).

4.7.1 Constructing a TCG Curve for a Conventional UT Group

Use the controls in the **TCG** tab of the **UT Settings** dialog box to construct a TCG curve for a UT conventional group or for a 0-degree linear phased array group to which the same TCG curve is applied to all laws.

NOTE

The TCG function is not available for a group using a logarithmic amplifier on legacy acquisition units.

To define a TCG curve

1. Choose a layout that includes an A-scan view.
2. Enable the TCG function proceeding as follows:

   a) On the **Component** toolbar, click  (UT Settings).

   b) In the **UT Settings** dialog bar, click the **TCG** tab and then select the **Enable** check box.

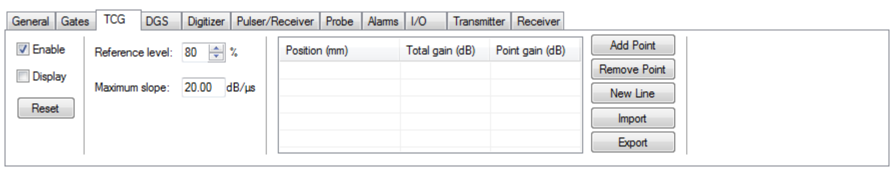


Figure 4-28 The TCG tab of the UT-Settings dialog box

3. Place the probe on a calibration block having same size reflectors at different known depths.
4. Position the probe to get well-defined echoes on the A-scan (see Figure 4-29 on page 256).

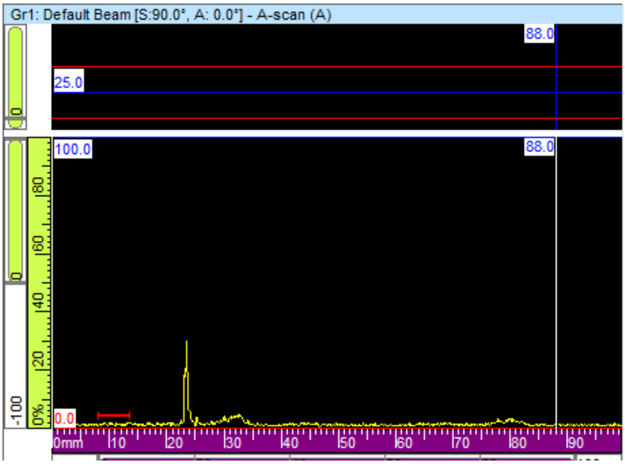


Figure 4-29 The A-scan before adding the first TCG point.

5. In the **UT Settings** dialog box, set the **Reference level** parameter to the amplitude level for the echo, expressed in percent of the full screen height. The 80 % default setting is suitable as a basic setting in most cases.
6. On the A-scan view, position the Reference and the Measurement cursors to the left and the right of the echo by double-clicking with the left and right mouse buttons.
7. Maximize the signal amplitude by moving the probe over the indication. Use the envelope tool to help you find the maximum signal.
8. In the **UT Settings** dialog box, click **Add Point** to add the chosen point to the TCG curve. When needed, click **New Line** to add an empty line.

   TomoView adjusts the gain to bring the peak of the selected echo to 80 % of the full screen height.

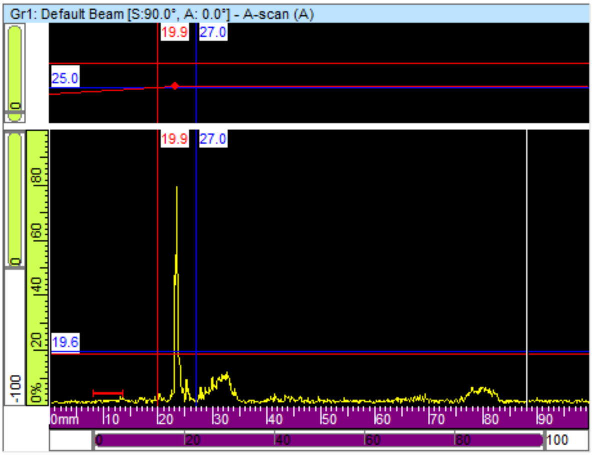


Figure 4-30 The A-scan after adding the first TCG point

9. Repeat steps 6 to 8 for each point you want to add to the TCG curve. A minimum of two points is required to define a functional TCG curve.

IMPORTANT

A **Point gain** with a negative value is not applied. Ensure to construct your TCG curve without negative point gains. Point 0 corresponding with the interface is not always the one with the highest amplitude.

4.7.2 Displaying or Hiding the TCG Curve

The TCG curve can be displayed or hidden using the following steps:

To display or hide the TCG curve

1. Select the A-scan view in which you want to display or hide the TCG curve.
2. On the **Component** toolbar, click  (UT Settings).
3. In the **UT Settings** dialog box, select the **TCG** tab.
4. Select or clear the **Display** check box to respectively display or hide the TCG curve.

The TCG curve appears as a red line at the top of the corresponding A-scan view.

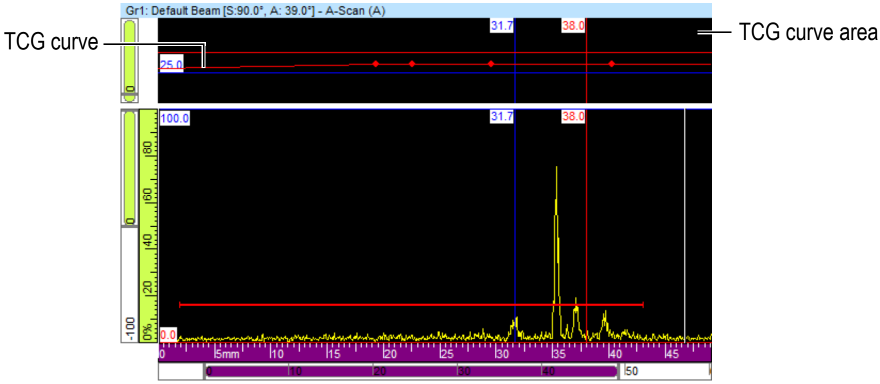


Figure 4-31 Example of an A-scan view showing the TCG curve

4.7.3 Constructing a TCG Curve for a Phased Array Group

Use the controls in the **TCG** tab of the **Phased Array Calibration** dialog box to construct a TCG curve for a phased array group. For angle beam inspections such as in weld applications, you need to use the following procedure to calculate the TCG gain independently for each beam.

To construct the phased array TCG curve

1. Enable the TCG function by proceeding as follows:

    a) On the **Component** toolbar, click  (UT Settings).

    b) In the **UT Settings** dialog bar, click the **TCG** tab, and then select the **Enable** check box.

2. On the **Component** toolbar, click **Phased Array Calibration** ().
3. In the **Phased Array Calibration** dialog box, click the **TCG** tab.
4. In TomoView, in a Side (B), End (D), or, Sector (S) view (depends on probe skew), position the reference (red) and measurement (blue) cursors respectively above and below the reflector indication (see Figure 4-18 on page 245).
5. Place the probe on the calibration block and perform a first scan on the calibration block above one reference reflector.
During the scan, TomoView draws the amplitude envelope (blue line) obtained

for each beam and constructs a curve (red line) representing the maximum amplitude of the envelope (see Figure 4-32 on page 259).

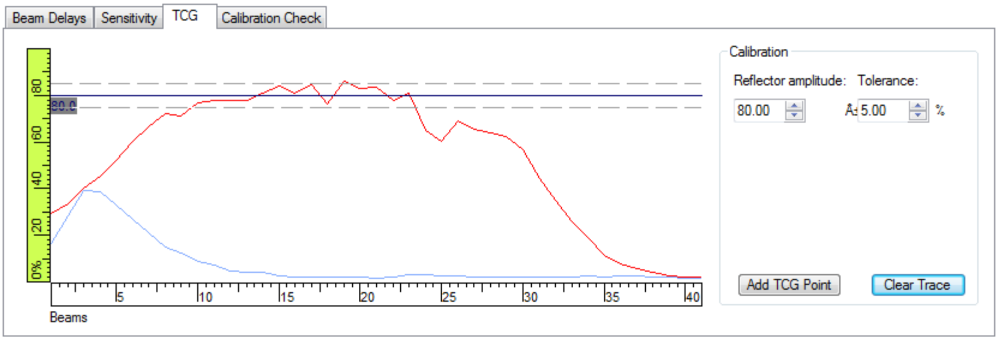


Figure 4-32 TCG curve construction example after a first scan

6. In the **Phased Array Calibration** dialog box:
   a) Set the **Reflector amplitude** parameter to the amplitude (%) at which you wish to see the reference reflector maximum amplitude appear.
   b) In the **Tolerance** parameter, enter the acceptable amplitude tolerance.
   c) Click **Clear Trace** to clear the data from the previous scan.
7. Perform a new scan on the calibration block.
8. To use this reflector to create a TCG curve point, click **Add TCG Point**. TomoView adjusts the gain for each beam to obtain the required reference amplitude for the depth corresponding to the inspected reflector.
9. Repeat steps 4 to 8 for each reference reflector used to construct the complete TCG curve.

TIP

You can review the TCG gain for each beam from **TCG** tab in the **UT Settings** dialog box.

4.7.4 Importing/Exporting a TCG Curve

The TCG curve points can be easily imported and exported directly from the TomoView interface using simple .csv (Comma Delimited) files using the following procedure.

To Import a TCG file into TomoView

1. Create the **.csv** file containing the different TCG points for a particular group.
   a) A line must be generated for every beam using the following structure where the **Positions** are defined in **mm** and the **Gains** are in **dB**. Every number is entered with a three digits decimal precision.

   Position 1, Gain 1; Position 2, Gain 2;....

   b) Comments can be added at the end of every line as long as the as preceded by a "#" symbol.

   c) The file must be saved or renamed as a **.csv** file to be interpreted correctly.

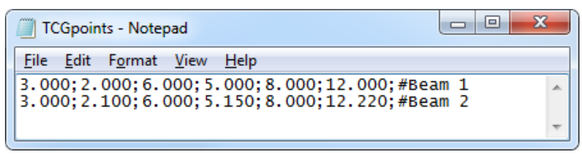


Figure 4-33 The TCG points import text

2. In the **TomoView Manager**, select the group on which you want to apply the TCG points.
3. In the **TCG** tab of the **UT Settings** dialog bar:
   a) Click **Import**.
   b) Select the **.csv** file containing the TCG points.
   c) Click **Open**.

NOTE

The TCG points position can be different from the ones defined in the **.csv** file if the position defined in the **.csv** file don't match the available TCG points position slots.

4.8 Working with Layouts

In TomoView, a layout is an organization of two or more views appearing in the document window. TomoView offers a set of ten layouts available from the **TomoView Manager** dialog box as a quick selection. (See Figure 3-15 on page 96.)

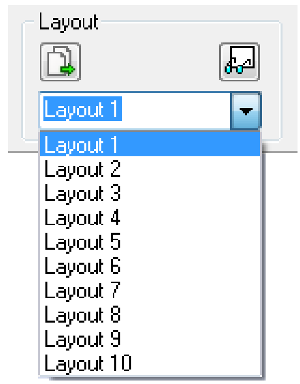


Figure 4-34 The set of ten layouts

4.8.1 Applying a Template Layout File

TomoView comes with a certain number of template layout files each containing up to ten layouts that can be selected to modify the current display. Additional layout files can also be created and defined as template layout files (see section 4.8.3 on page 263 for more information).

To apply a template layout

1. In the **TomoView Manager** dialog box, click  to select a Template layout file (see Figure 4-35 on page 262).

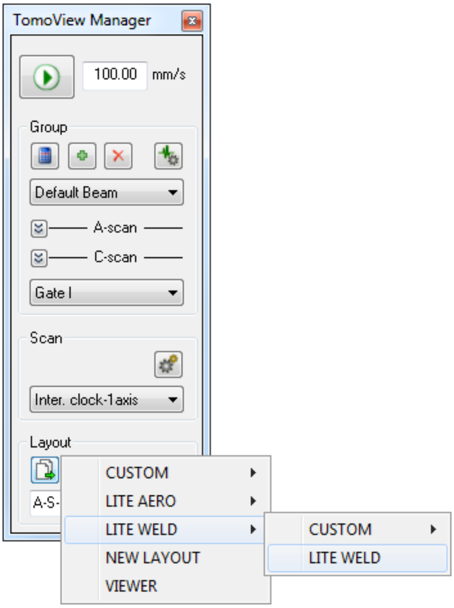


Figure 4-35 The Template Layout menu on the Control bar

2. From the list of available layouts, select the most appropriate. See Figure 4-36 on page 262.

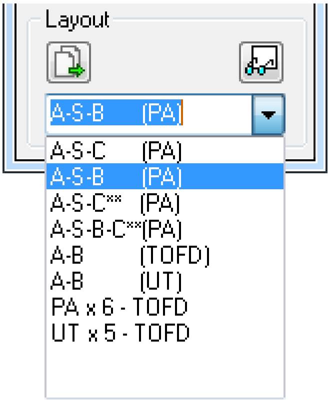


Figure 4-36 The list of available layouts

4.8.2 Displaying or Changing the View in a Pane

Once a layout has been selected, the views can be modified in order to customize the display.

To change the view in a pane

1. Activate the pane you want by clicking anywhere in it.
2. In the menu, click **View > Contents**, or click  on the **View** toolbar to open the **Contents** floating dialog box (see Figure 4-37 on page 263).

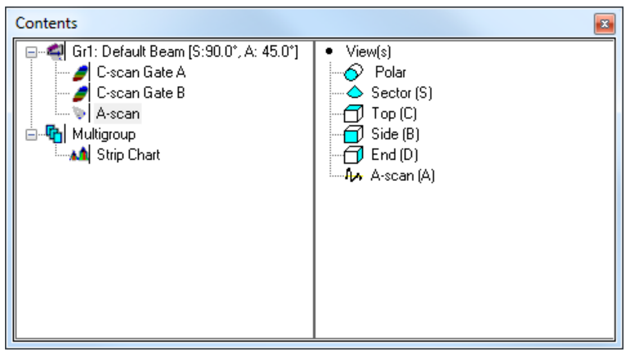


Figure 4-37 Example of the Contents dialog box

3. Double-click on the view to have in the active pane.
4. If required, activate another pane by clicking on it and select the data and the view to be displayed in this pane.

   The right section is updated to list the view types that are possible with the selected data type.

4.8.3 Saving Layout Files and Defining New Template Layouts

The list of template layouts described in section 4.8.1 on page 261 matches the content of the layout folder. The default layout folder is [Installation Folder]\TomoView210\Layouts Files (see Figure 4-38 on page 264). You can therefore customize the list of predefined layouts by adding, removing, or modifying the layout files and their subfolders.

CAUTION

Do not modify or delete the following layout files:
[Installation Folder]\TomoView210\Layout Files\Viewer.rst
[Installation Folder]\TomoView210\Layout Files\Lite Aero\Lite Aero.rst
[Installation Folder]\TomoView210\Layout Files\Lite Weld\Lite Weld.rst
Changing or deleting these files prevents the corresponding TomoView edition from starting (TomoViewer, Lite Aero, or Lite Weld). If this occurs, re-install TomoView to fix the problem.

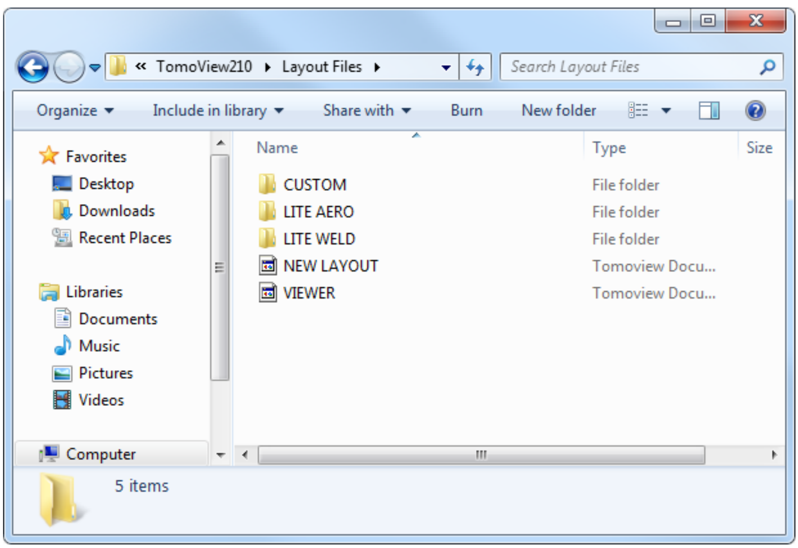


Figure 4-38 Default layout folder

To save a set of ten customized layouts:

1. In the menu, select **File > Save Layout as (*.rst)**.
2. In the **Save As** dialog box (see Figure 4-39 on page 265).
   a) Enter the **File Name** for the **.rst** file.
   b) Click **Save**.

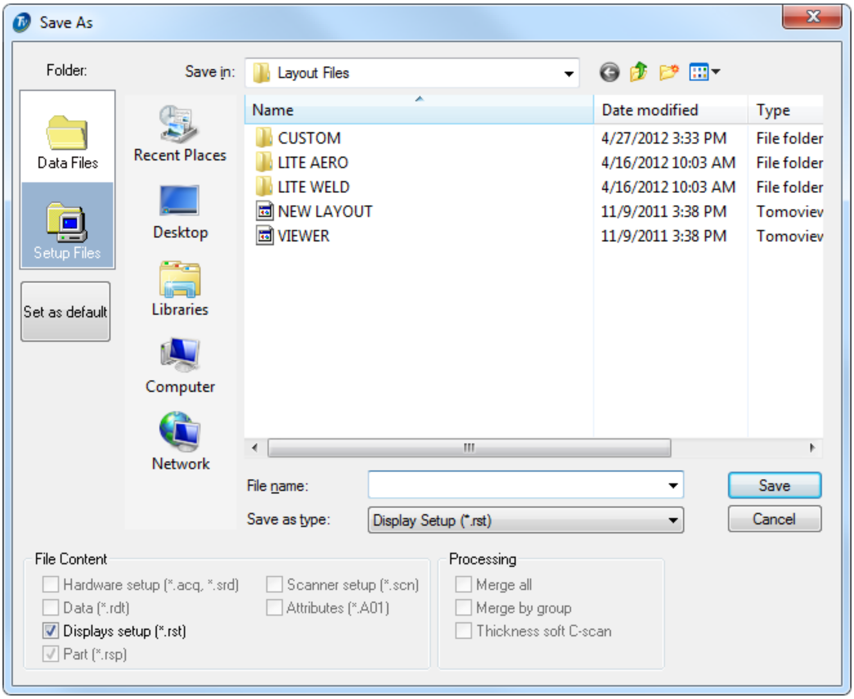


Figure 4-39 The Save As dialog box

You can also save up to five custom layout files available directly under the **Custom** section of the template layouts.

To save a set of ten customized layouts directly in the template layouts

1. In the menu, select **File > Save Custom Layout.**
2. In the **Save Custom Layout** dialog box (see Figure 4-40 on page 266):
   a) Select one of the five available custom layout slots.
   b) Define the name for the selected custom layout.
   c) Click **OK.**

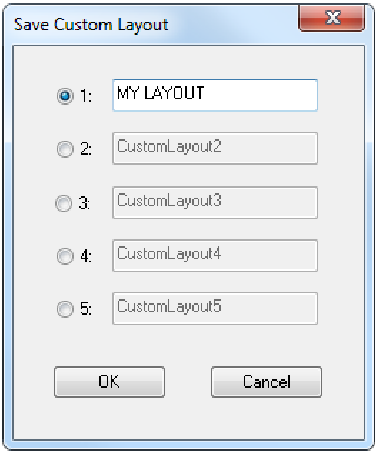


Figure 4-40 The Save Custom Layout dialog box

3. The new custom layout will be available in the same location as the other template layouts (see Figure 4-41 on page 266)

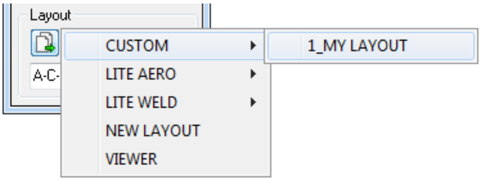


Figure 4-41 The Custom Layout saved templates

IMPORTANT

With the **Lite Aero** and **Lite Weld** editions, you can only load *Lite ready* predefined layout files, respectively saved in the folders:

[Installation Folder]\TomoView210\Layout Files\LITE AERO
[Installation Folder]\TomoView210\Layout Files\LITE WELD

4.9 Working with the DGS Curve

The distance gain size (DGS) is a method for sizing indications based upon a calculated DGS curve for a given transducer, material, and a 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 configuration only requires one reflector and the knowledge of the ultrasound propagation speed in the inspected material and in the wedge material.

The DGS function is more commonly used with conventional UT groups (see section 4.9.1 on page 267). You can however use the DGS function with a phased array group using special DGS qualified PA probes and specific group configuration (see section 4.9.3 on page 272).

4.9.1 Configuring the DGS for a Conventional UT Group

To configure the DGS for a conventional UT group

1. Create a conventional UT group (see section 4.2.2 on page 239).
2. In the **TomoView Manager**, ensure the newly created conventional UT group is selected.
3. On the **Component** toolbar, click  (UT Settings).
4. In the **UT Settings** dialog box, to ensure that the DGS function operates properly, perform the following tasks:
   a) In the **Pulser/Receiver** tab, under **Receiver**:
      (1) Ensure that **Rectification** is not set to **RF**.
      The DGS does not operate when **Rectification = RF**.
      (2) When working with µTomoscan and MultiScan MS5800 acquisition units, set **Scale type** to **Lin**. The DGS is not available for other scale types with these acquisition units.
   b) In the **Pulser/Receiver** tab, under **Pulser** and **Receiver**, set **Connector** to the appropriate value.
   In pulse-echo mode, they are the same.
   c) In the **TCG** tab, clear the **Enable** check box.
   The TCG must not be active when performing the DGS calibration.
5. Perform the calibration of the ultrasonic velocity and wedge delay (see "Calibrate" on page 122).

6. In the **UT Settings** dialog box, in the **Probe** tab:
   a) Under **Selection**, select **Modify probe** to make the other probe parameters editable.
   b) Under **Position**, adjust the **Scan offset** and the **Index offset** parameters to appropriate values.
   c) Under **Beam Orientation**, adjust the **Refracted angle** and the **Skew angle** parameters to appropriate values.
   d) Select the appropriate probe from the probe database.

4.9.2 Calibrating the DGS for a Conventional UT Group

To calibrate the DGS for a Conventional UT Group

1. On the **Component** toolbar, click  (UT Settings).
2. In the **UT Settings** dialog box, select the **DGS** tab (see Figure 4-42 on page 268) and then perform the following tasks:

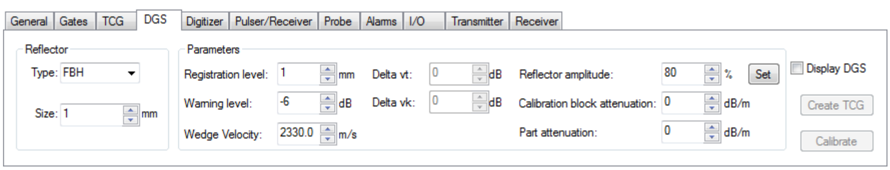


Figure 4-42 The DGS tab of the UT Settings dialog box

   a) Under **Reflector**, select the **Type** of reflector that you are using for the DGS calibration.
   The available choices are:
   - **Back Wall** (for linear 0-degree groups only)
   - **FBH**: flat bottom hole
   - **K1-IIW** (see IIW calibration block in Figure 4-43 on page 270)
   - **K2-DSC** (see DSC test block in Figure 4-43 on page 270)
   - **SDH**: side-drilled hole

NOTE

The list of available reflector types changes depending on the type of probe that you are using (contact or angle beam).

b) For **FBH** and **SDH** reflector types, under **Reflector**, set **Size** value to the diameter of the known reflector.

c) For K1-IIW and K2-DSC reflector types, on the DGS chart supplied with your probe, find the ΔV_K value, and then enter it in the **Delta vk** parameter.

d) If you know the energy transfer loss difference between the calibration block and the inspected part, enter the value in the **Delta vt** parameter.
Unlike changing the global gain, **Delta vt** only affects the signal gain, not the curve.

e) Enter the ultrasound velocity for the wedge material.

f) Under **Parameters**, enter the desired **Registration level** value.
The registration level is the reference reflector size. The maximum amplitude echo of a reflector of this size falls on the DGS curve, independently of the depth of the reflector.

NOTE

You can change the registration level, the gain, and the range without having to recalibrate the DGS.

g) Enter the desired **Warning level** value.

h) If you know the attenuation value for the calibration block, enter the value in the **Calibration block attenuation** parameter.

i) If you know the attenuation value for the part material, enter the value in the **Part attenuation** parameter.

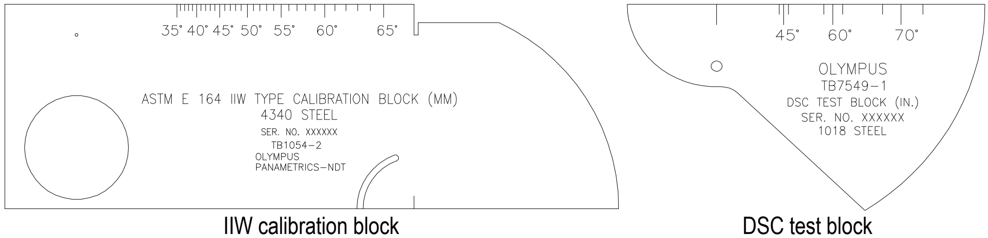
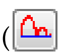


Figure 4-43 The IIW and DSC calibration blocks

3. Move the probe on the calibration block to the position of the maximum amplitude echo (see Figure 4-44 on page 270).

TIP

Use the envelope tool () from the View toolbar to guide you to find the maximum amplitude echo.

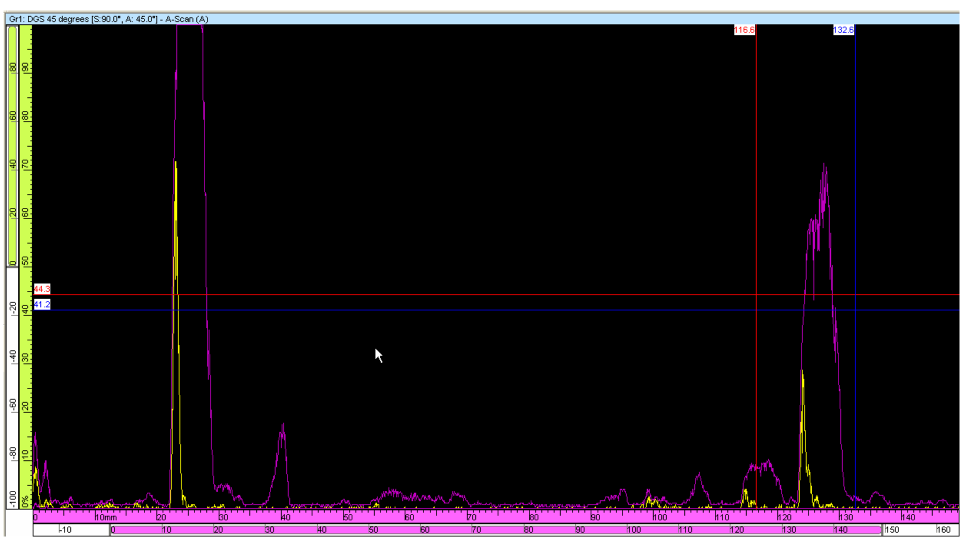


Figure 4-44 Finding the maximum amplitude using the Envelope tool

4. Move the Reference (red line) and Measurement (blue line) cursors on either side of the reflector echo used for the calibration.
5. Once the signal is maximized, in the **UT Settings** dialog box, in the **DGS** tab:
   a) Click **Set**.
   The amplitude of the reflector echo is modified to match the value of the **Reflector amplitude** parameter.
   b) Click **Calibrate**.
   c) Select the **Display DGS** check box to view the DGS curve on the A-scan (see Figure 4-45 on page 271).
   d) You can export the DGS curve to a TCG by clicking **Create TCG**.
   TomoView transfers the DGS parameters to the TCG and the DGS curve disappears.

NOTE

Once you enable the TCG, you cannot change DGS parameters.

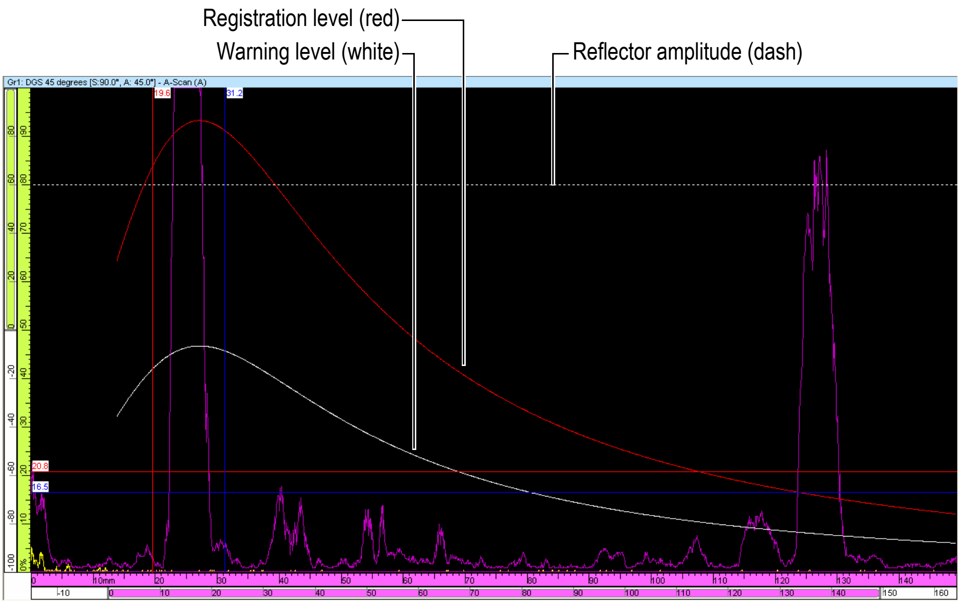


Figure 4-45 Example of the DGS registration and warning level curves

4.9.3 Configuring the DGS for a Phased Array Group

You can use the DGS function with a PA group as long as the group contains only one beam.

NOTE

You cannot configure a phased array group using a non-phased array MultiScan MS5800 acquisition unit.

To configure a phased array group for DGS

1. Create a phased array group as described in section 4.2.1 on page 231.
2. In the TomoView Manager, click the Advanced Calculator button ( ) to return in the Advanced Calculator and perform the following adjustments (see Figure 4-46 on page 273):
   a) Select the **1-D Linear array** tab.
   b) Under **Scan Type**, select **Static** to limit the number of beams to one.
   c) Under **Beam Angles Selection**, for the **Refracted Angle** parameter, set the **Start** value to the desired angle.
   d) Under **Focal Points Selection**, since the DGS does not support beams, set **Emission focus position** to a large value such as **1000**.
   e) Under **Probe**, select the **Integrated Wedge** probe category, and then select one of the DGS probes (**4L16-DGS1** or **2L8-DGS1**).
   f) At the bottom of the Advanced Calculator window, click **Replace** to return the calculated focal law parameters to TomoView.
3. Perform the necessary calibrations (see section 4.3 on page 245).

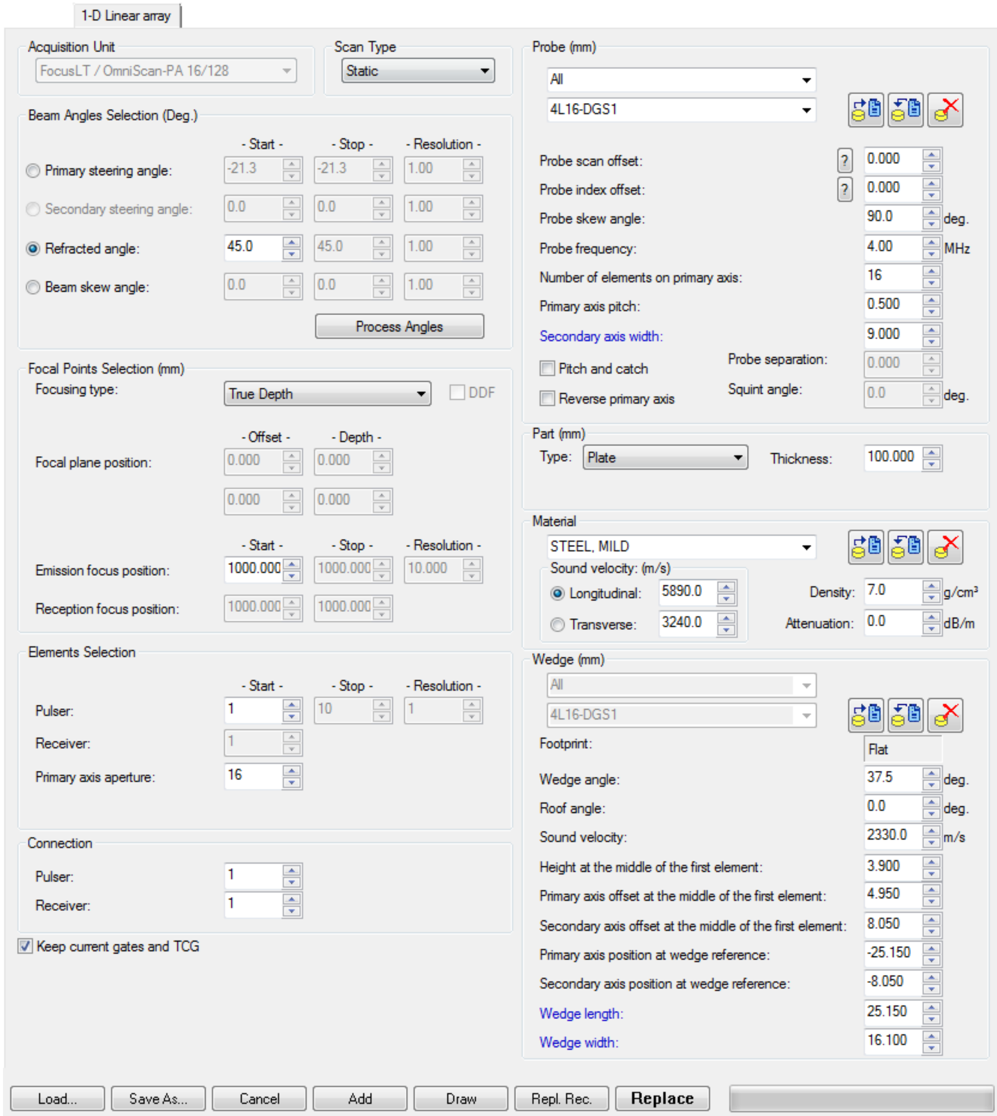


Figure 4-46 An example of the Advanced Calculator 1-D Linear array tab configured for the DGS

4.9.4 Displaying DGS Readings

TomoView features special DGS readings that can be displayed only when the Expert mode is enabled. See section 3.8 on page 114 for more information on readings and information groups.

To display DGS readings

1. To make the DGS readings available, activate the Expert mode:
   a) On the menu select **File > Preferences**.
   b) In the **Preferences** dialog box, select the **General Settings** tab.
   c) Under **Interface**, select the **Expert mode** check box.
2. On the menu select **View > Readings > Edit Groups**.
3. In the **Information Groups** dialog box (see Figure 4-47 on page 275):
   a) Expand the **Group[*n*]** in which you want to add DGS readings.
   b) Expand the **DGS** section.
   c) Select the check box in front of the desired DGS readings.
   d) Click **OK**.

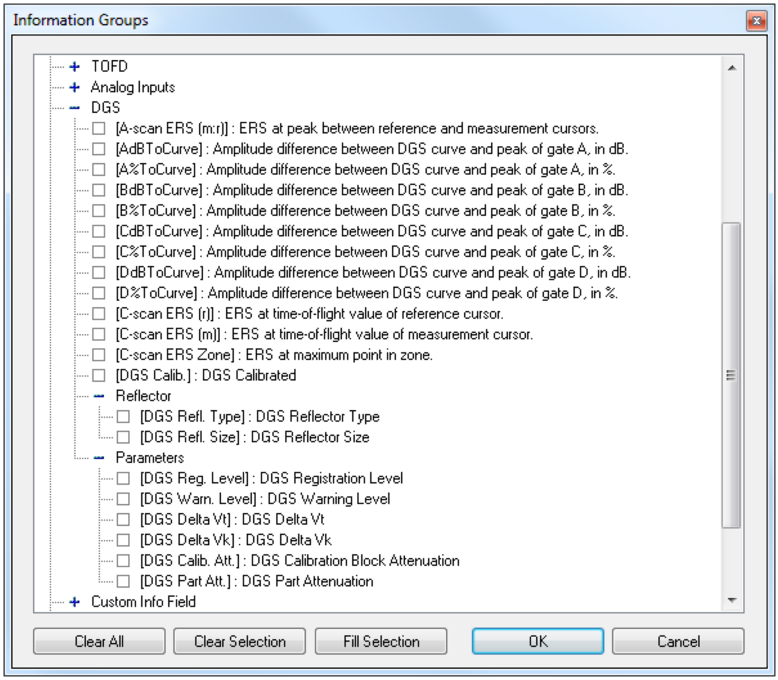


Figure 4-47 The DGS readings in the Information Groups dialog box

4. To display the readings on the desired view:
   a) Right-click on the title bar of the view.
   b) In the contextual menu, select or clear **Show Info Group**.

4.9.5 Changing the DGS Curve Colors

You can also change the color of the DGS curves from the **Preferences** dialog box.

To change the DGS curve colors

1. On the menu, select **File > Preferences**.
2. In the **Preferences** dialog box, select the **View Colors** tab.
3. Under **DGS Curves**, click **Standard** or **Warning** to open the **Color** dialog box respectively for the registration level and the warning level DGS curves.
4. In the **Color** dialog box, select the desired color for the curve, and then click **OK**.

5. Back in the **Preferences** dialog box, click **OK**.

4.10 Working with the Readings

TomoView computes reading values for various parameters for analyzing ultrasonic data. Readings are calculated using cursor, zone, acquisition, or segment (3-D Cursor) parameters and can be displayed at the top of a view (see Figure 4-50 on page 279).

4.10.1 Reading Management

You can define four groups of readings using the readings **Information Groups** dialog box (see Figure 4-48 on page 277). You access the **Information Groups** dialog box by double-clicking in the reading zone that appears at the top of a view or by selecting **View > Readings > Edit Groups** on the menu.

The most frequently used readings are organized by categories under **Favorite Readings** in the **Information Groups** dialog box. Each category has a limited number of readings with short names similar in concept to those of the OmniScan software. The categories and lists of available readings are the same for each of the four groups and for all view types.

More readings are available from the **Information Groups** dialog box when the export mode is activated (see section 3.16 on page 220 for details).

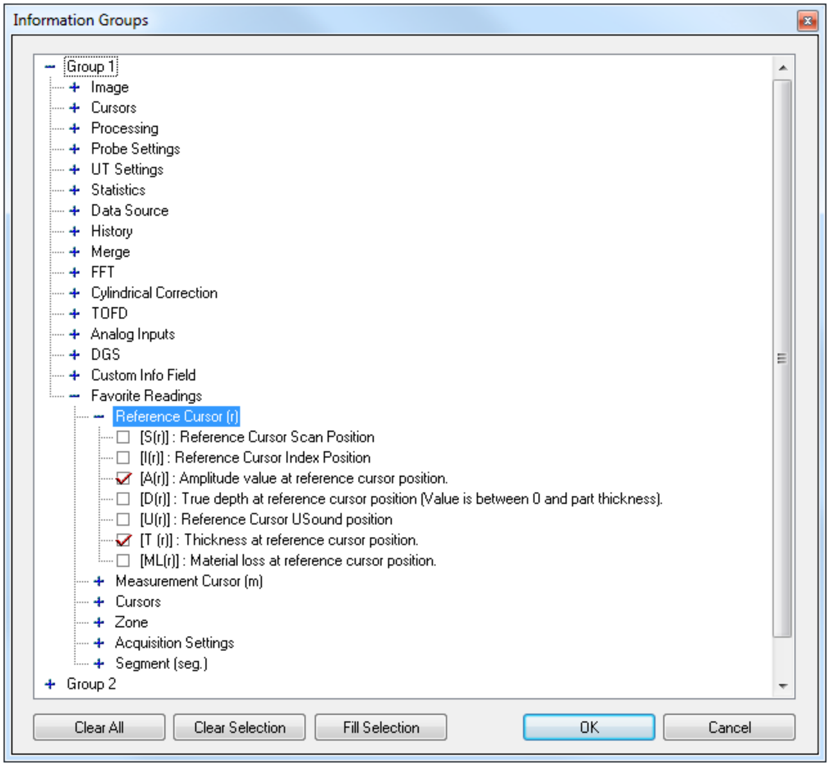


Figure 4-48 The Favorite Readings categories in the Information Groups dialog box

The readings configuration is saved in an .rst file that is included by default in the .acq file (see section 3.18 on page 221 for details).

4.10.2 Reading Examples

The following readings, illustrated in Figure 4-49 on page 278, provide useful numerical values for the indication amplitude and position within the inspected part:

A(r)

Is the amplitude in a percentage at the reference cursor position.

D(r)

Is the true indication depth (always between zero and the part thickness) at the reference cursor position.

T(r)

Is the thickness at the reference cursor position.

U(r)

Is the position of the reference cursor on the ultrasound axis (expressed in time of flight [µs], half path [mm] or true depth [mm]).

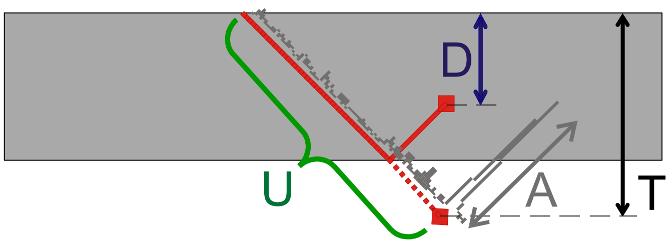


Figure 4-49 Illustration of the A, D, T, and U readings

The following readings provide useful information for corrosion applications:

ML(r)

Material loss (%) at the reference cursor position

T(Zmin)

Minimum thickness inside the zone

S(ZMin)

Position of the minimum thickness inside the zone on the scan axis

I(Zmin)

Position of the minimum thickness inside the zone on the index axis

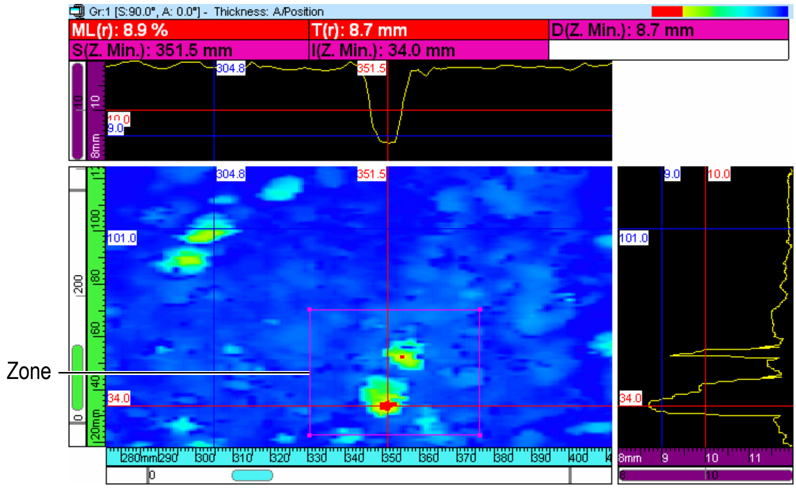


Figure 4-50 Corrosion readings example

When you position the pointer over a reading, a tooltip appears, providing a useful definition of the reading (see Figure 4-51 on page 279).

Figure 4-51 Example of a reading tooltip

You can display all readings on all views. However, a value appears only when the measurement can be computed in the view. For example, with an A-scan view, the amplitude value can be computed and displayed but not a zone value (see Figure 4-52 on page 279).

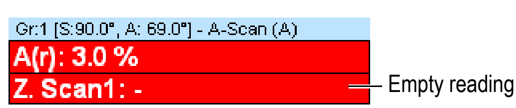


Figure 4-52 Example of an empty reading

4.10.3 Hiding or Showing Readings at the Top of a View

You can quickly hide or show readings configured to appear at the top of a view.

To hide or show a specific readings group at the top of a view

1. Select the view you want to modify.
2. In the View toolbar, select  to hide or show a specific readings group.

To hide or show all the readings at the top of a view

1. Right-click on the title bar of the view.
2. In the contextual menu, select or clear **Show Info Group**.

4.10.4 Customizing the Reading Color and Font

You can customize the background color and the font of readings.

To customize reading colors and the font

1. On the menu, select **File > Preferences** to customize reading background colors.
2. In the **Preferences** dialog box, select the **Tool Colors** tab (see Figure 4-53 on page 280).

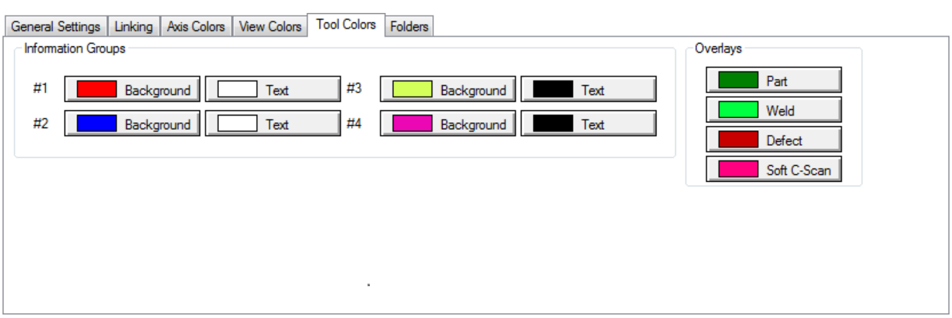


Figure 4-53 The Tool Colors tab of the Preferences dialog box

3. Under the **Information Groups** area, click the button corresponding to the element (**Background** or **Text**) and the reading group (#**1,** #**2,** #**3,** or #**4**) that you want to modify.
4. In the **Color** dialog box, click the desired color, and then click **OK** (see Figure 4-54 on page 281).

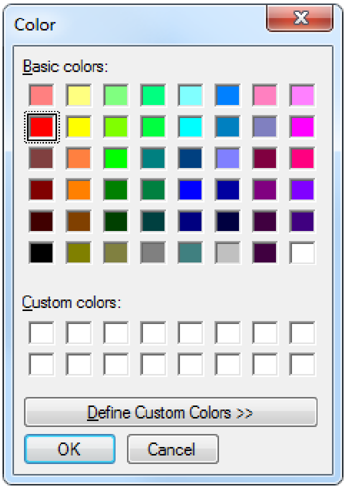


Figure 4-54 The Color dialog box

5. Repeat step 3 for each element for which you want to change the color.
6. In the **Preferences** dialog box, click **OK**.
7. On the menu, select **View > Readings > Properties** to customize the reading font.
8. In the **View Information Properties** dialog box (see Figure 4-55 on page 282), select the desired **Font** and **Size** for the readings, and then click **OK**.

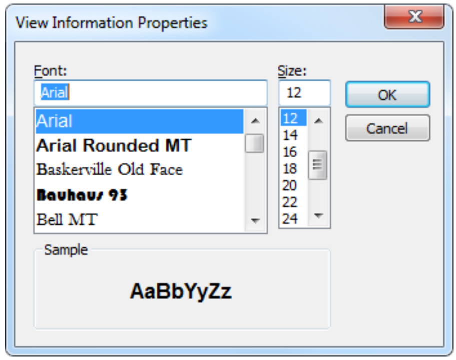


Figure 4-55 The View Information Properties dialog box

4.11 Working with Gates

The following steps describe how to configure the gates. This procedure must be independently performed for every group. For reference information on the **Gate** tab, see section 3.10.2 on page 123.

To define gates

1. On the **Component** toolbar, click  (UT Settings).
2. If not already done, display an A-scan view on the active pane.
3. On the UT Settings dialog bar, click the Gate tab and perform the following:
   a) Place the reference and measurement cursors on the ultrasonic axis so as to determine the starting position and length of the gate.
   b) Place the reference cursor on the amplitude axis by double-clicking with the left and right mouse buttons so as to determine the detection threshold level of gate 1.
   c) Click **Set Gate** button.
   d) Select the **POS & AMP** check boxes to enable the creation of a data group thereby enabling recording of the position and amplitude of the C-scan data.
   e) For Gate I, select the **POS B - POS A** check box to enable the creation of a dat group monitoring the position difference between Gates A and B.

4.11.1 Gate Adjustments

You can adjust the position and the width of a gate. In the A-scan, this is done by dragging-and-dropping a part of the gate line. The mouse pointer changes depending on where you click on the gate line, and indicates the task that you can perform (see Table 13 on page 283 for details).

Table 13 Moving and resizing a gate using the mouse

Task	Pointer	Action
To resize a gate		Drag-and-drop the end of the gate line
To move a gate		Drag-and-drop the middle part of the gate line
To move a gate only vertically		Press and hold the SHIFT key, and then drag-and-drop the middle part of the gate line
To move a gate only horizontally		Press and hold the CTRL key, and then drag-and-drop the middle part of the gate line

You can adjust basic gate parameters using the controls in the **TomoView Manager** dialog box under **Group** (see Figure 4-56 on page 284).

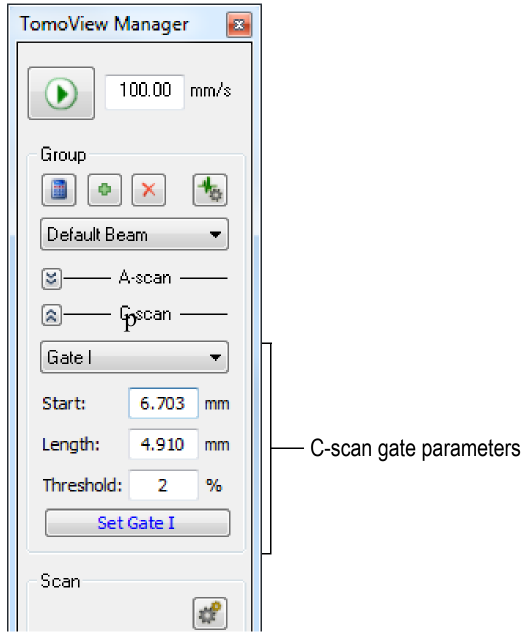


Figure 4-56 The gate parameters in the TomoView Manager dialog box

You can precisely configure the gates by pressing  on the **TomoView Manager**, and then use the parameters in the **Gates** tab of the **UT Settings** dialog box (see Figure 4-57 on page 284).

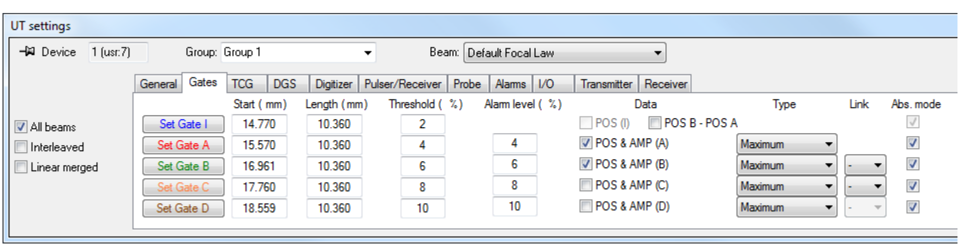


Figure 4-57 The Gates tab of the UT Settings dialog box

4.11.2 About Gate Synchronization

Synchronizing gates means that the start position of a gate depends on the start position of another gate. For example, when gate B has a start position of 10 mm and is synchronized with gate A, gate B starts 10 mm on the right of the start of gate A.

Synchronizing gates is useful for example in an immersed inspection where the distance between the probe and the inspected part fluctuates in time. Using gate I to detect the interface echo, and then synchronizing gate A on gate I ensures that gate A always captures echoes from the same position range in the immersed inspected part.

You can synchronize the position of a gate only with the position of the previous gate. For example, gate A can only be synchronized with gate I, gate B can only be synchronized with gate A and so forth.

To synchronize a gate on another gate

1. To synchronize gate A on gate I, set **Synchro**. to **Echo** in the **Digitizer** section of the **UT Settings** dialog bar.
2. To synchronize other gates, set **Link** to the desired value in the **Gates** section of the **UT Settings** dialog bar.

When TomoView acquires data with a FOCUS LT, you can also specify a negative start position for gate B that is synchronized on gate A. This allows you to create an anti-causal or pre-synchronization detection. This feature is useful when you need to detect a week echo appearing just before a strong echo. Pre-synchronization is possible because the FOCUS LT can generate up to 10 μs of signal prior to a gate. Note that while more than one pair of gate can use pre-synchronization, you cannot synchronize a gate to a synchronized gate that has a negative start position.

4.11.3 Gates in the Analysis Mode

You can also adjust the gate position and size in Analysis mode. The modified data is saved in a companion file (.A01). You can easily come back to the original gate parameter values by right-clicking on the title bar of the view, and then selecting **Restore Initial Gates** on the contextual menu.

NOTE

In Analysis mode, when you move a gate on an A-scan or S-scan view while the Ultrasound axis is in true depth, the new gate position is calculated in true depth. However, when you move the a gate on an A-scan view while the Ultrasound axis is not in true depth, the new gate position is calculated in sound path.

In Analysis mode, when programmed in half-path mode, the gated zone on the S-scan is true for the current beam. The position of the gate limits are automatically adjusted on the S-scan when you change the current beam (see Figure 4-58 on page 286).

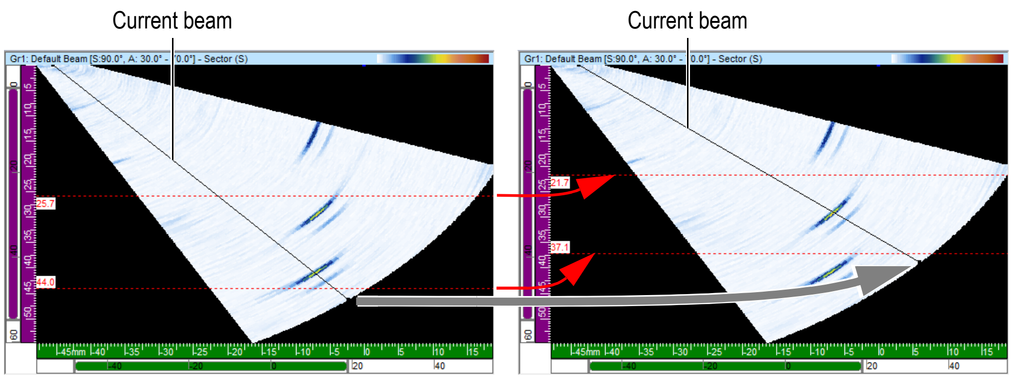


Figure 4-58 S-scan gate positions automatically adjusted when the current beam changes when programmed in half-path mode

4.11.4 Gates and Amplitude C-Scans

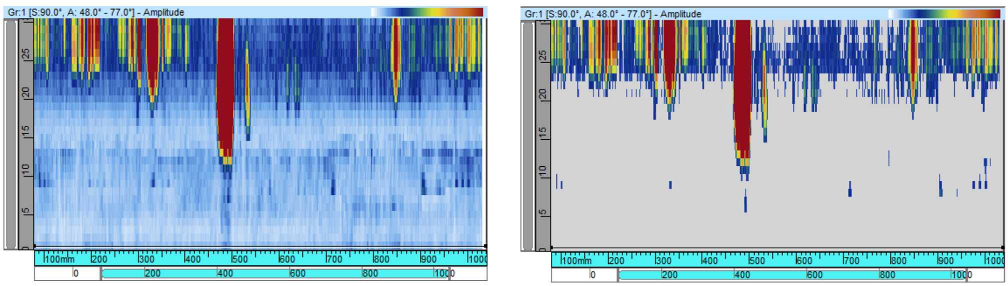
On an amplitude C-scan, TomoView normally does not show signal amplitudes that are below the gate level as the OmniScan software does. However, you can configure TomoView to always record the amplitude in acquisition even when the signal is below the gate level using the following procedure.

To configure TomoView to show data below the gate level in acquisition

1. Select **File > Preferences**.
2. In the **Preferences** dialog box, select the **General Settings** tab.

3. Under **C-Scan in Analysis**, select the **Always record amplitude in acquisition** check box.

In Analysis mode, after moving the gate, you can however configure TomoView to show this signal (see Figure 4-59 on page 287).



Showing data below gate — Not showing data below gate

Figure 4-59 Example of a C-scan showing and not showing the data below the gate

To configure TomoView to show data below the gate level in analysis

1. On the menu, select **File > Preferences**.
2. In the **Preferences** dialog box, select the **General Settings** tab.
3. Under **C-Scan in Analysis**, select the **Always record amplitude in analysis** check box.

4.11.5 Gate Usage Example

A water immersion inspection is a good example to illustrate the usage of multiple gates (see Figure 4-60 on page 288). Position and size gate I over the area where you expect to find the echo from the part interface. Position gate A to cover the path inside the inspected part without detecting the entry and back wall echoes. Finally, position and size gate B over the area where you expect to find the back-wall echo. In an immersed inspection, the distance between the probe and the part often fluctuates. To compensate for this fluctuation, the start position of gate A and gate B can be set to automatically change according to the position of the echo detected in gate I. This ensures that both gates constantly cover appropriate areas.

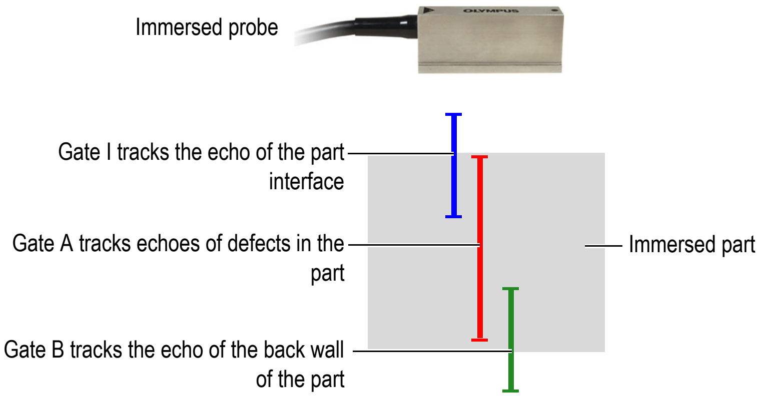


Figure 4-60 Example of gate usage in a water immersed inspection

4.12 Working with the Alarms

The following procedure describes how to define the alarms. For reference information on the **Alarms** tab of the **UT Settings** dialog box, see section 3.10.8 on page 141.

To define the alarms

1. On the **Component** toolbar, click  (UT Settings).
2. In the **Alarms** tab of the **UT Settings** dialog bar:
   a) In the **Output line** box, select the alarm output line.
   b) Define the alarm condition by selecting the appropriate options under the **Conditions** group box.
   c) Repeat these steps for each alarm condition.

   A set of indicators appears in the status bar representing the state of each alarm:
   

   .

4.13 Working with the Inputs/Outputs

The following procedure describes how to set the **Analog Inputs,** while the **General**

Outputs and **Digital out** can be set by simply selecting the corresponding check box. For reference information on the **I/O** tab of the **UT Settings** dialog box, see section 3.10.9 on page 142.

To set the Analog Inputs

1. On the **Component** toolbar, click (UT Settings).
2. In the **I/O** tab of the **UT Settings** dialog bar:
   a) Select the Analog 1 check box to enable the first analog input line.
   b) Define the associated Multiplier and Offset values by double-clicking on the current value.
   c) Edit, if necessary, the Measure type and Unit parameters.

4.14 Working with the Firing Sequencer

The **Firing Sequencer** module is used to modify the order in which the ultrasonic beams are fired. For some applications, the modification of the firing sequence can help reduce the effects of ghost echoes due to high recurrence.

For a phased array setup with multiple focal laws and channels, the firing repetition (recurrence) is very important to maximize the scanning speed without the presence of interference echoes. This can be critical for immersion techniques with the presence of signals coming from the water surface.

To activate the Firing Sequencer

1. On the **Component** toolbar, click (UT Settings).
2. In the **UT Settings** dialog bar, select **Interleaved**.
3. On the **Component** toolbar, click the **Fire Sequencing** button ().

When clicking on , the **Firing Sequencer** dialog bar will appear, which contains the list of all beams included in the different groups.

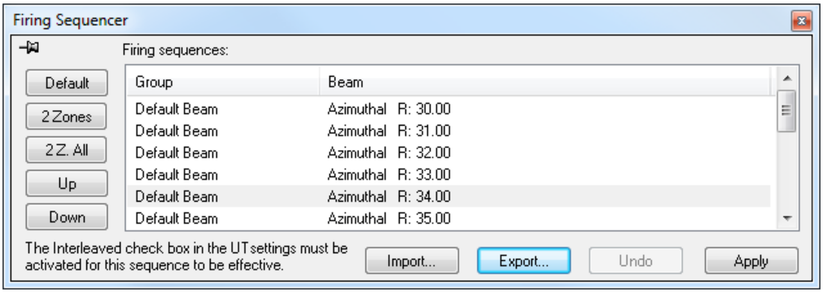


Figure 4-61 The Firing Sequencer (Interleave) dialog bar

The following describes the buttons of the **Firing Sequencer** which can be used to modify the firing order of the different beams.

Default

Brings the firing sequence order back to the default sequence (Group 1 - Beam 1, Group 1 -Beam 2, …, Group 2 - Beam 1, Group 2 - Beam 2, and so on), This means that the first generated beam is the first generated beam to be fired.

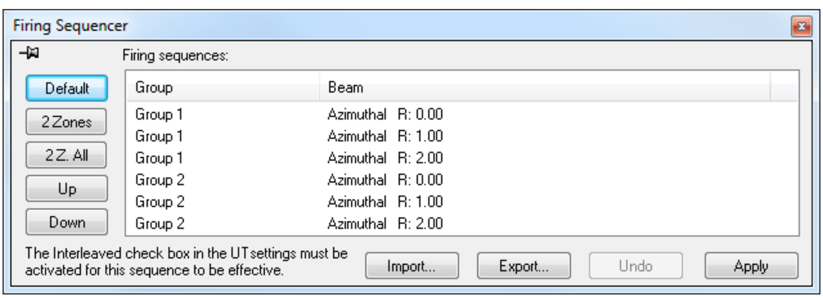


Figure 4-62 The Default dialog button

2 Zones

Creates a firing sequence with interlacing pairs of beams within each group (see example in Figure 4-63 on page 291).

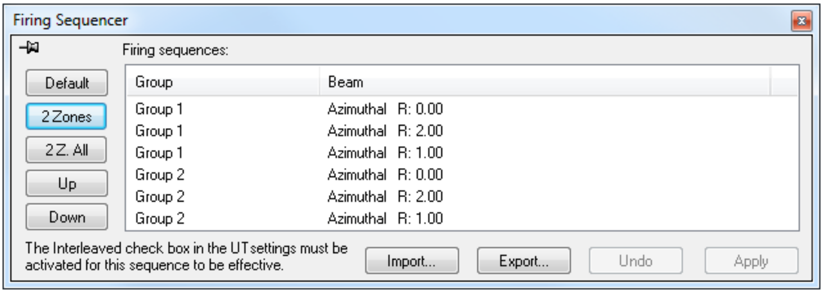


Figure 4-63 Example of firing sequence with 2 Zones interlacing pairs

2 Z. All

Creates a firing sequence with interlacing pairs of beams within all groups (see example in Figure 4-64 on page 291).

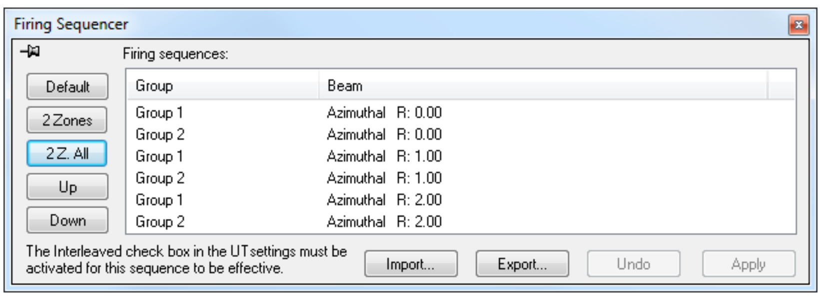


Figure 4-64 Example of firing sequence with 2 Z. All interlacing pairs

Up

Moves the selected beam or group of beams up in the **Firing sequences** list.

Down

Moves the selected beam or group of beams down in the **Firing sequences** list.

Import

Allows you to import a **.cfs** file containing a firing sequence from a previously exported configuration.

Export

Allows you to save the current firing sequence to a **.cfs** file.

4.15 Working with the Conditional A-scan

The **Conditional A-scan** allows the A-scans to be recorded only when an alarm was triggered. This allows the A-scans to be kept only in the areas where this information is important and therefore allows the user to inspect much larger areas in a single inspection.

To Set the Conditional A-scan

1. Go to the **Digitizer** section of the **UT Settings** panel and check the **Conditional** checkbox. See Figure 4-65 on page 292.

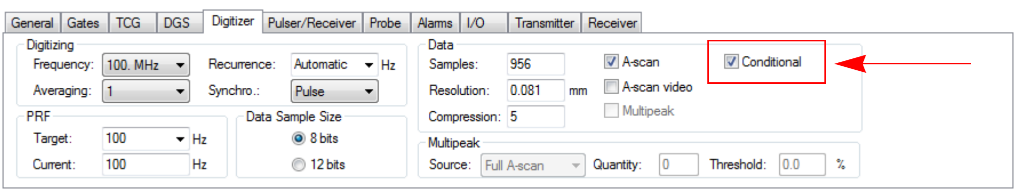


Figure 4-65 The Digitizer tab

2. In the **Alarms** section of the **UT Settings** dialog bar, configure the alarms that will trigger the A-scan to be recorded. Three different alarms can be configured, all of which will have the same effect. See Figure 4-66 on page 292.

Figure 4-66 The Alarms tab

3. Click  to start an acquisition.

   The A-scans will only be recorded in the areas where an alarm was triggered as can be seen on the following figures. See Figure 4-67 on page 293 and Figure 4-68 on page 293.

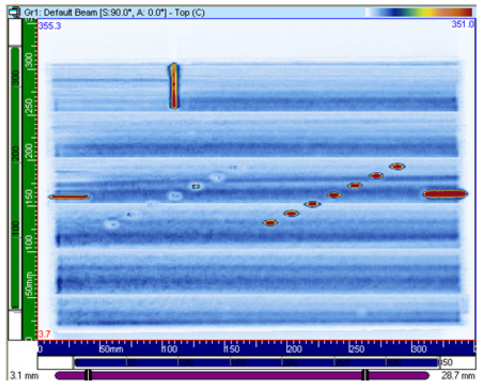


Figure 4-67 Acquisition with full A-scan recording

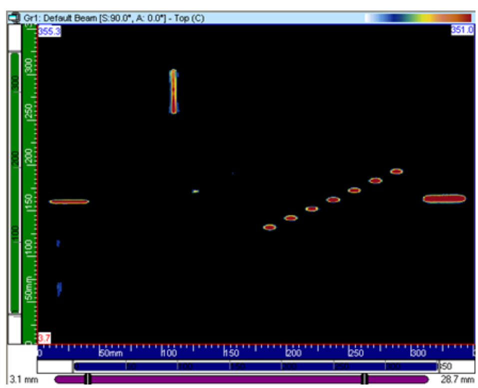


Figure 4-68 Acquisition with conditional A-scan recording

5. Performing Acquisitions

To simplify sequence programming, TomoView offers predefined sequence types with typical values. You have only to enter the inspected area's dimensions and set the parameters of your mechanical system. You can also modify the predefined sequences as needed, or define custom sequences.

5.1 Working with the Different Scan Types

TomoView has several types of available scans. They are listed and described in the following sections.

The different scan types available in TomoView are: **One-line scan, Free running, Bidirectional, Unidirectional, Helicoidal, Angular**, and **Custom**.

These different scan types are explained in the following sections.

5.1.1 One-Line Scan Sequence

The **One-line scan** sequence type identifies a linear-scan sequence. One position encoder is used to determine the position during the acquisition.

The linear scan (see Figure 5-1 on page 296) is unidimensional and proceeds along a linear path. The only settings that must be provided are the limits along the scan axis and the spacing between acquisitions.

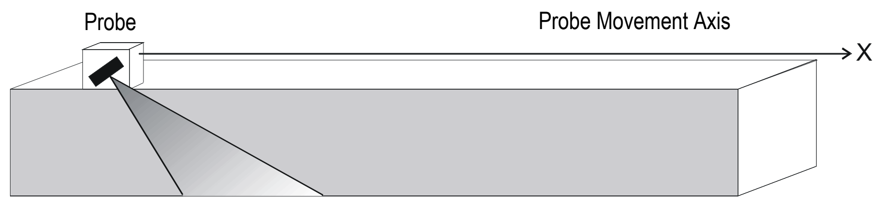


Figure 5-1 Linear scan

Operation mode

The **One-line scan** inspection sequence operates as follows:

1. The scanner proceeds to the position set in the **Scan: Start** box of the **Sequence** tab.
2. The scanner then moves on the scan axis to the position set in the **Scan: Stop** box, while performing data acquisition.
3. Data acquisition is performed at every interval set in the **Scan: Resolution** box.
4. The sequence is finished when the scanner has reached the position set in the **Scan: Stop** box.

Description of the Sequence tab

When you select **One-line scan** for the sequence inspection type, the **Sequence** tab contains the set of options shown in Figure 5-2 on page 296.

Figure 5-2 The Sequence tab for the One-line scan sequence

The **Sequence** tab for a **One-line scan** sequence contains the same options as for a **Bidirectional** sequence, except that it does not include the settings for the index axis. For a description of these options, please refer to section 5.1.1 on page 295.

5.1.2 Free Running Sequence

The **Free Running** sequence identifies an inspection sequence where data acquisition is performed at the rate specified in the **Acquisition rate** box of the **Digitizer** tab (**UT Settings** dialog bar). For TomoView, the data is recorded at only one position, at the origin of the scan and index axes.

Description of the Sequence tab

When you select **Free Running** for the sequence inspection type, the **Sequence** tab contains no options or parameters (see Figure 5-3 on page 297).

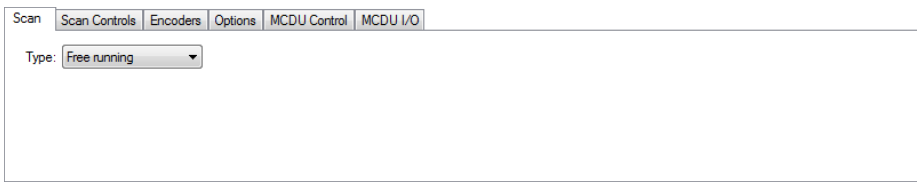


Figure 5-3 The Sequence tab for Free Running sequence

5.1.3 Bidirectional Scan

A **Bidirectional** scan type is a bidimensional surface scan (also known as a raster scan) sequence in which two encoders are used to determine the position on the scan and index axes during the acquisition.

A surface scan uses two axes: (1) the scan axis, which is the mechanical axis of the scanning lines (2) the index axis, which is the mechanical axis of movement between the scanning lines. At the end of each scan along the scan axis, an increment is added to the position along the index axis. Data acquisition for bidirectional sequences is carried out in both the forward and backward directions along the scan axis, as shown in Figure 5-4 on page 298.

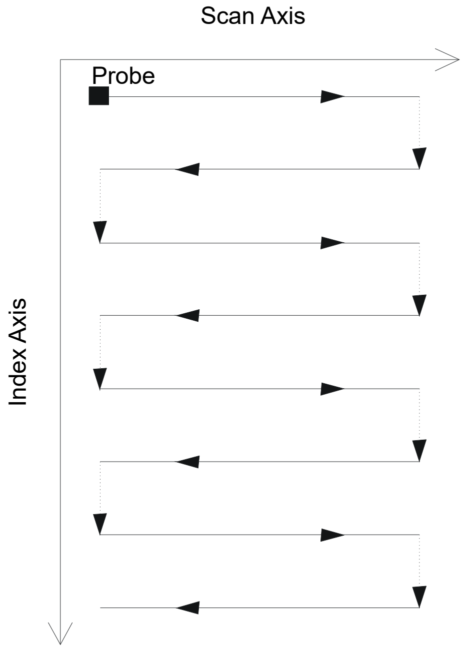


Figure 5-4 Bidirectional surface scan

The user must supply the limits of the inspection surface as well as the spacing between acquisitions. Figure 5-5 on page 299 shows the scan-axis and index-axis parameters in the reference system of the scanning mechanism.

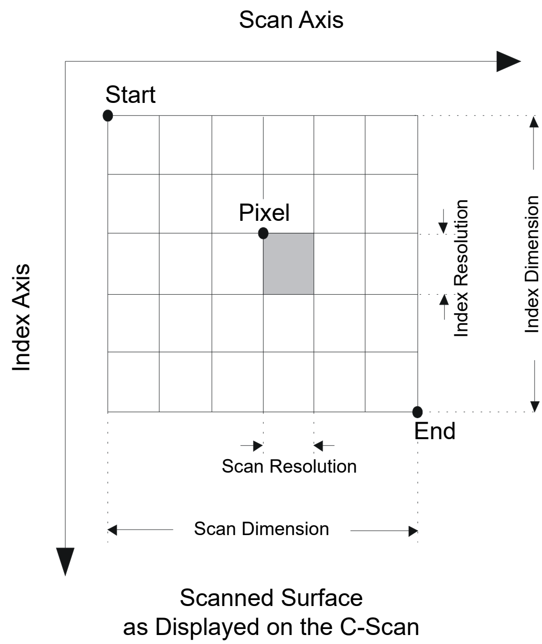


Figure 5-5 Reference system of the scanning mechanism

Operation mode

The **Bidirectional** inspection sequence operates as follows:

1. The scanner proceeds to the **Scan: Start** position set on the **Sequence** tab.
2. The scanner proceeds to the **Index: Start** position set on the **Sequence** tab.
3. The scanner then moves on the scan axis to the position set in the **Scan: Stop** box, while performing data acquisition.
4. Data acquisition is performed at every interval of the **Scan: Resolution** value set on the **Sequence** tab.
5. The scanner moves on the index axis for the distance set in the **Index: Resolution** box.
6. The scanner moves to the **Scan: Start** position, while performing data acquisition.
7. The scanner moves on the index axis for the distance set in the **Index: Resolution** box.
8. The scanner then moves on the scan axis to the position set in the **Scan: Stop** box, while performing data acquisition.

9. Steps 4 to 7 are repeated until the scanner has reached the position set in the **Index: Stop** box. Then, the sequence is finished when the scanner has reached the position set in the **Scan: Stop** box (if the index axis has an odd number of steps), or when it has reached the **Scan: Start** position (if the index axis has an even number of steps).

5.1.4 Unidirectional Scan

The **Unidirectional** scan type is a bidimensional surface-scan (also known as a raster scan) sequence in which two encoders are used to determine the position during the acquisition.

A surface scan uses two axes: (1) the scan axis, which is the mechanical axis of the scanning lines and (2) the index axis, which is the mechanical axis of movement between the scanning lines. At the end of each scan along the scan axis, an increment is added to the position along the index axis. Data acquisition for unidirectional sequences is carried out in one direction only along the scan axis, as shown in Figure 5-6 on page 301. This type of scanning sequence is typically used with scanning mechanisms revealing a rather large backlash phenomenon in scan direction.

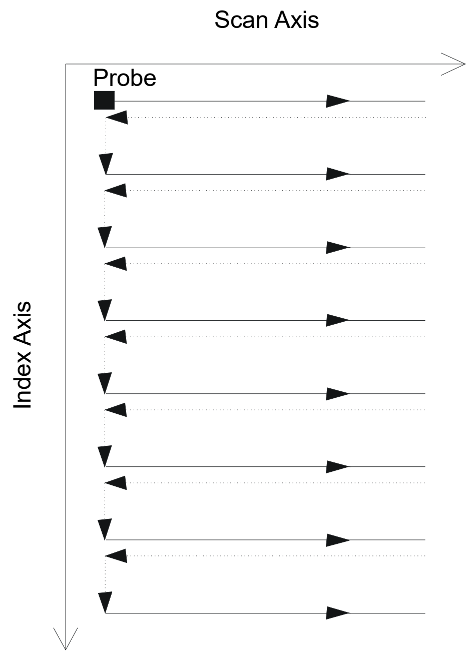


Figure 5-6 Unidirectional surface scan

The user must supply the limits of the inspection surface as well as the spacing between acquisitions (see Figure 5-5 on page 299).

Operation mode

The **Unidirectional** inspection sequence operates as follows:

1. The scanner proceeds to the **Scan: Start** position set on the **Sequence** tab.
2. The scanner proceeds to the **Index: Start** position set on the **Sequence** tab.
3. The scanner then moves on the scan axis to the position set in the **Scan: Stop** box, while performing data acquisition.
4. Data acquisition performed at every interval of the **Scan: Resolution** value set on the **Sequence** tab.
5. The scanner returns to the **Scan: Start** position. No data acquisition is performed during this step.

6. The scanner moves on the index axis for the distance set in the **Index: Resolution** box.
7. The scanner then moves on the scan axis to the position set in the **Scan: Stop** box, while performing data acquisition.
8. Steps 4 to 6 are repeated until the scanner has reached the position set in the **Index: Stop** box. Then, the sequence is finished when the scanner has reached the position set in the **Scan: Stop** box.

Description of the Sequence tab

When you select **Unidirectional** for the sequence inspection type, the **Sequence** tab contains the set of options shown in Figure 5-7 on page 302.

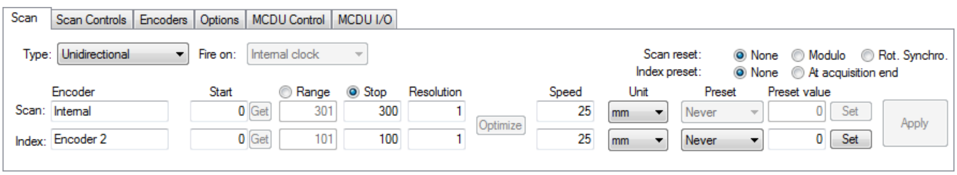


Figure 5-7 The Sequence tab for Unidirectional sequence

The **Sequence** tab for a **Unidirectional** sequence contains the same options as for a **Bidirectional** sequence. For a description of these options, please refer to section 5.1.1 on page 295.

5.1.5 Helicoidal Scan

The **Helicoidal** scan type is similar to the **Bidirectional** scan. With this type of scan, the inspection mechanism performs a helicoidal movement around a cylinder.

A helicoidal scan uses two axes: (1) the scan axis, which is the mechanical axis of the scanning lines (rotation), and (2) the index axis, which is the mechanical axis movement between the scanning lines (axial). The **Helicoidal** sequence operates as follows:

1. The scanner proceeds to the **Start** position specified in the **Sequence** tab for both axes.
2. The scanner then moves on the scan axis and index axis to the **Stop** position specified by the **Range** or **Stop** value of the **Sequence** tab.

3. There is motion on both axes at the same time.
4. Data acquisition is performed at every interval of the **Scan: Resolution** value set in the **Sequence** tab.

Operation mode

The **Helicoidal** inspection scan is similar to the **Bidirectional** scan, but instead of executing a raster scan, the scanner executes an helicoidal movement. In this helicoidal-scan sequence, the two mechanical axes are driven by two motors controlled by an external control unit or axes of a manually driven scanner (slave mode), or by the MCDU-02 (master mode).

The **Helicoidal** inspection sequence operates as follows:

1. The scanner proceeds to the position set in the **Scan: Start** and **Index: Start** boxes of the **Sequence** tab.
2. The scanner then moves on the scan axis and index axis to the position set in the **Scan: Stop** and **Index: Stop** boxes, while performing data acquisition.
3. Data acquisition is performed at every interval of the **Scan: Resolution** value set on the **Sequence** tab.
4. In a helicoidal sequence, the scan axis is projected along the circumference of the cylinder. The **Scan: Start** and **Scan: Stop** values refer to the circumference origin point (0), in distance units or angular units.
5. A signal or a modulo can be used to reset the scan-axis encoder to the **Scan: Start** value after each complete rotation.
6. The sequence is finished when the scanner has reached the position set in the **Index: Stop** box.

Description of the Sequence tab

When you select **Helicoidal** for the sequence inspection type, the **Sequence** tab contains the set of options shown in Figure 5-8 on page 303.

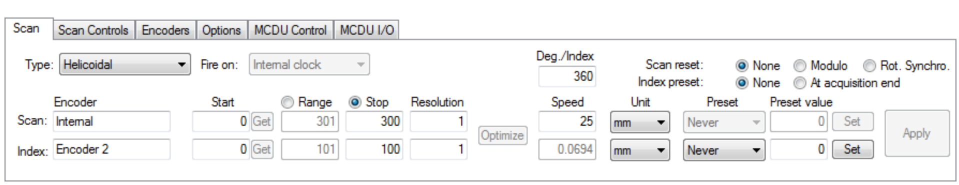


Figure 5-8 The Sequence tab for Helicoidal sequence

The **Sequence** tab for a **Helicoidal** sequence contains the same options as for a **Bidirectional** sequence. For a description of these options, please refer to section 5.1.1 on page 295. This tab also contains three additional option buttons and one additional parameter:

deg/Index

In master mode, this text box defines the distance along the scan axis (in degrees) that is completed for each index increment. The **Index axis inspection speed** value is then deduced from this value, the **Scan axis inspection speed,** and the **Index resolution**.

The chosen **Deg/Index** value is usually slightly larger than 360 degrees, to obtain sufficient overlap between adjacent helicoidal scan lines.

This parameter has no implication in slave mode.

Scan axis reset

Click one of these option buttons to select one of the options used to reset the scan-axis encoder to position zero:

None: scan-axis encoder is never reset.

Modulo: scan-axis encoder is reset to position zero when a maximum value (modulo) corresponding to the **Scan: Stop** value is reached.

Rot. Synchro.: a synchronization signal is used to reset the scan-axis encoder to the **Scan: Start** value.

5.1.6 Angular Sequence

The **Angular** sequence type identifies a two-dimensional surface-scan sequence, where the scan and index axes do not correspond to the orientation of the mechanical axes, such as in the **Bidirectional** and **Unidirectional** sequences. Instead, the scan and index lines form a certain angle with the orientation of the mechanical axes (see Figure 5-9 on page 305). Two position encoders are used to determine the position during the acquisition.

A surface scan uses two axes: (1) the scan axis, which is the mechanical axis of the scanning lines, and (2) the index axis, which is the mechanical axis displacement between the scanning lines. At the end of each scan along the scan axis, an increment is added to the position along the index axis. In this sequence, the mechanical axes work together in such a way as to produce the desired scanning pattern. Data acquisition for angular sequences is carried out in both the forward and backward directions along the scan axis.

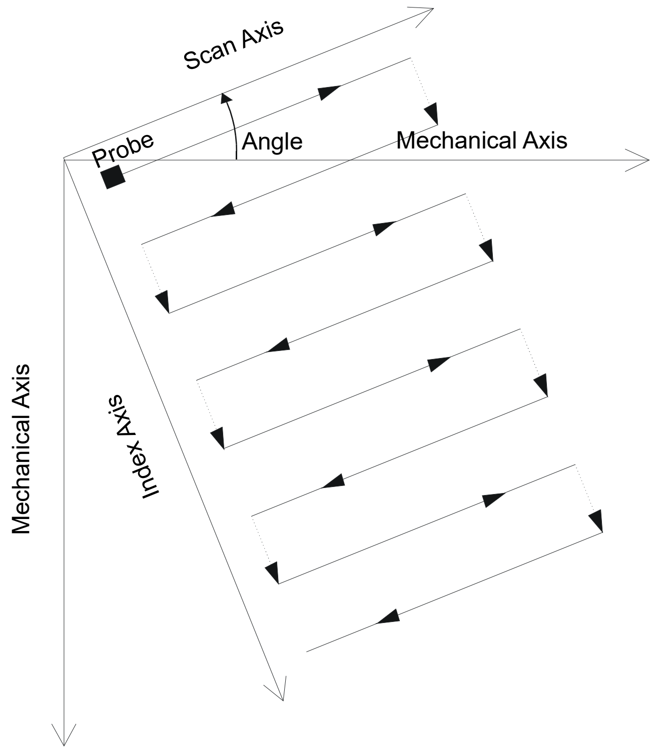


Figure 5-9 Angular surface scan

Operation mode

The **Angular** inspection sequence operates as follows:

1. The scanner proceeds to the position set in the **Scan: Start** and **Index: Start** boxes of the **Sequence** tab.
2. The scanner then moves on the scan axis, according to the specified angle, to the position set in the **Scan: Stop** box, while performing data acquisition.
3. Data acquisition is performed at every interval of the **Scan: Resolution** value set on the **Sequence** tab.
4. The scanner moves on the index axis, according to the specified angle, for the distance set in the **Index: Resolution** box.

5. The scanner moves on the scan axis, according to the specified angle, to the **Scan: Start** position, while performing data acquisition.
6. The scanner moves on the index axis, according to the specified angle, for the distance set in the **Index: Resolution** box.
7. The scanner then moves on the scan axis, still according to the specified angle, to the position set in the **Scan: Stop** box, while performing data acquisition.
8. Steps 4 to 7 are repeated until the scanner has reached the position set in the **Index: Stop** box. Then, the sequence is finished when the scanner has reached the position set in the **Scan: Stop** box (if the index axis has an odd number of steps), or when it has reached the **Scan: Start** position (if the index axis has an even number of steps).

Description of the Sequence tab

When you select **Angular** for the sequence inspection type, the **Sequence** tab contains the set of options shown in Figure 5-10 on page 306.

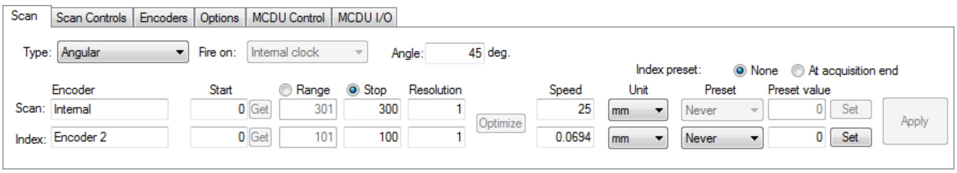


Figure 5-10 The Sequence tab for Angular sequence

The **Sequence** tab for an **Angular** sequence contains the same options as for a **Bidirectional** sequence. For a description of these options, please refer to section 5.1.1 on page 295. This tab also contains an additional parameter, which is required for this sequence type:

Angle

This box is used to set the angle that the scan line forms with the orientation of the mechanical axis.

5.1.7 Custom Sequence

Selecting the **Custom** sequence type automatically opens the **Load Custom Program File** dialog box (see Figure 5-11 on page 307). This dialog box is used to select and load a special type of sequence predefined in a .gal file.

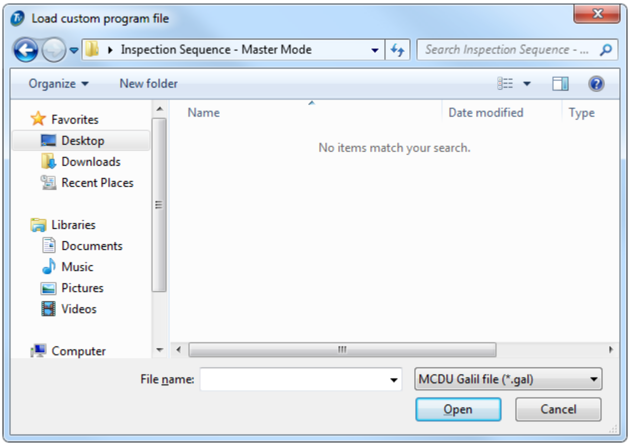


Figure 5-11 The Load custom program file dialog box

5.2 Working with Encoders

You can use one or more encoders in your setup to measure the position of the probe on the scanned area.

Tomoview supports different types of encoders (see section 5.2.1 on page 307).

You need to calibrate each encoder (see section 5.2.2 on page 309).

5.2.1 About Encoder Types

TomoView supports various types of encoders. You can select the encoder type on the **Encoders** tab of the **Scan and Mechanical Settings** dialog box (see Figure 5-12 on page 307).

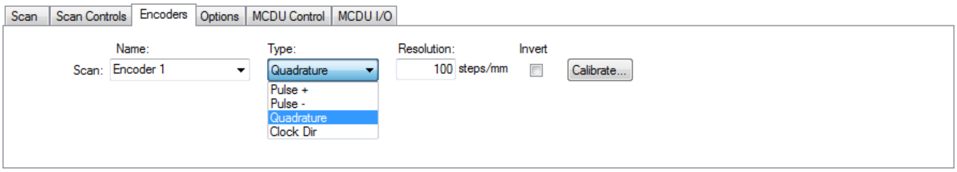


Figure 5-12 Selecting the encoder type on the Encoders tab

Each encoder input on your acquisition unit has two channels, A and B, enabling a dual-channel encoder for quadrature reading of the resolution.

The available encoder types are:

Clock Dir

Select this option when you use a stepper controller and its documentation specifies that the position output signal is a clock/direction type (5 V pulse for the position/speed and 5 V signal for the direction).

Quadrature

Select this option when the attached encoder (5V TTL output) is a dual-channel output encoder. The channels are generally named A and B. When the encoder is rotating clockwise (from left to right in Figure 5-13 on page 308), 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 a falling edge on channel A or channel B. This means that if the real encoder resolution is 1,000 steps/revolution, the final resolution with the quadrature reading is 4,000 steps/revolution.

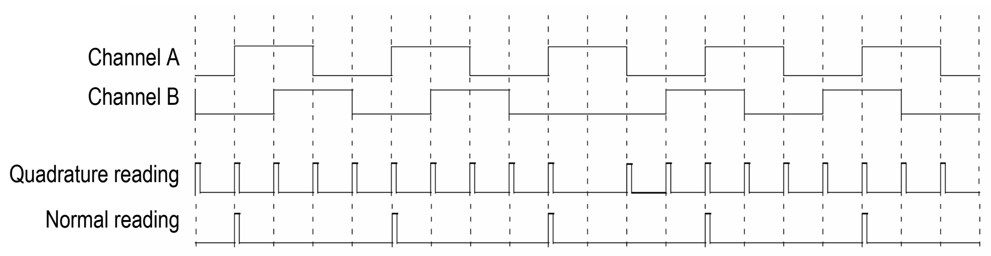


Figure 5-13 Quadrature and normal reading of the resolution

Up

The decoder only reads channel A and increments the counter even if the encoder is turning counterclockwise. The final resolution is the real encoder resolution.

Down

The decoder only reads channel A and decrements the counter even if the encoder is turning clockwise. The final resolution is the real encoder resolution.

Clock/Dir Up

The decoder only reads channel A and increments the counter. When the signal of channel_B (direction) is high, the acquisition stops to prevent overwriting the data while you move the probe back and the counter is decremented.

Clock/Dir Down

The decoder only reads channel A and decrements the counter. When the signal of channel_B (Dir) is high, the acquisition stops to prevent overwriting the data while you move the probe back and the counter is incremented.

Quad Up

The decoder reads channel A and channel B in quadrature mode (4 times the encoder resolution) and increments the counter when the encoder rotates clockwise. When the encoder rotates counterclockwise, the acquisition stops to prevent overwriting the data and the counter decrements.

Quad Down

The decoder reads channel A and channel B in quadrature mode (4 times the encoder resolution) and decrements the counter when the encoder rotates counterclockwise. When the encoder rotates clockwise, the acquisition stops to prevent overwriting the data and the counter increments.

5.2.2 Calibrating an Encoder

You need to calibrate an encoder to validate, fine tune, or determine its resolution. The resolution of an encoder is the number of steps corresponding to a travelled distance of 1 mm or 1 in., or 1 degree.

Perform the following procedure for each encoder that you are using.

To calibrate the encoder of an axis

1. On the **Component** toolbar, click the Scan and Mechanical Settings button ().
2. In the **Inspection Sequences and Mechanical Settings** dialog box, click the **Scan** tab, and then perform the following tasks:
   a) In the **Type** box, select the appropriate scan type for your application. Do not select **Free running** as this type does not support encoders.
   b) In the **Unit** box, select the appropriate linear or angular units.
3. In the **Encoders** tab (see Figure 5-14 on page 310):

a) In the **Type** box, select the type of your encoder (see section 5.2.1 on page 307).

b) Click **Calibrate**.

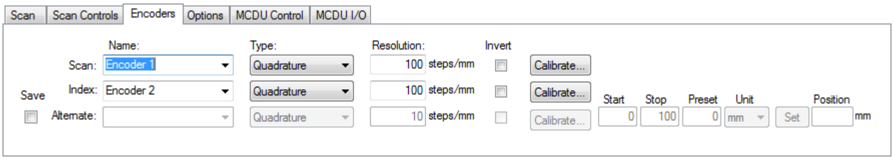


Figure 5-14 The Encoders tab of the Scan and Mechanical Settings dialog box for an Encoded - 2 axis scan type

4. While in the **Calibration of Encoder** dialog box (see the example shown in Figure 5-15 on page 311):

   a) Optionally, move the required encoder, or the mechanics attached to the encoder, to a known position on the axis, and then click **Set**.
   This operation sets the current position to the value of the **Preset value** box that has been specified on the **Sequence** tab.

   b) Click **Set Begin**.

   c) Move the encoder over a defined distance on the axis in the direction that is considered positive. In the case of master mode, use the **Movement** buttons to move the scanner.

   d) Click **Set End**.

   e) In the **Set Distance** text box, enter the travelled distance, and then click **Set Distance**.
   The resulting encoder resolution appears in the **Calculated resolution** box.

   f) When needed, click **Clear** to reset the parameter values, and then return to step 4.*a*.

   g) Click **OK** to apply the calculated encoder resolution.

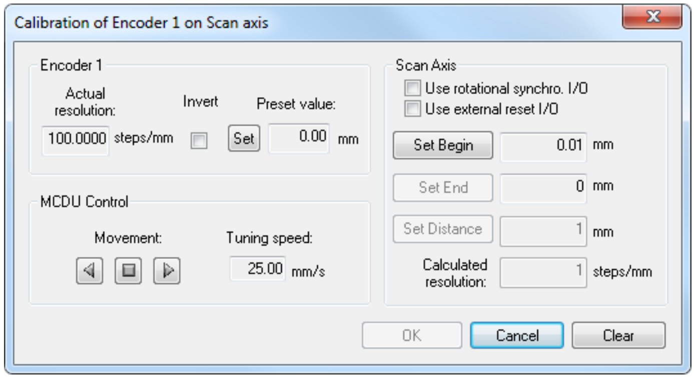


Figure 5-15 The Calibration of Encoder dialog box for encoder 1 on the scan axis

5.3 Working with the Position-Dependent Setup

The **Position-Dependent** module was developed for the inspection of samples having changing geometries requiring different setup files in a single acquisition.

5.3.1 About the Position-Dependent Setup Module

Using the Position-Dependent Setup module, you can synchronize the different setup and data files. The module loads the appropriate setup and initiates the data storage on a disk, based upon the current manipulator position on the index axis.

For example, a first setup is loaded and a scan is performed from y_1 to y_2: The data is stored when the position value (x_1,y_2) is reached. Then, setup file 2 is loaded and data acquisition starts from index line y_3 to y_4. On completion of the scan (x_2,y_4), the data is stored in a new file and setup file 3 is loaded. Data acquisition starts at point (x_2,y_5) and ends at (x_1,y_6). On completion, the data is stored and a new file is created containing all data. Figure 5-16 on page 312 illustrates this mechanism.

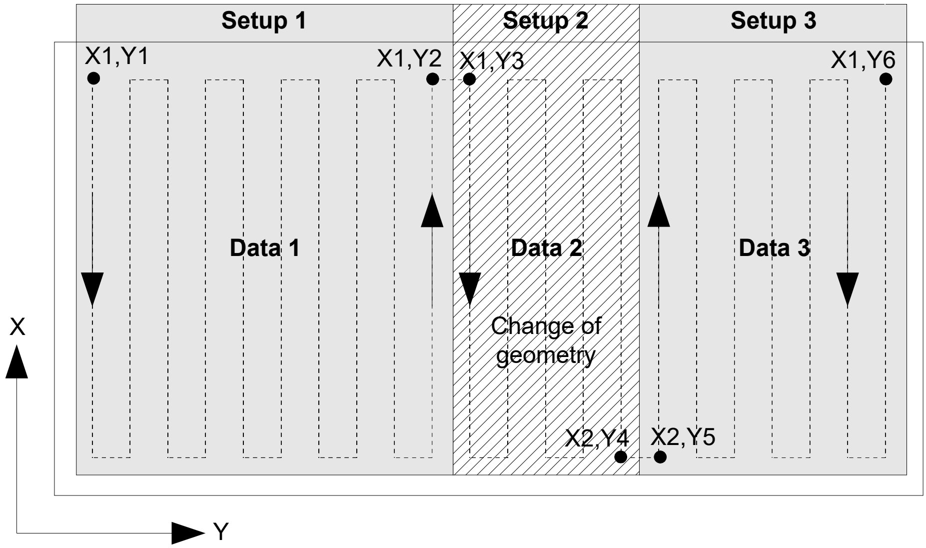


Figure 5-16 Synchronization of setup and data files

5.3.2 Position Dependent Setup Dialog Bar

The position-dependent setup is set with the **Position Dependent Setup** dialog bar (see Figure 5-17 on page 312). You open this dialog box by clicking  on the **Main** toolbar.

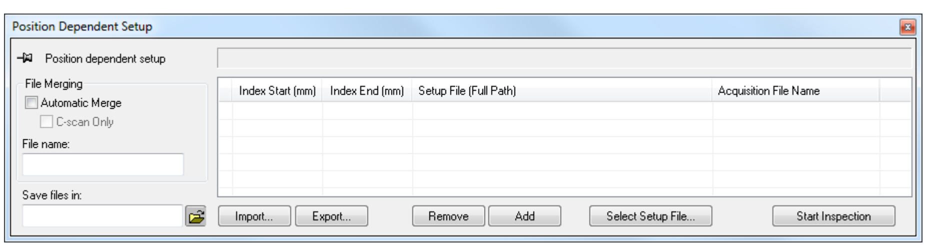


Figure 5-17 The Position Dependent Setup dialog bar

The **Position Dependent Setup** dialog bar contains the following options:

Control list

The columns in this list box display the following information:

Setup File (full path): list of setup files used for the inspection

Acquisition File Name: associated data acquisition file

Index Start and **Index End**: index positions where the setups are valid

Add

This button adds a line for a new setup to be used during the inspection.

Remove

This button removes a line (a setup) from the list.

Select Setup File

This button selects the setup to be used for a particular line.

Import, **Export**

These buttons load or save a particular setup sequence.

Automatic merge

This check box, when selected, specifies that the data acquisition files are to be merged into one at the end of the inspection.

File name

This box specifies the name of the final merged file.

Save files in

This box specifies the name of the folder where the data acquisition files are to be saved.

Start Inspection

This button starts the inspection, with the setup files and the acquisition file names defined in the control list.

5.3.3 Use of the Position-Dependent Setup Module

Once the different setup files to be used throughout the acquisition have been created, follow these steps to configure the Position-Dependent module:

IMPORTANT

Before using the position-dependent setup module, you must properly define the setups, particularly the mechanical sequence of each setup.

To use the position-dependent setup module

1. Set the scan and index positions to the start of the first setup to be used.
2. On the **Main** toolbar, click  to open the **Position Dependent Setup** dialog bar.
3. Click **Add**.

   A new line appears in the control list, with an asterisk (*) in the left column.

   a) Click **Select Setup File** and choose a setup file to be associated to the first section of the sample to inspect.

   b) In the **Index Start** and **Index End** columns, enter the index positions to be associated to the this setup file.

   c) In the **Acquisition File Name** column, enter the dat file name that will be generated at the end of the section.
4. Repeat step 3 for all setups to be used for the inspection.
5. Select the **Automatic merge** check box if you want the individual data acquisition files to be merged into one.

   a) Select the **C-Scan Only** if only the C-Scan data is to be merged.

   b) In the **File Name** box, enter the name of the merged file.
6. In the **Save files in section**, click  and select the folder where files are to be saved in the dialog box.
7. Click **Start Inspection** to start the acquisition according to the specified setup files.

5.3.4 Limitations

The position-dependent setup module has the following limitations:

- The change in setup can only be defined along the index axis.
- During setup change, the manipulator must be stopped for TomoView to load the setup.

- At inspection start, the first setup loaded is the one closest (at higher values) to the current index position.
- The **Index Start** and **Index End** positions must be exactly the same as the ones defined in the setup files.
- The total size of the merged file cannot be higher than the size allowed by TomoView available in the **Acquisition File Properties** section on the **General Settings** tab of the **Preferences** dialog box.

5.4 Defining the Automatic Savings Options

In the **Options** tab of the **Scan and Mechanical Settings** dialog bar (see Figure 3-85 on page 159) you can use the **File Naming Options** section to configure the way the data files will be saved at the end of the inspection sequences. See Figure 5-18 on page 315

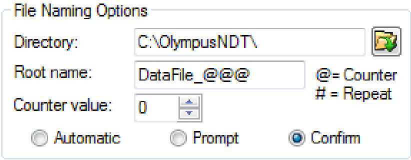


Figure 5-18 The File Naming Options dialog box

To configure the automatic saving options

1. Set **Directory** to the folder where the files are to be saved.
2. Set **Root Name** to the name of the data files to be saved.
   - The @ character inserts a counter, which will be automatically increase the file name by one. For example, entering **test@** will generate test0.rdt, test1.rdt, test2.rdt, and so on.
   - The # character adds the desired number of digits for repeats. For example, typing **test@##** will generate test000.rdt, test100.rdt, test200.rdt, and so on.
   - If a file exists (for example, test000.rdt) then the new file will be saved under the following name: test001.rdt (first repeat of the test000.rdt file name).
3. Set **Counter value** to configure the counter start value of the counter that is inserted in the file name with the @ character.
4. Select the saving mode from **Automatic**, **Prompt**, or **Confirm**.

- The **Automatic** mode will save the data files without asking for a confirmation.
- The **Prompt** mode will deactivate the automatic file naming and ask you to enter the data file name at the end of every acquisition.
- The **Confirm** mode will ask you to confirm the file name defined in the Root name before saving the data file.

6. Basic Analysis Using TomoView

Once acquisitions have been completed, TomoView can be used to analyze the resulting data files. This section describes the basic analysis in TomoView, demonstrating how you can manage data files to create simple and convenient reports making the inspection results easy to understand.

6.1 Working with Data Files in TomoView

This section describes how to open TomoView and OmniScan data files and merge data files in order to combine multiple inspection data to a single file that can then be analyzed and described in a simple report.

6.1.1 Opening a TomoView Data File

The **Open** dialog box (see Figure 6-1 on page 318) is used to select and load an ultrasonic data file as well as the other data types that can be linked to this file.

To open a TomoView data file

1. In the **Document** toolbar, click .
2. Make sure that **Data** is selected in the upper left section of the **Open** dialog box.
3. Select **File Content** and **Processing** options (see below for more information on these options).
4. Click **Open**.

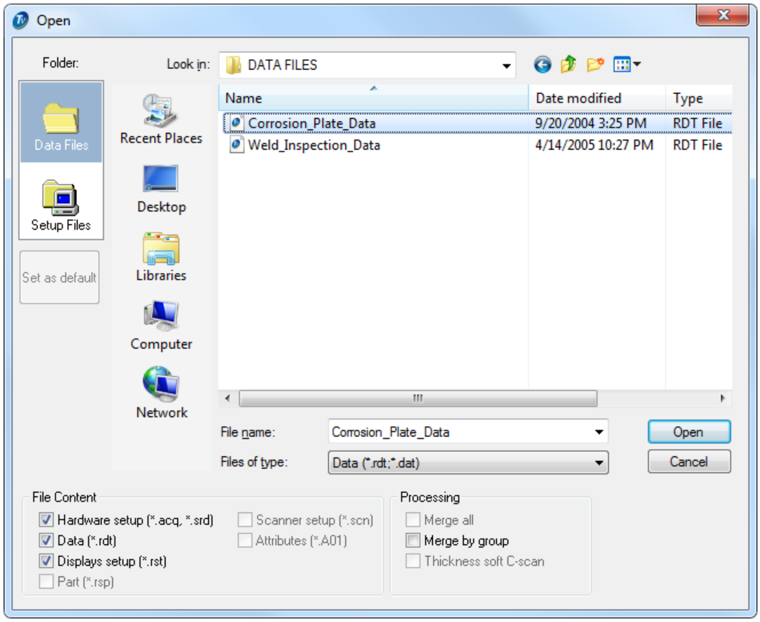


Figure 6-1 The Open dialog box

The **File Content** and **Processing** group boxes contain the following options:

File Content

This group box encloses check boxes, with which you can select the associated files (for a description of the file types, see section 3.18 on page 221).

For example, an .rdt data file can be saved with an .A01 data file resulting from analysis processing. You can then either open the original data alone (.rdt file), or open the original data along with the data modified in analysis (.rdt and .A01 files). The principle of associated files allows the original data to remain intact after the data file is saved.

Only the file types that are associated with the .rdt file when saving the data can be selected. The dimmed file types are not available.

Processing

You can use the check boxes enclosed in this group box to select optional calculations to be performed on the file to be opened. After opening the file, TomoView calculates the selected processing options and adds the associated data groups to the file.

- The **Merge all** check box performs the same calculations as the menu command **Processing > Automatic Volumetric Merge**.
- The **Merge by group** check box performs the same calculations as the menu command **Processing > Automatic Volumetric Merge by Group**.
- The **Thickness soft C-scan** check box is only available when converting OmniScan data files (see section 6.1.3 on page 322 for more information).

6.1.2 Importing OmniScan Data Files

This section explains how to import OmniScan data files and convert them into TomoView files using the **OmniScan File Converter**. By converting OmniScan data files, you can use the powerful analysis tools offered by TomoView to analyze OmniScan data.

To import an OmniScan file into TomoView

1. On the **Component** toolbar, click  to open the **OmniScan File Convertor** (see Figure 6-2 on page 320).

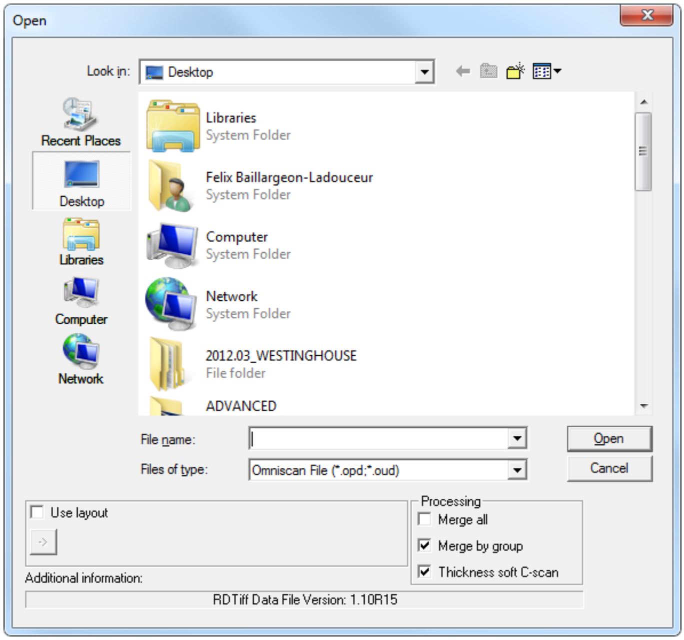
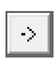


Figure 6-2 The Open dialog box

2. To apply one of the template layouts on the converted data files, select the **Use layout** check box and the  to select a layout. When the converted files are opened in TomoView, all views, readings, and preferences will be properly set (see Figure 6-3 on page 320).

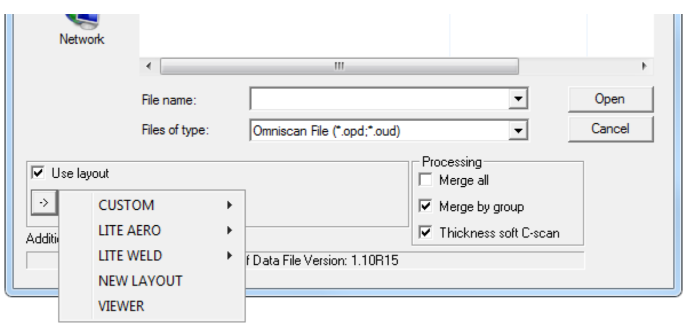


Figure 6-3 The Use Layout list

3. In the **Processing** group box:
   - Select the **Merge all** check box to launch the volumetric merge processing after the conversion (the same calculations as the menu command **Processing > Automatic Volumetric Merge**).
   - Select the **Merge by Group** check box to launch the volumetric merge by group processing after the conversion (the same calculations as the menu command **Processing > Automatic Volumetric Merge by Group**).
   - Select the **Thickness Soft C-Scan** check box to create a thickness C-Scan using the original OmniScan thickness information.

NOTE

If you do not select any layout file, you view blank panes on the TomoView screen. However, it is possible to select a template layout when TomoView is opened.

---

4. Select an OmniScan **.oud** (Conventional UT) or an**.opd** (Phased Array) data file and click **Open**.

   You have a choice between an .oud file for data acquired with conventional ultrasonic technology and an .opd file for phased array ultrasonic technology.

   A **Save As** dialog box appears (see Figure 6-4 on page 322). You can use this dialog box to select where the template file is to be saved.

NOTE

If the file that you are trying to convert is not supported, you might need to use the latest Stand Alone OmniScan Converter or upgrade to a more recent TomoView version that supports new formats.

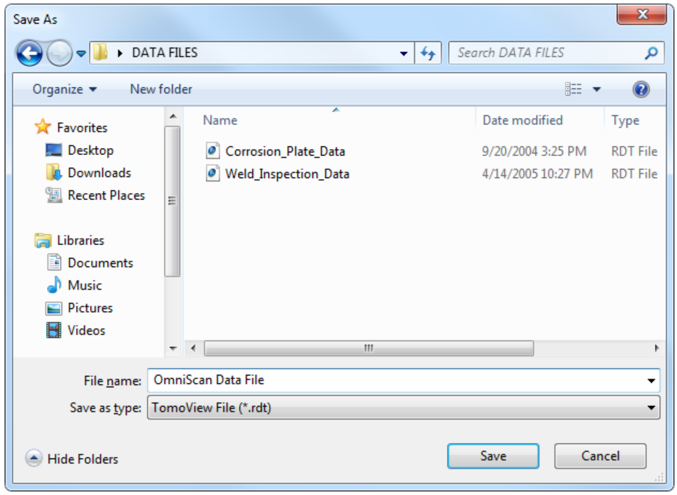


Figure 6-4 The Save As dialog box

5. In the **File name** text box, type a name for the **.rdt** file that will be created.
6. Click **Save**.

6.1.3 Merging Data Files

With the data file merger component, you can merge multiple data files into one to create a new data (.rdt) file that contains all the data (groups) of the original files. You can perform the file merge process on both conventional and phased array data files.

As the resulting data file is a common .rdt file, you can apply most of the generic TomoView processing algorithms to this file (for example: volumetric merge of data, hysteresis correction of data).

The data file merger component in combination with the volumetric merge function can be extremely useful for applications where you perform the part or weld inspection in various sections, each contained in a different data file. You can use the consecutive application of the **Data File Merger** function and the **Volumetric Merge** function to create data groups covering the complete inspected volume. In addition, you can correct mechanical offsets from the **View > Properties** dialog box, in the **Parameters** tab.

NOTE

The total size of the merged file cannot exceed 1 GB.

To merge data files

1. On the menu, click **File > Merge Files**.
2. In the **Data File Merger** dialog box (see Figure 6-5 on page 323):

   a) Click the browse button () on the right of the **Destination file** text box to select a folder and a file name for the final merged file.

   b) Click the add button () to select the first data file (.rdt) to be merged.

   c) Repeat step 2.*b* to add the other data files to be merged.

   d) If files that you want to merge are currently open in TomoView, close them.

   e) Select the **Merge companion file (A01)** check box to also include the companion file data in the merge.

   f) Select the **C-scan only** check box to only merge C-scan data. Since A-scans are not saved, this option allows to create files with significantly larger patch area before reaching the data file size limit.

   g) Click **Merge**.

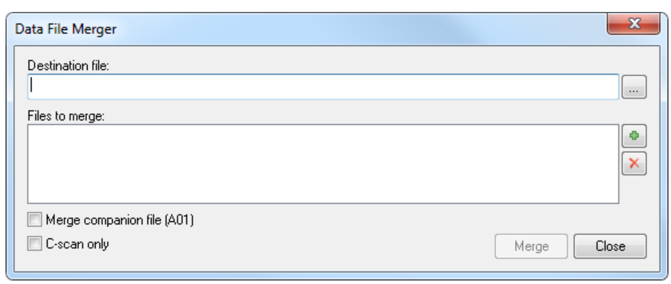


Figure 6-5 The Data File Merger dialog box

6.2 Working with the Indication Table and Report Component

The indication table is a key feature of TomoView (see Figure 6-6 on page 324). Use the **Indication Table** to gather flaw indication information and to create an HTML report. The basic steps are:

- Use the cursors and the Zone tool to highlight an indication
- Open the indication table and add an indication
- Add comments and extra readings to the table
- Customize the report
- Preview and produce the HTML report

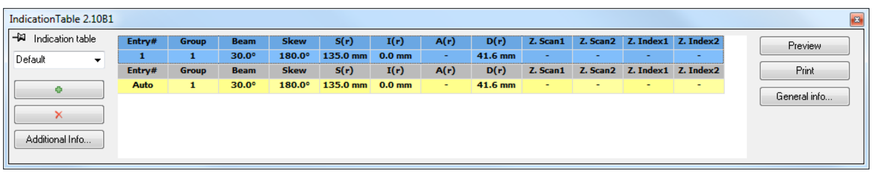


Figure 6-6 The indication table

NOTE

TomoView automatically saves the **Indication Table** in the display setup file (.RST) and automatically saves the numerical reading indication information contained in the **Indication Table** in an attribute file (.R01).

6.2.1 Adding an Indication to the Indication Table

Use the Zone tool and the indication table to quickly select the area corresponding to an indication and create a record for the indication.

TIP

You can customize the sizing settings in the **Preferences** dialog box, in the **General Settings** tab under **Flaw-Sizing Settings**.

To add an indication to the indication table

1. Select a layout with views that best illustrate the indications in the part (see the example shown in Figure 6-7 on page 325).

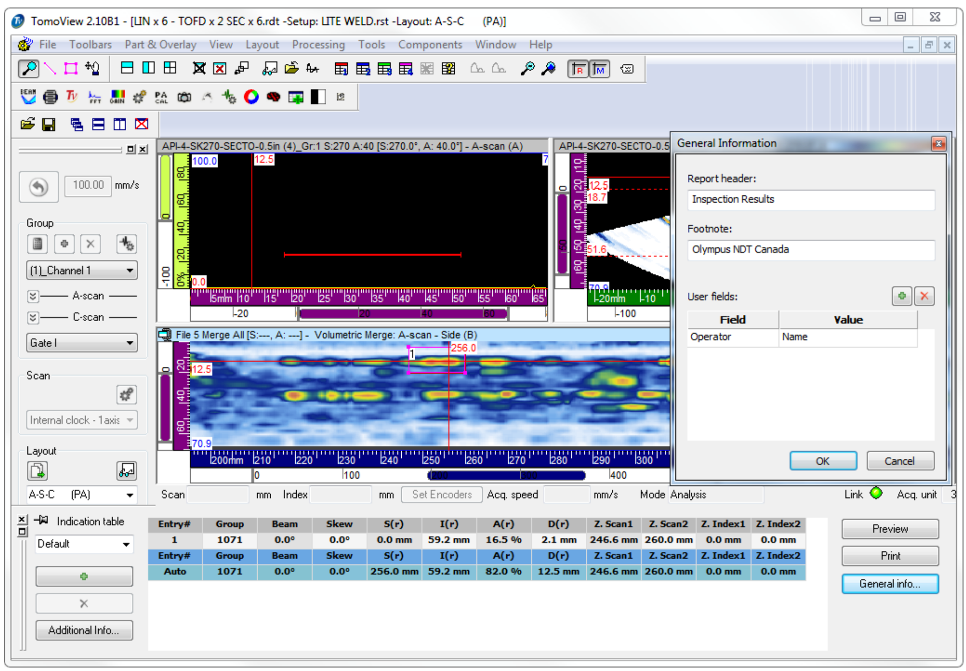
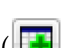


Figure 6-7 Example of the indication table, the cursors, and the Zone tool used to document an indication

2. On the Component toolbar, click the Indication Table button () to display the **Indication Table** dialog box.
   The first line of the table indicates the current value of the selected readings.

3. Position the reference and measurement cursors to mark the indication.
4. Using the Zone tool, draw a zone around a flaw indication.
5. In the **Indication Table** dialog box, select a predefined reading category (see Figure 6-8 on page 326) to select the readings that appear in the table.

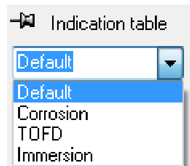


Figure 6-8 Selecting a predefined reading category in the indication table

TIP

You can select a different reading category for each indication table entry.
The indication image is a snap shot of the data display views taken when you click . When you want to put the emphasis on the defects characteristics, configure the views accordingly before clicking .

6. In the **Indication Table** dialog box, click .

   The selected flaw indication is added to the **Indication Table** list and a red marking rectangle labeled with the indication number appears on the view.
7. As needed, repeat steps 3 to 6, to mark other indications.
8. You can add a comment to an indication:
   a) In the **Indication Table** dialog box, select the indication to which you want to add a comment by clicking on the corresponding line in the table.
   b) Click **Additional Info**.
   c) In the **Custom Readings** dialog box that appears (see Figure 6-9 on page 327), type your comment for that indication in the **Comment** text box.
   The comment then appears in the **Comments** section of the report for the selected indication.

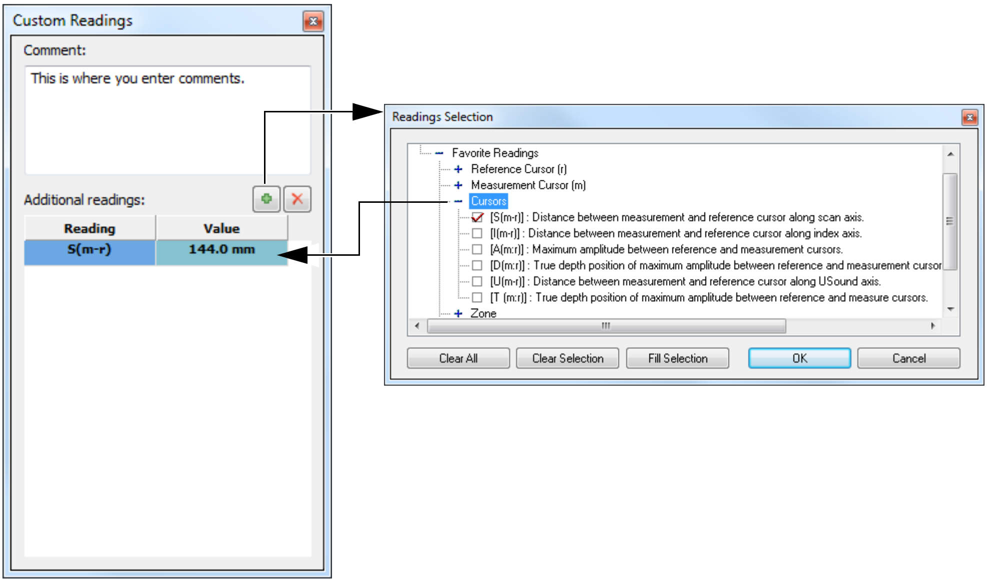


Figure 6-9 Adding a comment and an additional reading to an indication

9. You can also include additional readings to the indication table (see Figure 6-9 on page 327):
   a) In the **Indication Table** dialog box, select the line for the **Entry# = Auto** indication.
   b) In the **Custom Readings** dialog box, click .
   c) In the **Readings Selection** dialog box that appears, select the check box of one or more readings that you want to add to the selected indication, and then click **OK**.
   The selected readings appear in the **Custom Readings** dialog box, under **Additional readings**.

NOTE

Additional readings appear only for new indications, not for those already entered in the indication table.

6.2.2 Producing an Ultrasonic Inspection Report

Once you have completed the indication analysis using the indication table, you can produce an HTML ultrasonic inspection report from the **Indication Table** dialog box. This report includes the following elements for each defined group of the configuration:

- Setup information
- Inspected part information
- Scanned area information
- Content of the indication table
- Views for each entry of the indication table
- Customized information

To produce an ultrasonic inspection report

1. Gather indication information in the indication table (see section 6.2.1 on page 324).
2. From the **Indication Table** dialog box click **General info** to open the General Information dialog box and customize the HTML report (see Figure 6-10 on page 329)
   a) In the **Report Header** text box, type your report header information.
   The text appears at the beginning of the report (see Figure 6-11 on page 329).
   b) In the **Footnote** text box, type your report footnote information.
   The text appears in the **Notes** section at the end of the report (see Figure 6-12 on page 330).
   c) Click .
   d) Type the custom information label in the **Field** text box and the corresponding value in the **Value** text box.
   The added user fields appear in the second section from the top of the report (see Figure 6-11 on page 329).
   e) Click **OK**.

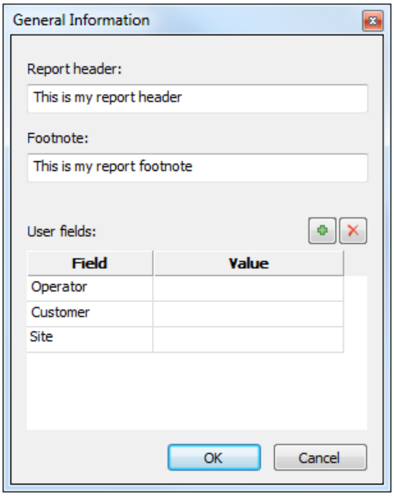


Figure 6-10 Example of the General Information dialog box

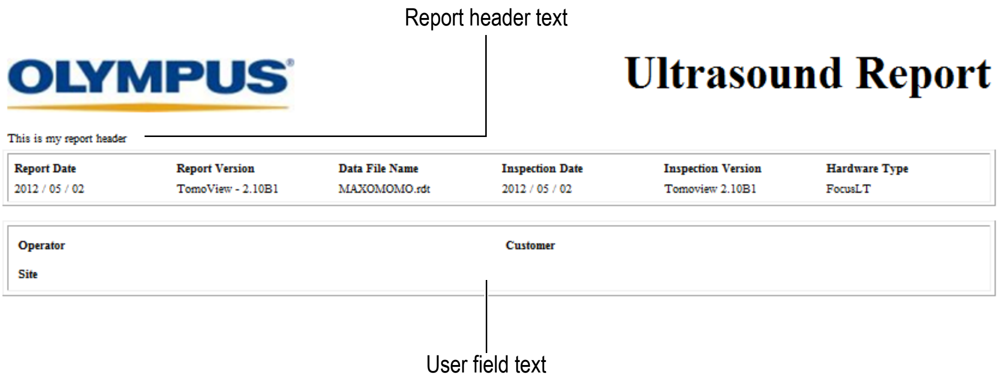


Figure 6-11 Example of a report header and user fields

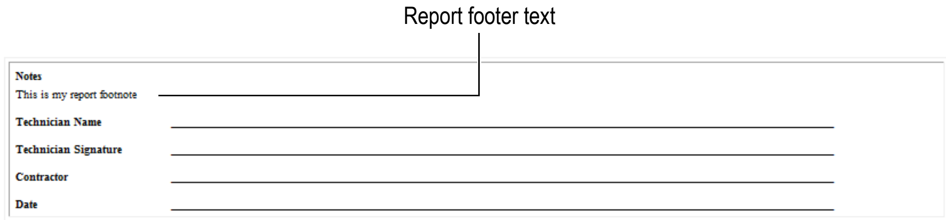


Figure 6-12 Example of a report footer

3. Back in the **Indication Table** dialog box, click **Preview**.
   The HTML report appears in your Internet browser.
4. Review the report.
5. In the **Indication Table** dialog box, click **Print**.
6. In the **Print** dialog box, select the printer, and then click **Print** to print the report.

6.2.3 Changing the Logo in the Inspection Report

The Evident logo appears by default at the top of the inspection report. You can change the logo.

TIP

Ensure to change the logo before creating the report. Otherwise, the old image will appear in the report. You then need to delete the report file (.r01) and add the indications again.

To change the logo appearing at the top of the inspection report

1. Create a small image of the logo that you want to see at the top of the inspection report.

TIP

Use a logo image that is about the same size as the Evident logo image (200 pixels wide by 38 pixels high).

2. In the [Installation Folder]\TomoView210 folder:
   a) If you do not want to loose the Evident logo file, rename the logo.jpg file to Evident_logo.jpg.
   b) Save the image of your logo in the JPEG format using the logo.jpg file name.

   Your logo appears at the top of the next inspection report that you generate.

7. Advanced Analysis Using TomoView

This chapter describes how to use the multiple advanced analysis features available in TomoView.

7.1 Working with the Thickness C-Scan Process

A thickness C-Scan is a part thickness color illustration used to highlight thickness variations. This view is useful for the monitoring of part wear or corrosion.

To create a thickness C-Scan

1. From the menu, select **Processing > Create Thickness C-Scan.**
2. In the **Create Thickness C-Scan** dialog box, (see Figure 7-1 on page 334
   a) In the **Groups** list, select the group to create a thickness C-Scan.
   b) In the **Thickness types** list, for example, select **A ^ - I/.**
   c) To customize the color palette limits, from the **Thickness gate** area, select **Override with user values**, and then set **Min.** and **Max.** parameters to the desired values.
   d) Click **Create**.
   e) Click **Done**.
3. Choose a view in a layout to be configured as a thickness C-Scan.
4. On the menu, choose **View > Contents**.
5. In the **Contents** dialog box, select a thickness C-Scan view (see Figure 7-2 on page 334).

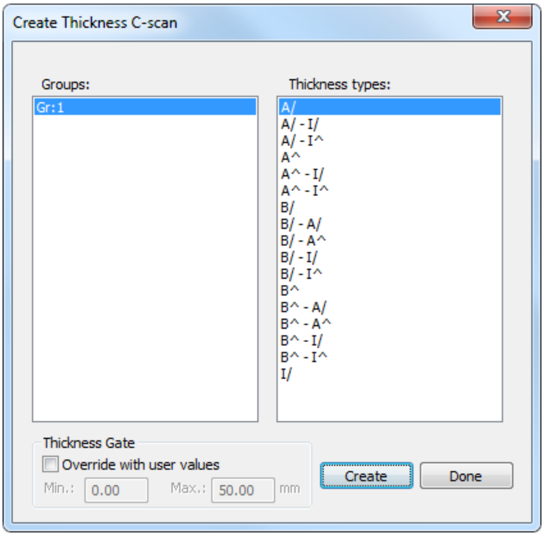


Figure 7-1 The Create Thickness C-Scan dialog box

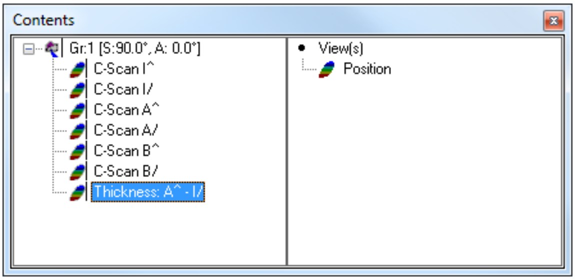


Figure 7-2 Selecting a thickness C-Scan view

The thickness C-Scan illustration appears in the view (see example in Figure 7-3 on page 335).

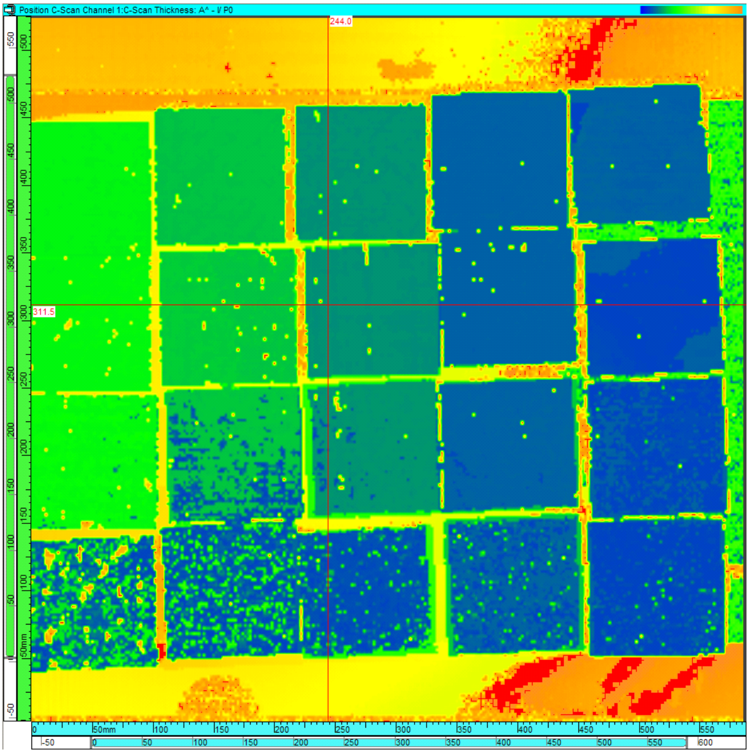


Figure 7-3 Example of a thickness C-Scan

7.2 Working with the Volumetric Merge Process

A TomoView view can only display the data of one data group. The volumetric merge function condenses the information of several groups into one group, so that you can view more information in a single view.

In data analysis mode, you can use the **Volumetric Merge** function to perform a merge of the ultrasonic data acquired with various groups. The merging process compares the amplitude obtained at each point of the inspected volume by the considered groups, and creates a new data group with the maximum amplitude observed at each position in the inspected volume.

The volumetric merge function is a wizard taking you step by step through the merging configuration process. You can save the configuration defined with the wizard in a volumetric merge configuration file (.vmc), and recall the file to quickly perform a similar volumetric merge. Refer to section 7.2.1 on page 336 for details on the volumetric merge configuration file format.

TomoView saves the created merged data in the attributes file (.A01) that is saved as part of the data file (.rdt).

You can also use the **Automatic Volumetric Merge** that uses the default volumetric merge parameters, thus bypassing the steps of the volumetric merge wizard. All groups (except TOFD groups) are merged together but the original data is always preserved (see section 7.2.2 on page 342). You can also perform an **Automatic Volumetric Merge** when converting OmniScan data files.

7.2.1 Performing a Volumetric Merge

The volumetric merge function is a wizard taking you step by step through the merging configuration process.

To perform a volumetric merge

1. Open a data file containing more than one beam or more than one data group.
2. On the menu, choose **Processing > Volumetric Merge**.
   The **Volumetric Merge** wizard starts.
3. Under **Groups Selection** (see Figure 7-4 on page 337), select which beams or groups to merge:
   a) Under **Merge Data**, select the beam or group type to be displayed in the **Available Groups** list.
   b) In the **Available Groups** list, select the beams and groups you wish to merge, and then click **Add**.
   The selected beams or groups appear in the **Selected Groups** list.
   c) If you want to use a previously defined volumetric merge configuration (VMC), click **Load Configuration from VMC File**, and then select the desired file (.vmc).
   d) Click **Next**.

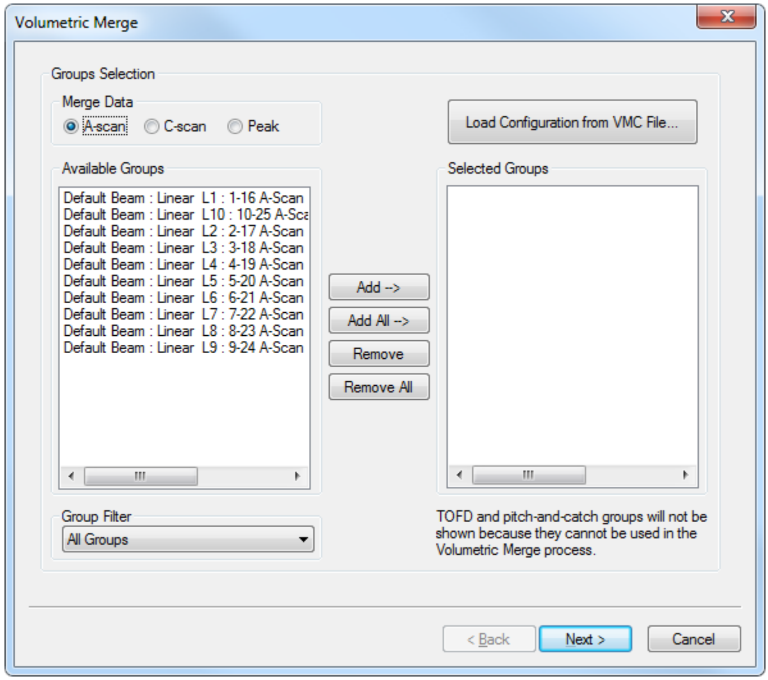


Figure 7-4 The Groups Selection group box

NOTE

The volumetric merge function does not work when a cylindrical specimen is defined. Also, it is impossible to merge different data types together (for example, A-scan with C-scan).

4. Under **Parameters Settings**, select which beams or groups to merge:
   a) Under **Processing Options**, select which skips to use.
   b) Click **Next**.
5. Referring to (see Figure 7-5 on page 339), perform the following tasks:
   a) Under **New Group Information**, type a name for the beam or group to be created.

b) Under **Merge Type**:

- Select **Full** to perform a merge of all probe positions for the considered beams or groups, and then set **Thickness** to the thickness of the inspected part. It is taken into account for the calculation of the rebounds.

  OR

  Select **Probe position selective** to perform a merge for particular probe positions, which you specify in the **From** and **To** boxes for the **Scan**, **Index**, and **USound** axes.

c) Under **Generated Data**:

(1) Select **Include group information** to keep the information related to the origin of each point of the merged data group and save it in the attributes file (.A01) with the merged data.

(2) Select **Generate peak A-scan instead of A-scan** to generate a peak A-scan for the merged group.

(3) Set **Limit** to the amplitude threshold level for the generation of the peak A-scan.

(4) Set **Peak number** to the number of peaks (maximum value is 127) for the generation of the peak A-scan.

d) Click **Next**.

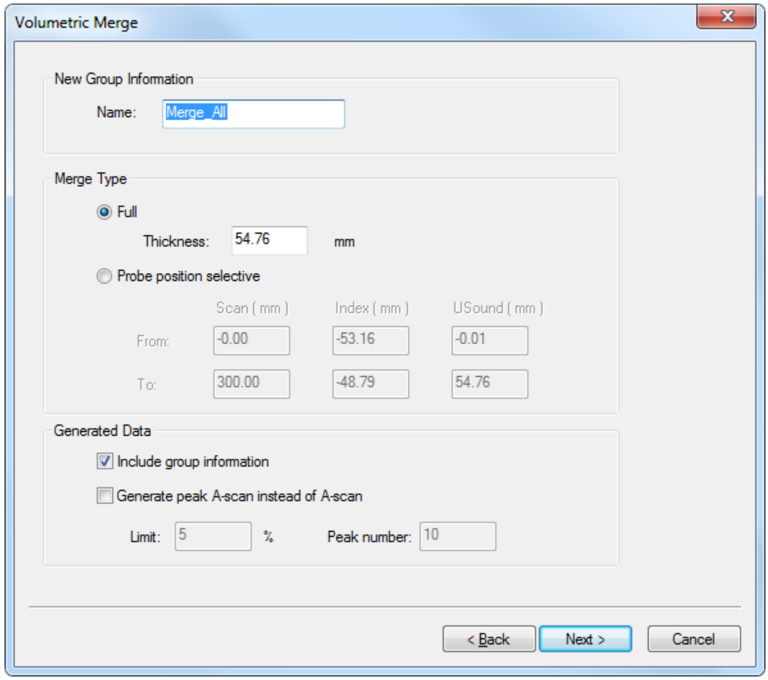


Figure 7-5 The New Group Information, Merge Type, and Generated Data group boxes

6. Referring to (see Figure 7-6 on page 340), perform the following tasks:
   a) Under **Merge Dimension**, set the merge dimensions for the three axes. The **Memory required** parameter displays the corresponding required memory to process and store the data.
   b) If needed, click **Reset to Defaults** to reset the limits of the inspected volume for the merged data group to the **Overall Dimensions**, and the resolution to the nominal value.
   c) Click **Next**.

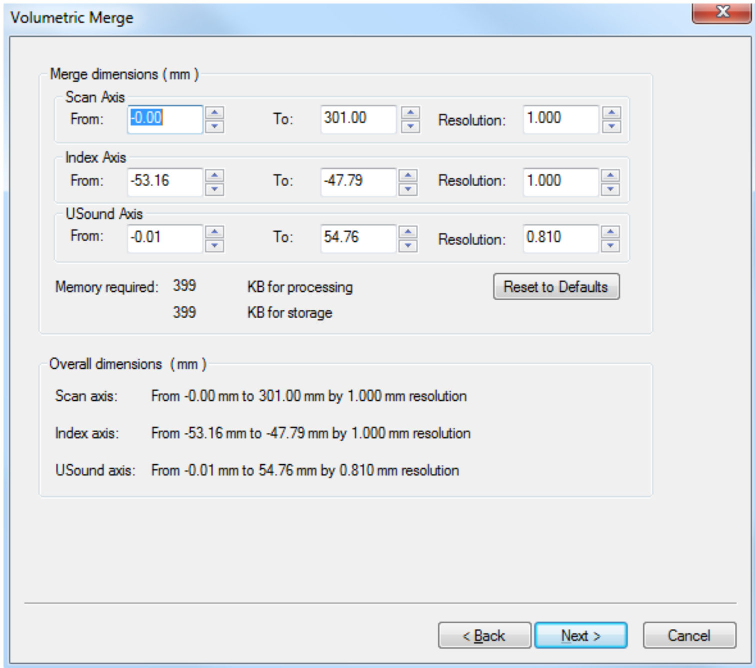


Figure 7-6 The Merge dimensions and Overall dimensions group boxes

7. Referring to (see Figure 7-7 on page 341), perform the following tasks:
   a) Consult the values in the **Data Information** and **Volume Information** sections to validate the volumetric merge parameters.
   b) Click **Back** if you need to adjust parameters in the previous steps.
   c) If you want to save your volumetric merge configuration to a file (.vmc), click **Save to File**, and then specify the folder and file name for the file.
   d) Click **Finish** to perform the volumetric merge.
   The merged group is added to the contents (see Figure 7-7 on page 341 and Figure 7-8 on page 341).

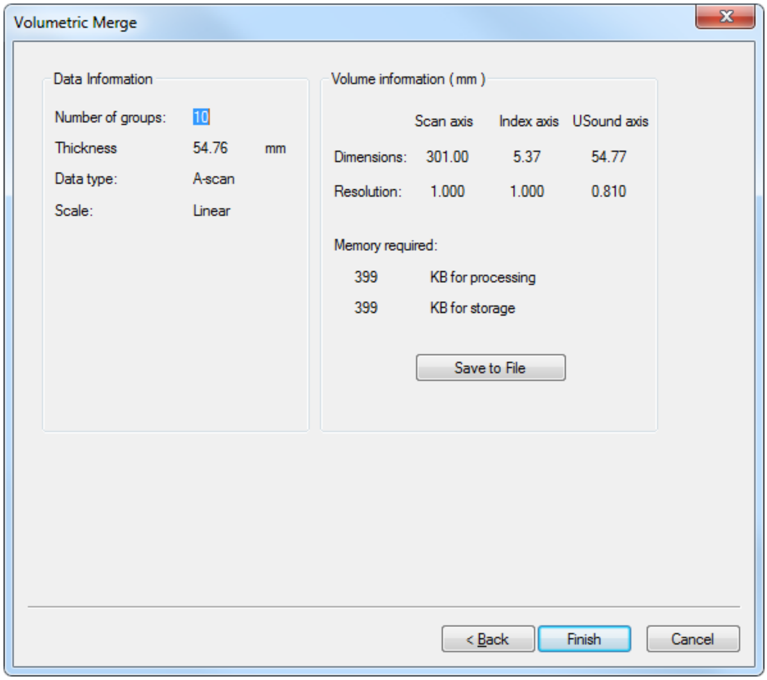


Figure 7-7 The Data information and Volume information group boxes

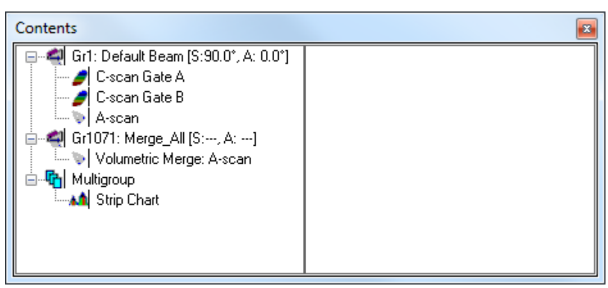


Figure 7-8 Example of the volumetric merge group created

7.2.2 Performing an Automatic Volumetric Merge

The **Automatic Volumetric Merge** function performs the volumetric merge of all beams in the currently selected file using the default values of the **Volumetric Merge** dialog box (see section 7.2.1 on page 336). The merge process transforms the sectorial group data into bitmap images of end-view slices.

To perform the volumetric merge

1. Open the file on which you wish to perform a volumetric merge.
2. On the **Processing** menu, select **Automatic Volumetric Merge**.

   TomoView processes the data and displays the new data on the screen.

7.2.3 Performing an Automatic Volumetric Merge by Group

The **Automatic Volumetric Merge by Group** function performs the volumetric merge of all beams for each group using the default values of the **Volumetric Merge** dialog box (see section 7.2.1 on page 336) and creates one merged data group per group.

To perform the volumetric merge by group

1. Open the file on which you wish to perform a volumetric merge.
2. On the **Processing** menu, select **Automatic Volumetric Merge by Group**.
   The merged data groups are added to the view **Contents** list.

7.3 Customizing Readings

Readings can be customized in Analysis mode in the same way as in the Setup mode. For details on how to customize readings, see section 4.10 on page 276.

7.4 Working with the Amplitude Drop Sizing Utility

You can use TomoView to display the various readings related to the active data file and the data views (see section 7.3 on page 342). With user-defined flaw sizing settings, you can use some of these parameters to get information on the position and the size of a flaw indication.

7.4.1 Defining or Modifying Flaw Sizing Settings

The **View Information** parameters, related to indication sizing, allow for –X dB amplitude drop sizing of a flaw indication in its three dimensions simultaneously (scan, index, and ultrasonic axes). The value of X can be defined independently for each axis.

To define or modify the flaw sizing settings

1. On the menu, select **File > Preferences**.
2. In the **Flaw sizing settings** group box of the **Preferences** dialog box, click **General Settings** tab (see section 4.10 on page 276).
   a) In the **Amplitude drop (-X dB)** group box, enter the -X value for each axis independently (only negative values are accepted).
   b) In the **Peak holding algorithm** group box, enter the number of samples to be used for the peak holding algorithm for each axis independently.
   The default value is 0, but if for a given axis a value *n* larger than 0 is entered, the sizing algorithm, along this axis, "ignores" an amplitude drop below -X dB that is smaller than *n* samples. This tool can be used to group several flaws, or to avoid the problems related to the use of nonsmoothed A-scan signals.
   c) If the check boxes are selected in the **Fill acquisition gaps** group box, the sizing algorithm ignores the missing acquisition points or lines along the considered axis by interpolating between the adjacent valid acquisition points or lines.

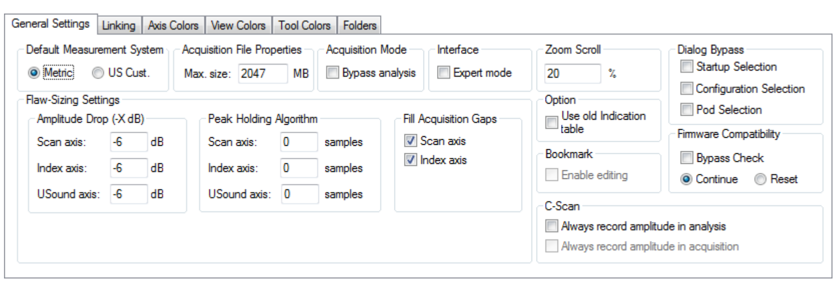


Figure 7-9 General Settings tab

NOTE

Applying the peak holding algorithm, for a given axis, can oversize the indication by 2×n samples for this axis.

7.4.2 Determining the Location and Dimensions of Flaw Indications

The **View Information** parameters can be used not only to determine the dimensions of a flaw indication using –X dB amplitude drop sizing, but also to determine the location of the flaw indication in the inspected volume. The following procedure uses the **Zone** tool of the TomoView software.

To determine the location and dimensions of a flaw indication in 3-D

1. Create a suitable display layout, showing at least the Top (C), Side (B), or the End (D) views.
2. In one of the displayed views display the parameters (as shown in Table 14 on page 344).

Table 14 Parameters for flaw indication

Parameter	Description
Z. Max. Ampl.	Zone maximum amplitude
Z. Max. Scan Pos.	Zone maximum amplitude position on the scan axis
Z. Max. Index Pos.	Zone maximum amplitude position on the index axis
Z. Max. UT Path Pos.	Zone maximum amplitude position on the ultrasonic axis
Z. Scan1 –X dB	Position 1 along the scan axis of “maximum amplitude –X dB” position
Z. Scan2 –X dB	Position 2 along the scan axis of “maximum amplitude –X dB” position

Table 14 Parameters for flaw indication *(continued)*

Parameter	Description
Z. Index1 –X dB	Position 1 along the index axis of "maximum amplitude –X dB" position
Z. Index2 –X dB	Position 2 along the index axis of "maximum amplitude –X dB" position
Z. USound1 –X dB	Position 1 along the ultrasonic axis of "maximum amplitude –X dB" position
Z. USound2 –X dB	Position 2 along the ultrasonic axis of "maximum amplitude –X dB" position
Z. Scan2 – Scan1 –X dB	Length of the flaw indication at –X dB along the scan axis
Z. Index2 – Index1 –X dB	Length of the flaw indication at –X dB along the index axis
Z. USound2 – USound1 –X dB	Length of the flaw indication at –X dB along the ultrasonic axis

These readings can be found in the **Statistics > Zone** category of the **Information Groups** dialog box (see Figure 7-10 on page 346). Note that the Expert mode must be activated for these readings to be available (see section 3.16 on page 220).

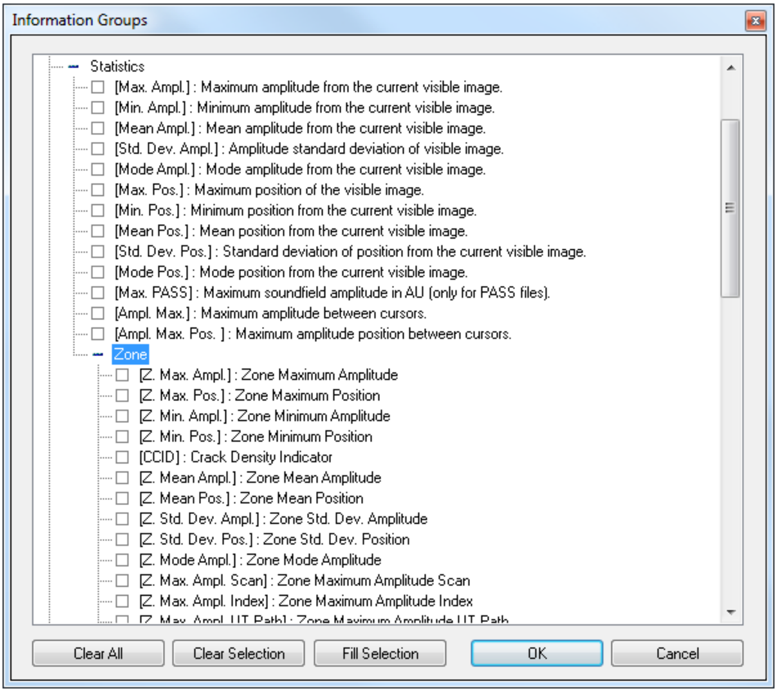


Figure 7-10 Statistics parameter category of the Information Groups dialog box

3. Visualize the region of interest in the considered recorded data (see Figure 7-11 on page 347).

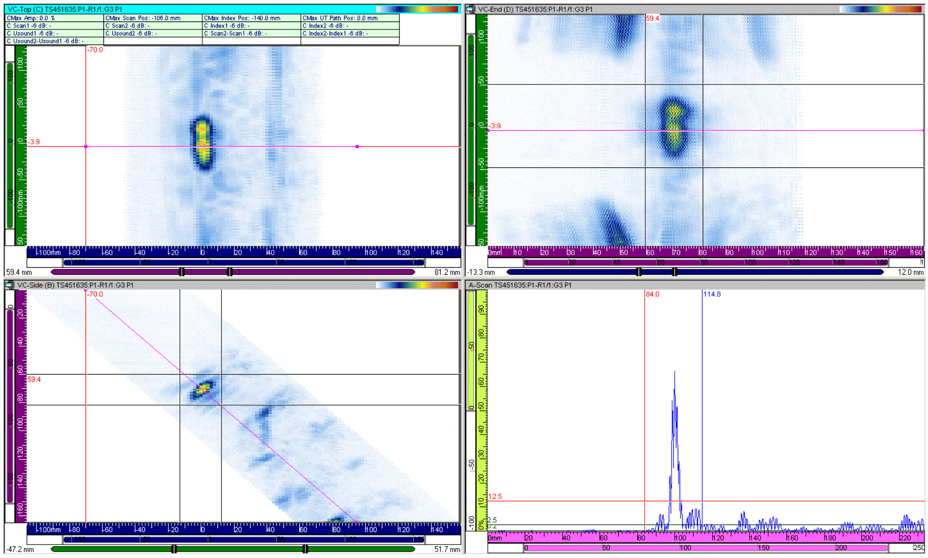


Figure 7-11 Data display examples

4. Position the gate selectors for the ultrasonic axis so that the flaw indication to be sized is located between the gate selectors.
5. Create a zone in the Side (B) view [End (D) view] containing the considered flaw indication.
6. Create a zone in the Top (C) view containing the considered flaw indication.
7. The zone in the Side (B) or End (D) view is automatically adjusted to the gate selector limits. In fact, you have now created a volumetric zone defined by the contour in the Top (C) view and the gate selectors in the ultrasonic axis (see Figure 7-12 on page 348). The parameters in the **Information Groups** dialog box give the maximum amplitude of the flaw indication, its position along each axis, and its size along each axis, with the user-defined amplitude drop value.

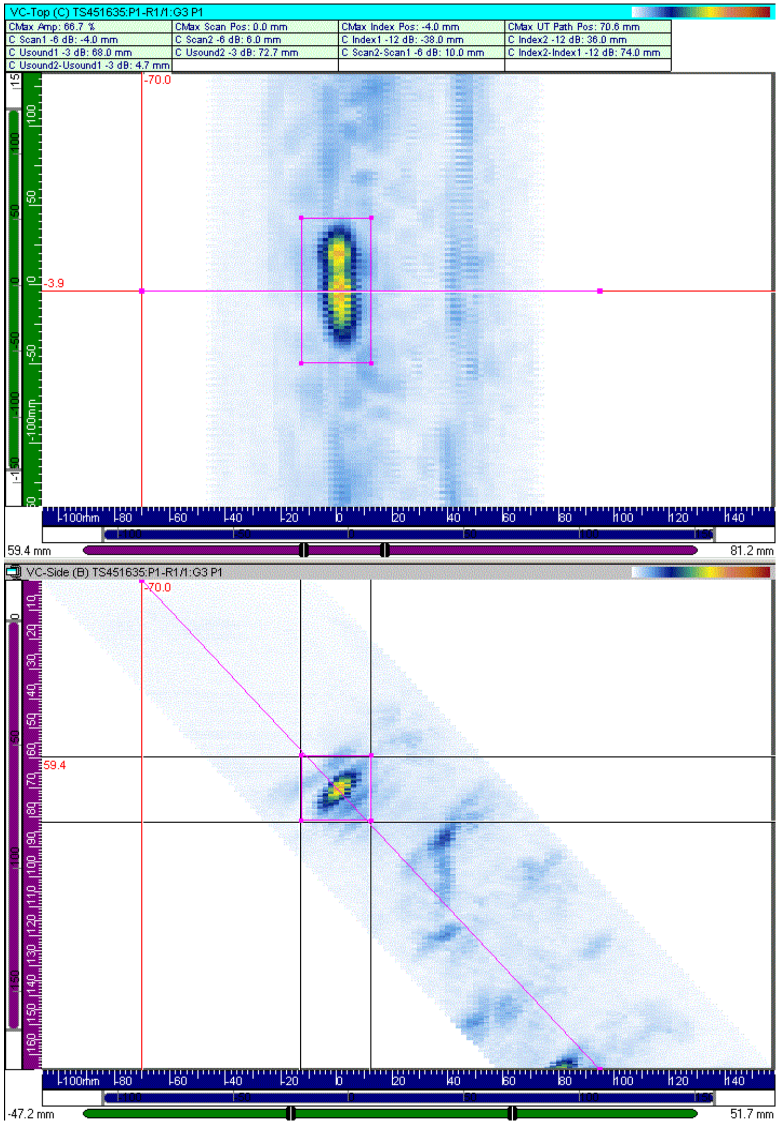


Figure 7-12 Examples of flaw indication sizing with the zone tool

7.5 Working with the Gain Information Component

The **Gain Information** dialog bar (see Figure 7-13 on page 349) is used to monitor the different gain settings, and to adjust the software gain applied to the signal and/or the color palette settings in either setup or analysis mode.

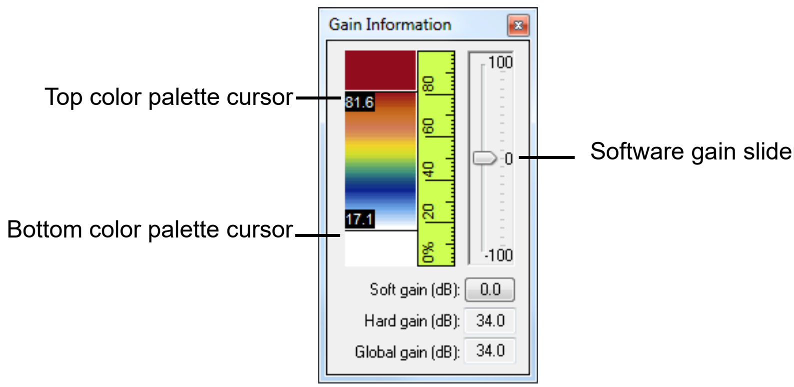


Figure 7-13 The Gain dialog bar

The **Gain** dialog bar displays the current color palette. Double-clicking between the top and bottom cursors reduces the current palette range by a factor of two. Double-clicking outside the cursors restores the default full-palette range.

Dragging the top or bottom color palette cursors, respectively, sets the upper and lower limits of the color palette range in percentage (linear data), or in decibels (logarithmic data). Dragging the color palette itself sets the lower and upper limits of the color palette range.

Dragging the slider, or using the arrow keys, sets the software gain, which ranges from –100 dB to +100 dB. The software gain slider sets a software-calculated amplitude gain that is applied to both the displayed images and the amplitude values in the readings, but leaves the raw acquisition data unchanged.

When applying negative software gain to ultrasonic data, pixels over 100 % FSH keep applicable color code for saturation and the corresponding information view keeps returning 100 % for the signal amplitude.

To add or remove software gain

1. Select the view for which you want to modify the gain information.
2. On the **Main** toolbar, click the **Gain Information** button ( ).
   The **Gain** dialog bar appears (see 7.14 on page 400).

NOTE

The **Hard Gain** information box indicates the applied hardware gain. This information can be modified in the is defined in the **UT Settings > General** dialog bar.

---

3. To modify the software gain:
   - Move the software gain slider cursor up or down using the mouse.
   - Click in the **Gain information** dialog box and use the up and down keyboard arrow keys to increase or decrease the software gain in 0.1 dB increments.
   - Click above or below the slider cursor ( ) to increase or decrease the software gain in 6 dB increments.
   - Click on the **Soft Gain (dB)** button to sete the software gain.
   - On the color palette, double-click between the cursors to divide the current palette range by two, or outside the cursors to restore the default full palette range.

7.6 Working with the TOFD Manager Component

The TOFD Manager is a complete tool for time-of-flight diffraction (TOFD) inspections that performs both basic and advanced analysis of TOFD examination data. Using the TOFD Manager, you can perform online and offline TOFD calibrations with either conventional or phased array probes.

You can also use the TOFD Manager to perform advanced analysis using typical TOFD cursors, algorithms for both lateral wave synchronization (straightening) and lateral wave removal, and a synthetic aperture focusing technique (SAFT) algorithm. All processing algorithms are developed to leave the raw inspection data unaffected. However, all processed data can be saved in separate data groups in the attributes file (.A01).

You can open the TOFD Manager from the Component toolbar by clicking the TOFD Manager button ( ) or from the menu by selecting **Components > TOFD Manager Window**. The **TOFD Manager** dialog box is shown in Figure 7-14 on page 351.

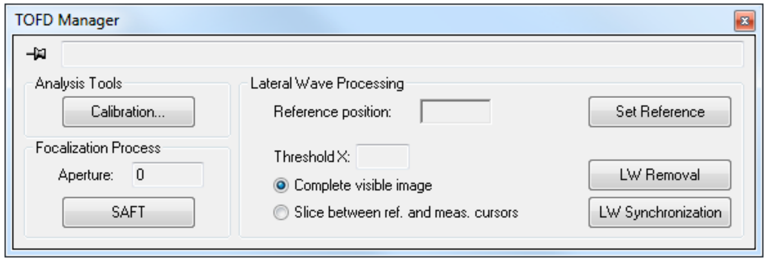


Figure 7-14 The TOFD Manager dialog box

In the **Information Groups** dialog box, when the expert mode is active, a **TOFD** group of reading is available (see Figure 7-15 on page 351) for fast and easy length and through-wall sizing of indications in combination with the **Indication Table** (see section 6.2.1 on page 324).

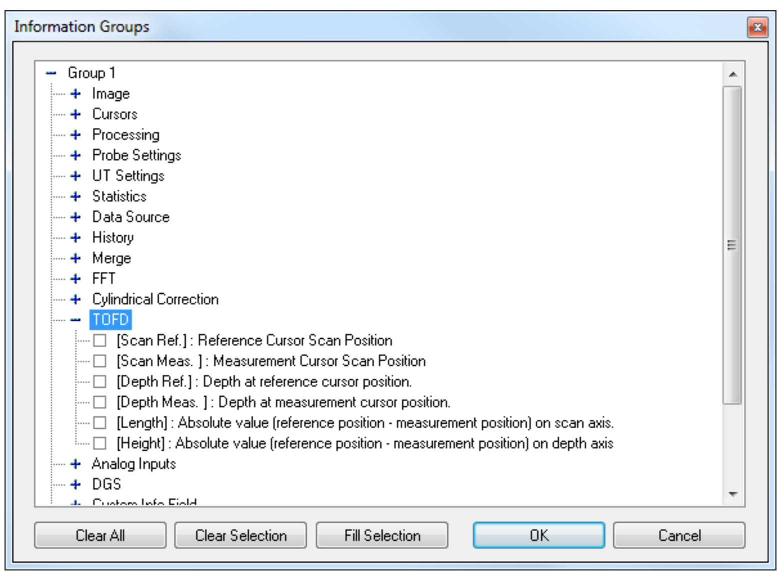


Figure 7-15 The TOFD readings in the Information Groups dialog box

NOTE

The TOFD Manager is not available in the Lite Aero and Lite Weld editions.

7.6.1 Description of the TOFD Manager Dialog Box

This section describes the **TOFD Manager** dialog box (see Figure 7-16 on page 352).

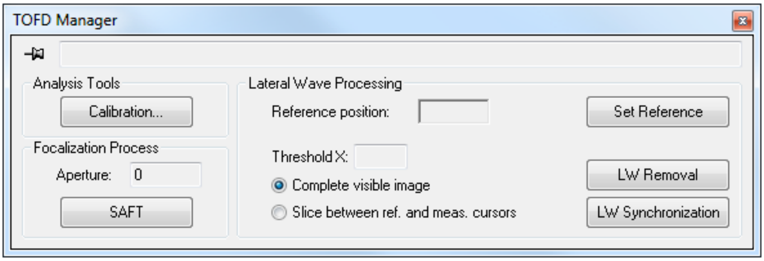


Figure 7-16 The TOFD Manager dialog box

The **TOFD Manager** dialog box (see Figure 7-16 on page 352) contains the following group boxes:

Analysis Tools

Specifies parameters for the calibration of the ultrasonic axis (depth) for TOFD data.

Focalization Process

Allows the operator to use the SAFT algorithm on TOFD data.

Lateral Wave Processing

Specifies parameters for lateral wave processing (synchronization and removal).

7.6.1.1 Analysis Tools Group Box

The **Analysis tools** group box contains the following buttons:

Calibration

Opens the **TOFD** dialog box in order to perform TOFD calibration (see Figure 7-17 on page 353).

TOFD dialog box

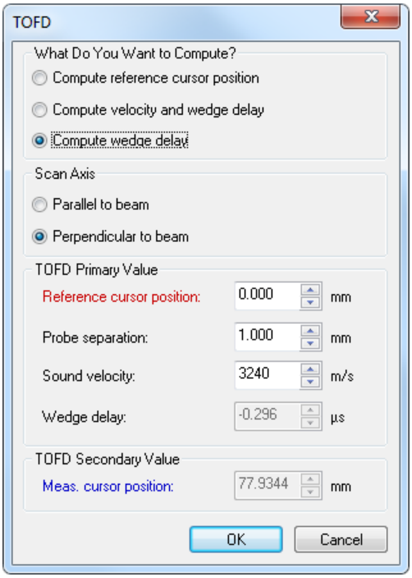


Figure 7-17 TOFD dialog box

The **TOFD** dialog box contains text boxes (on a white background) for the values that can be changed, or display boxes (appearing dimmed) for the values that are calculated from the other values. The dialog box contains the following elements:

What Do You Want To Compute?

This group box encloses option buttons used to select the parameter that is to be calculated from the other parameters.

Compute reference cursor position

This option is used to adapt the TOFD ruler to set its zero position at the reference cursor position. Position the reference cursor on the first echo signal, and then click **OK**.

Compute velocity and wedge delay

This option is used to calculate both the material sound velocity and the probe wedge delay, by using two reference signals, typically the lateral wave and the back-wall echo.

Compute wedge delay

This option is used to calculate the probe wedge delay, by using a fixed value of the sound velocity and one reference signal, typically the lateral wave or the back-wall echo.

Scan Axis

This group box encloses option buttons used to select the scan axis orientation relative to the ultrasonic beam projection: **Parallel to beam** or **Perpendicular to beam**.

Reference cursor position

This box defines the real (true depth) reference cursor position in millimeters (mm) or inches (in.).

Probe separation

This box defines the distance between the probe exit points in millimeters (mm) or inches (in.). This distance is always measured along the component surface.

Sound velocity

This box defines the material sound velocity in meters per second (m/s) or inches per microsecond (in./µs).

Wedge delay

This box defines the probe wedge delay in microseconds (µs).

7.6.1.2 Lateral Wave Processing Group Box

The **Lateral wave processing** group box (see Figure 7-16 on page 352) contains the following options:

Reference position

This box indicates the position, on the scan axis, of the selected reference A-scan.

Set Reference

This button sets the reference A-scan defined by the reference cursor position.

Complete visible image

When this option is selected, the processing is done on the complete image.

Slice between ref. and meas. cursors

When this option is selected, the lateral wave synchronization, or lateral wave removal, is performed on the A-scans enclosed between the reference and the measurement cursors on the displayed Side (B) view.

LW Removal

This button performs the lateral wave (LW) removal process on the data group in the active view, and creates a new data group containing the processed data.

LW Synchronization

This button performs the lateral wave synchronization process on the data group in the active view, and creates a new data group containing the processed data.

7.6.1.3 Focalization Process Group Box

The **Focalization process** group box contains the following elements:

Aperture size

This box indicates the aperture of the SAFT algorithm, which is the total number of A-scans used to calculate the processed A-scan. The value is always odd, and is determined by the position of the cursors on the active Side (B) view.

SAFT

This button performs the SAFT (synthetic aperture focusing technique) algorithm.

7.6.2 Performing TOFD Calibration in Analysis Mode

This section provides a description on how to use the TOFD calibration of the TOFD Manager component in Analysis mode. For information on how to perform TOFD calibration in setup mode, see section 4.5 on page 251.

To use the TOFD calibration function in Analysis mode

1. Open a data file (.rdt).
2. Visualize a **Side (B)** view.
3. Position the reference cursor at the start of the lateral wave signal.
4. Position the measurement cursor at the start of the back-wall signal.
5. On the **View Properties > Units** dialog bar, in the **Type** list box of the **USound** axis, select **TOFD**.
6. On the **Component** toolbar, click the TOFD Manager button ().

The **TOFD Manager** dialog box appears (see Figure 7-16 on page 352).

7. In the **TOFD Manager** dialog box, select the appropriate options, and then enter the appropriate values (see section 7.6.1 on page 352).

NOTE

If you select the **Compute velocity and wedge delay** option button in the **What do you want to compute?** group box of the **TOFD** dialog box, the measurement cursor has to be positioned on a second reference signal (for example, lateral wave).

8. Click **OK**.

   The ultrasonic axis is now calibrated in Analysis mode.

7.6.3 Performing Data Processing of TOFD Files

IMPORTANT

Although the functionality and mathematical correctness of all processing algorithms have been thoroughly checked and validated, no guarantees in terms of flaw detection and sizing performances can be given. The performance enhancement provided by a processing algorithm is definitely related to the specificity of each application, and is therefore the responsibility of the user.

7.6.3.1 Lateral Wave Synchronization

This section describes how to synchronize the lateral wave of a TOFD group.

IMPORTANT

Calibrate the TOFD group prior to applying the lateral wave synchronization algorithm.

To synchronize the lateral wave of a TOFD group

1. Open a data file (.rdt).

2. Select the **Side (B)** view on which you want the lateral wave synchronization to be applied.
3. In the **Side (B)** view, position the reference (red) cursor to select a reference A-scan in an unflawed zone (see Figure 7-18 on page 357).
4. In the **Side (B)** view, position the ultrasonic reference cursor and the ultrasonic measurement (blue) cursor on either side of the lateral wave signal (see Figure 7-18 on page 357). Appropriate setting of the cursors determines the result of the operation.

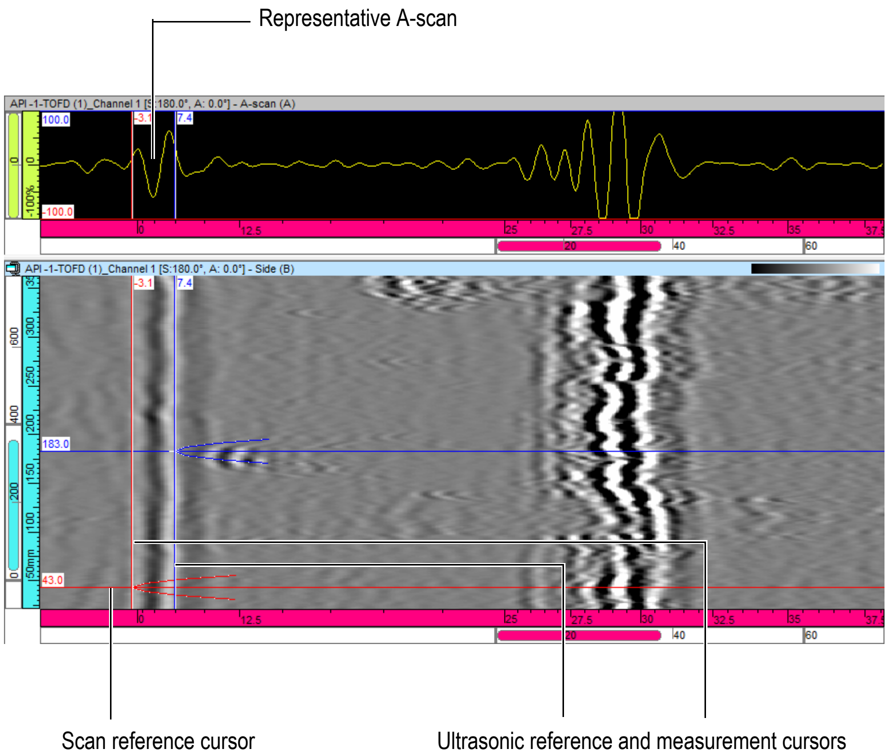


Figure 7-18 Selection of a reference A-scan

5. In the **TOFD Manager** dialog box, in the **Lateral Wave Processing** group box, click **Set Reference**.

By default, the lateral wave synchronization is applied to the **Complete visible image**, but alternatively, the **Slice between Ref. and Meas. cursors** option can be selected if appropriate for the considered TOFD data.

6. Click **LW Synchronization**.
7. In the menu, select **View > Contents** and display the newly created **LWS**. The lateral wave is now synchronized (see Figure 7-19 on page 358).

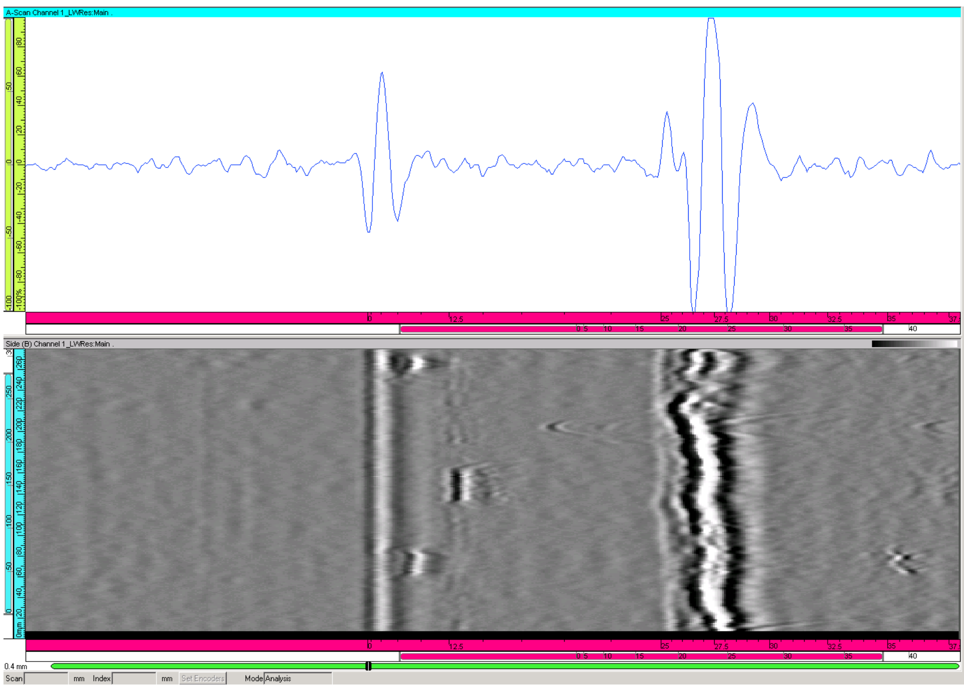


Figure 7-19 TOFD data after lateral wave synchronization

7.6.3.2 Lateral Wave Removal

This section describes how to remove the lateral wave of a TOFD group.

IMPORTANT

In order to obtain usable results from the lateral wave removal algorithm, it is mandatory to apply the algorithm to previously synchronized data.

To remove the lateral wave of a TOFD group

1. In the menu, select **View > Content** and display the lateral wave synchronized data.
2. In the **Side (B)** view, position the scan reference (red) cursor to select a reference A-scan in an unflawed zone (see Figure 7-20 on page 360). If the selection of the reference A-scan is correctly performed prior to the lateral wave synchronization, this operation does not need to be repeated.
3. In the **Side (B)** view, position the ultrasonic reference cursor (red) and the ultrasonic measurement cursor (blue) on either side of the lateral wave (see Figure 7-20 on page 360). If the lateral wave signal shows a lot of variation, it is important to position the cursors at least around the first oscillation of the lateral signal.

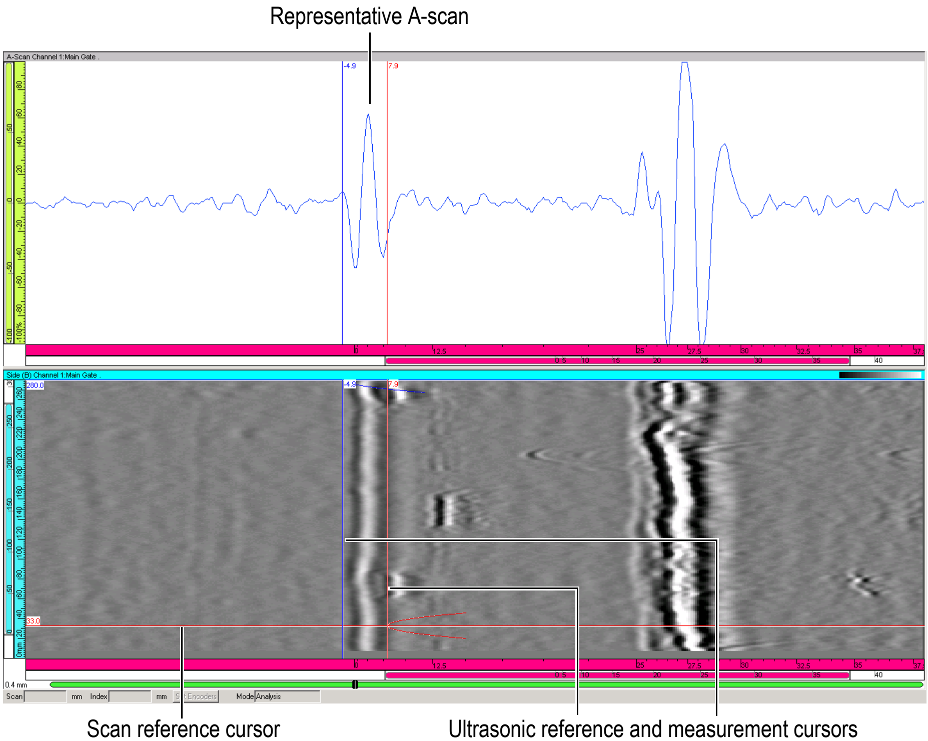


Figure 7-20 Selection of reference A-scan

4. In the **TOFD Manager** dialog box, in the **Lateral wave processing** group box, click **Set Reference**.

   By default, the lateral wave removal is applied to the **Complete visible image**, but alternatively the **Slice between Ref. and Meas. cursors** option can be selected if appropriate for the considered TOFD data.

5. Click **LW Removal**.
6. In the menu, select **View > Contents** and display the newly created LWR. The lateral wave is now removed (see Figure 7-21 on page 361).

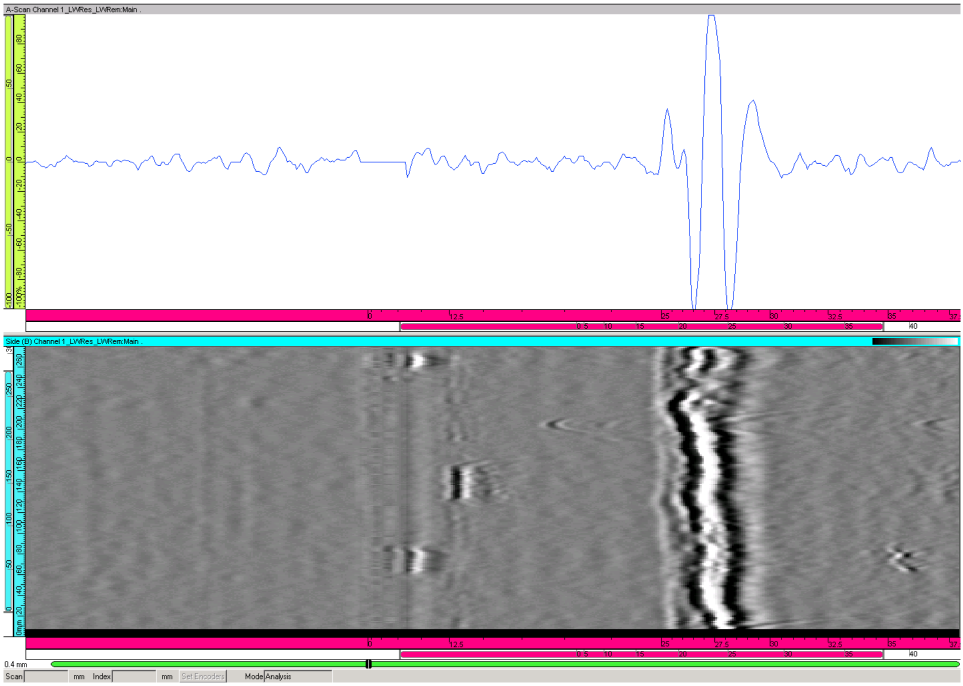


Figure 7-21 TOFD data after lateral wave removal

7.6.3.3 SAFT Processing

The synthetic aperture focusing technique (SAFT) is a computer enhanced imaging technique for the detection and characterization of discontinuities. The technique takes advantage of the nonlinear phase shift of a reflection as a discontinuity is linearly scanned. Improved lateral resolution and a higher signal-to-noise ratio are achieved by using this phase shift, mathematically simulating the focusing of an ultrasonic lens that is focused on every point in an inspected part.

IMPORTANT

The SAFT algorithm can be applied either to raw TOFD data or to processed TOFD data. It is the responsibility of the data analyst to select the best combination for a given application.

To apply the SAFT algorithm

1. Select the **Side (B)** view on which you want to apply the SAFT algorithm.
2. In this **Side (B)** view, perform the following procedure in order to select the adequate aperture to be used for the SAFT algorithm:
   a) Zoom in on a zone of interest containing a punctual TOFD indication (see Figure 7-22 on page 362), showing the typical parabolic shape.
   b) Position the reference and measurement cursors on either side of the flight-time minimum of the indication, so that the indication remains clearly visible at the selected scan axis positions; the SAFT algorithm tends to smoothen out and eventually erase indications if too large an aperture is used.

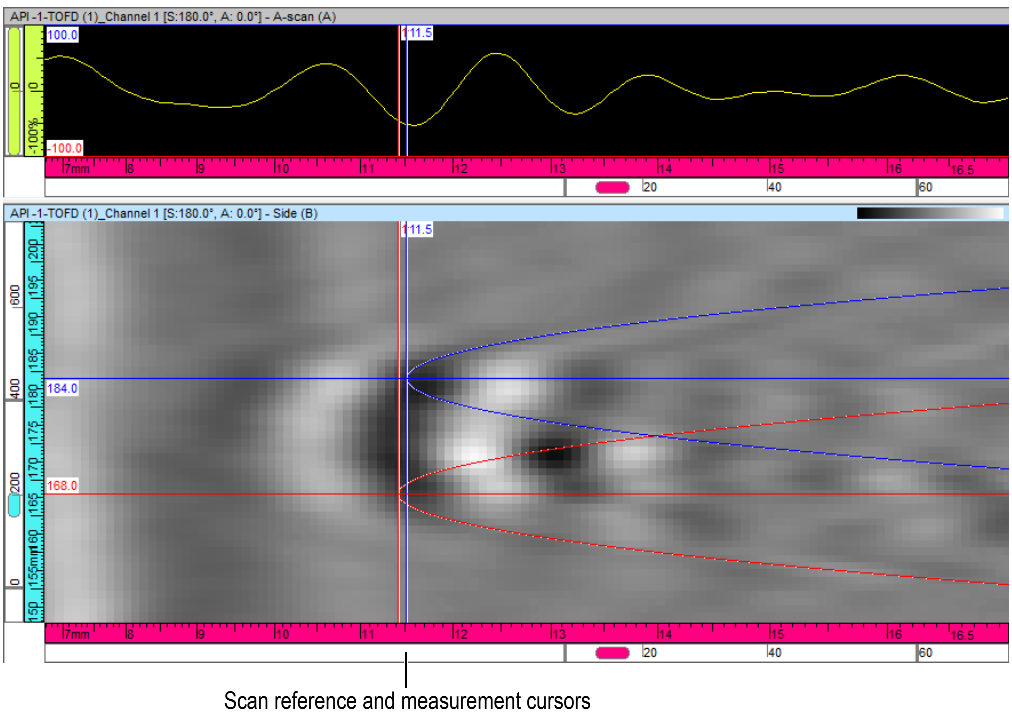


Figure 7-22 Definition of the aperture

3. The SAFT aperture as defined by the positions of the cursors is displayed in the **TOFD Manager** dialog box, in the **Focalization process** group box.

4. In the **TOFD Manager** dialog box, in the **Focalization process** group box, click **SAFT**.
5. In the menu, select **View > Contents** and display the newly created SAFT. The SAFT algorithm has been applied (see Figure 7-23 on page 363).

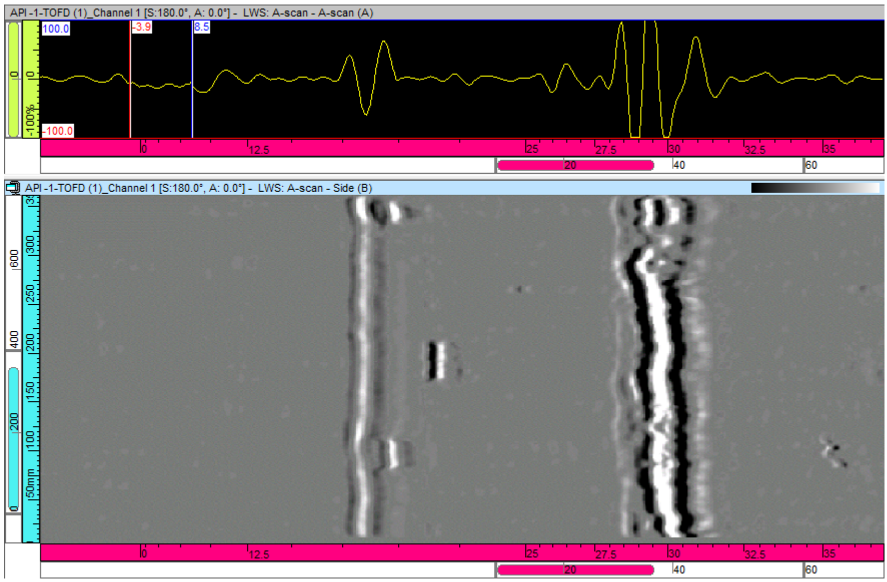


Figure 7-23 Overview of a SAFT ultrasonic image

As the SAFT algorithm involves averaging, which might tend to smoothen out high amplitudes, the use of a higher **Software Gain** value could be helpful to optimize the visualization of the processed data.

7.7 Working with the C-Scan Merge Process

In the Analysis mode, you can use the C-scan merge component to merge C-scan data acquired with various groups and/or beams.

The merge process compares the minimum amplitude, the maximum amplitude, or the minimum position obtained at each point of the inspected part acquired by the considered groups and/or beams. It then creates a new data group with the above mentioned criterion. TomoView also saves the created merged data in the Attributes file (.A01).

With the C-scan merge component, you can:

- Merge multiple C-scan images from different data files (see Figure 7-24 on page 364).
- Merge C-scan images from different gates (even if you modified the gate position and length). Can be used for multi-layers part with bonding (see Figure 7-25 on page 365).
- Merge Amplitude C-scan and preserve maximum amplitude (for indication detection) or minimum amplitude (for back-wall attenuation monitoring).
- Merge minimum position C-scan.

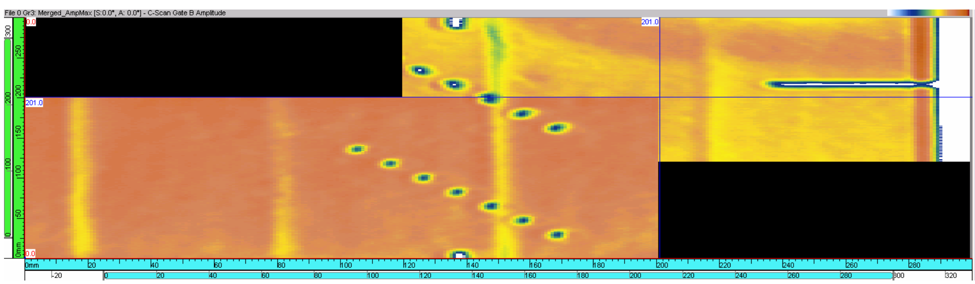


Figure 7-24 Two files merge example

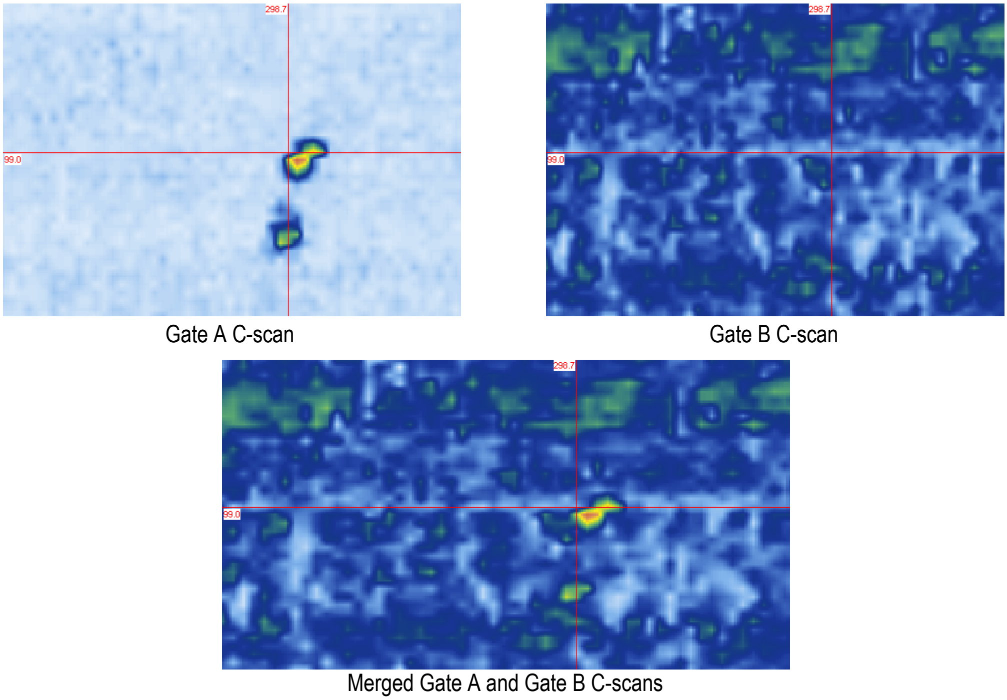


Figure 7-25 Gate A and Gate B C-scan merging example

To merge C-scan data

1. Open the file containing the data that you want to merge.
2. On the menu, select **Processing > C-Scan Merge**.
3. In the **C-Scan Merge** dialog box (see Figure 7-26 on page 366):
   a) In the **Data Selection** area, select the data that you want to preserve in the merge.
   b) In the **Additional Options** area, select **Merge associated data** to simultaneously create the associated amplitude or position data.
   c) In the **Group Selection** area, in the **Available groups** list, hold down the SHIFT key and click to select the groups that you want to merge, and then click **Add**.
   The selected groups appear in the **Selected groups** list.
   d) Click **Next**.

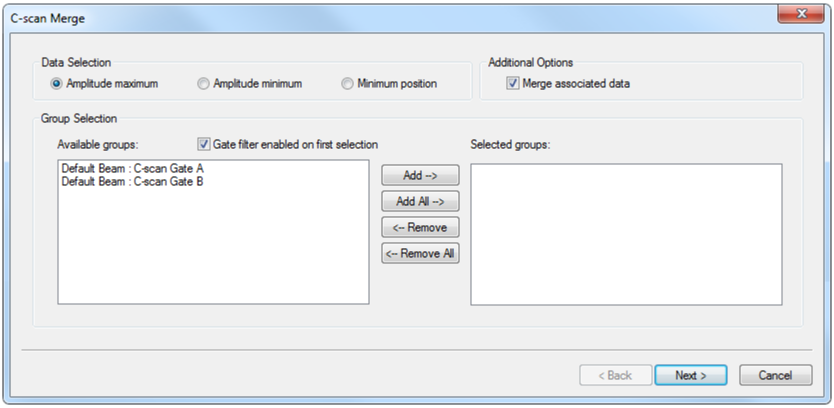


Figure 7-26 Example of the C-Scan Merge dialog box (first step)

4. In the second step of the **C-Scan Merge** dialog box (see Figure 7-27 on page 367), under **Merge Dimensions**:
   a) Use the parameters in the **Scan Axis** and **Index Axis** areas to set the merge dimensions.
   b) In the **Destination** area, type the desired **Group name** for the merged data.
   c) Click **Finish**.
   The merged group is added to the contents (see Figure 7-27 on page 367).

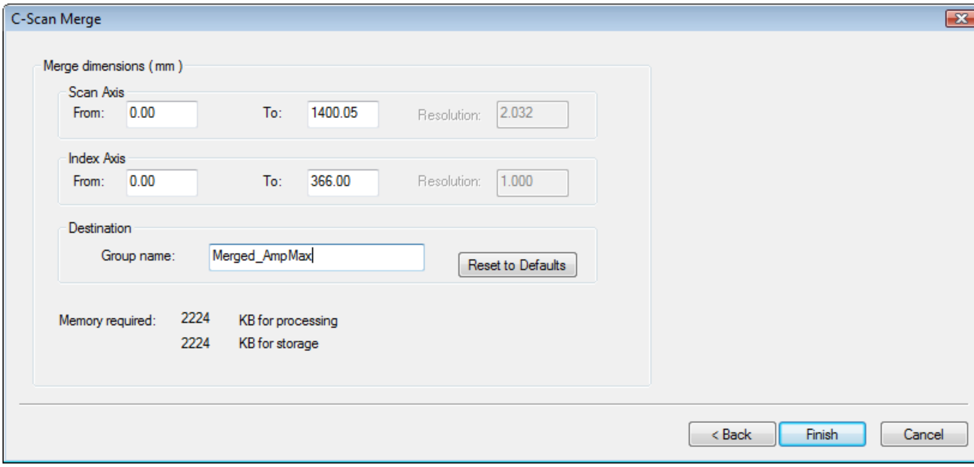


Figure 7-27 Example of the C-Scan Merge dialog box (second step)

5. To display the merged group (see Figure 7-28 on page 367):
   a) Select the view where you want to display the merged C-scan.
   b) On the menu, click **View > Contents**.
   c) In the **Contents** dialog box, in the list on the left, select the created C-scan merged group.
   d) In the list on the right, double-click the corresponding view to display. The selected data appears in the view.
   e) Close the **Contents** dialog box.

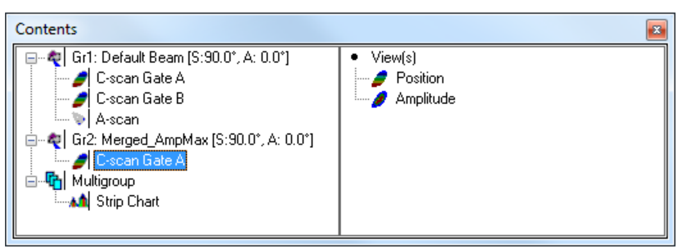


Figure 7-28 Example of the C-scan merge group created

7.8 Working with the SNR Analysis Component

TomoView includes a signal-to-noise ratio (SNR) function. You can use the **SNR Analysis Utility** to evaluate the noise level in a reference area on a C-scan view and then to calculate the indication surface area above the noise.

The SNR function is available in Analysis mode and with the Lite Aero edition. The SNR function is used for 0-degree inspections, often for aerospace industry parts. You can use the function on Top (C), amplitude, and position C-scan views.

7.8.1 Calculating the Area of an Indication Using SNR Analysis Utility

To calculate the area of an indication using the SNR Analysis Utility

1. Start the SNR analysis utility using one of the following methods:
   - On the Component toolbar, click the SNR Analysis Utility button ( ).
     OR
     On the menu, select **Processing > SNR Analysis Window**.
2. On the Component toolbar, select the Zone tool ( ), and then draw a rectangle over a zone representing a noise reference area (see the example shown in Figure 7-29 on page 368).

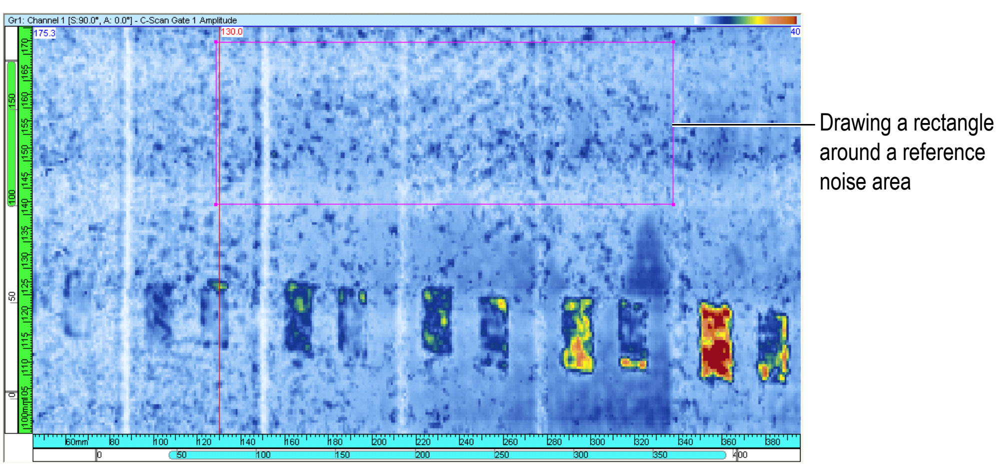


Figure 7-29 Selecting the noise reference area with the Zone tool

3. In the **SNR Analysis Utility** dialog box, click **Retrieve**.
   TomoView calculates the mean and the standard deviation value for the reference area, and displays a distribution chart of the number of pixels as a function of the signal amplitude (see Figure 7-30 on page 369).

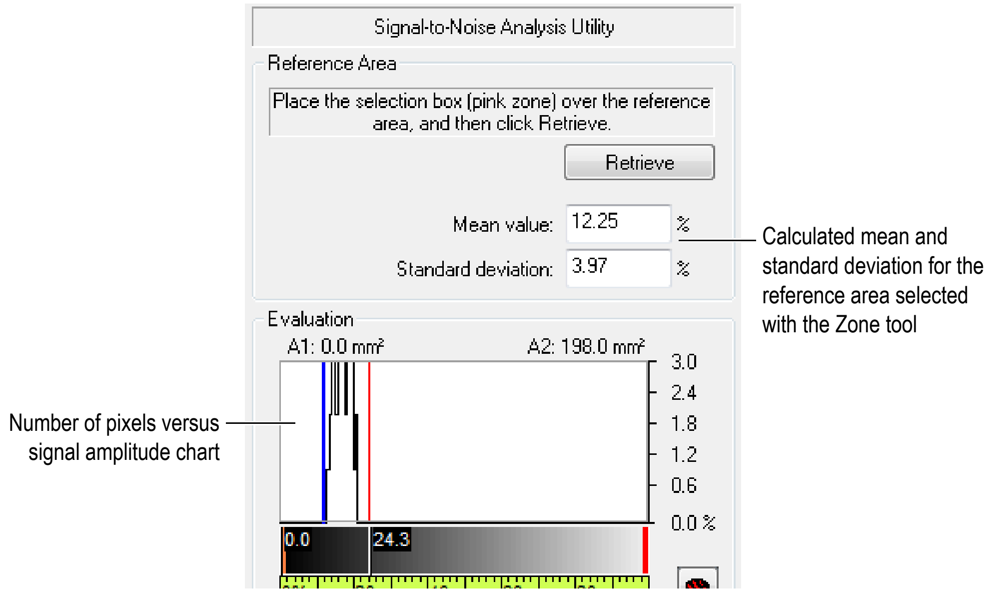


Figure 7-30 Example of reference area analysis results

4. Set the color palette to the gray scale to enhance highlighting of the defect area:
   a) Right-click on the title bar of a Top (C), amplitude, or position C-scan view.
   b) In the contextual menu select **Set Color Palette > Gray levels**.
   c) In the contextual menu select **Edit Color Palette** and in the **Limits** group box, select the < = 0 % color to orange and the > = 0 % color to red.
5. On the C-scan view, move the reference cursor and the measurement cursors around the indication as in the example shown in Figure 7-31 on page 370.

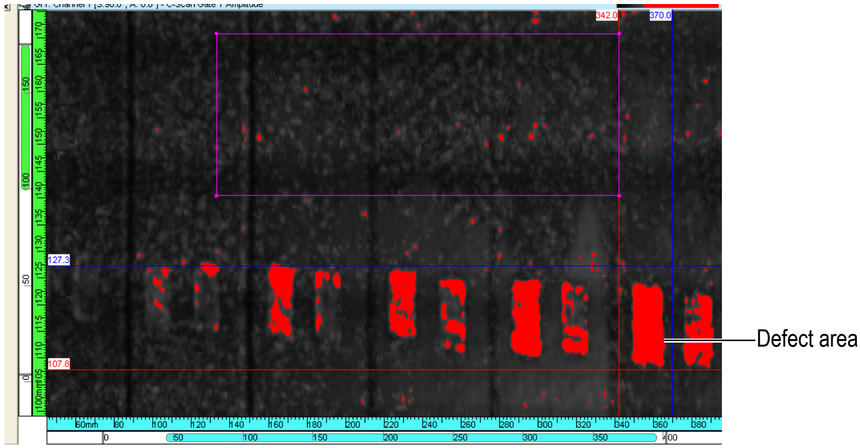


Figure 7-31 Positioning the cursors around the defect area

6. In the **SNR Analysis Utility** dialog box (see Figure 7-32 on page 371), the SNR value is directly related to the **K** parameter slider value:
   a) Move the **K** parameter slider to set the desired K or SNR value.
   The **K** value is often a final user requirement.
   Decreasing the value of **K**, increases the number of low contrasted indications in red color.
   b) Click  to update the C-scan view to the new **K** parameter value.
   The defect area value represents the set of pixels being above the reference noise area level.

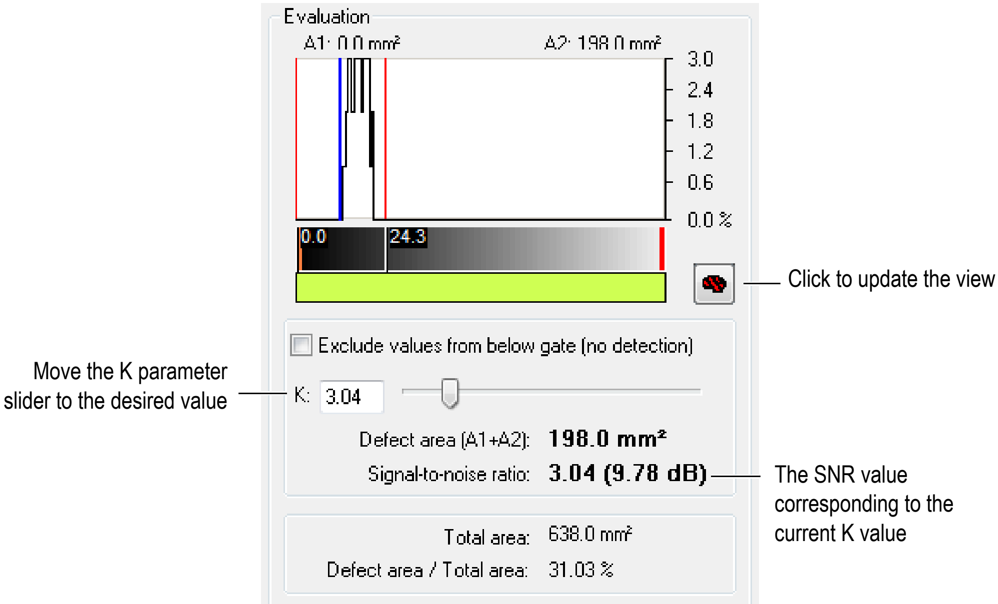


Figure 7-32 Adjusting the K value

In the example shown in Figure 7-33 on page 372:

- K = 3.04 corresponds to an SNR value equal to 9.78 dB.
- The defect area is equal to 198 mm² which represents 31.03 % of the total area defined between the reference and measurement cursors.

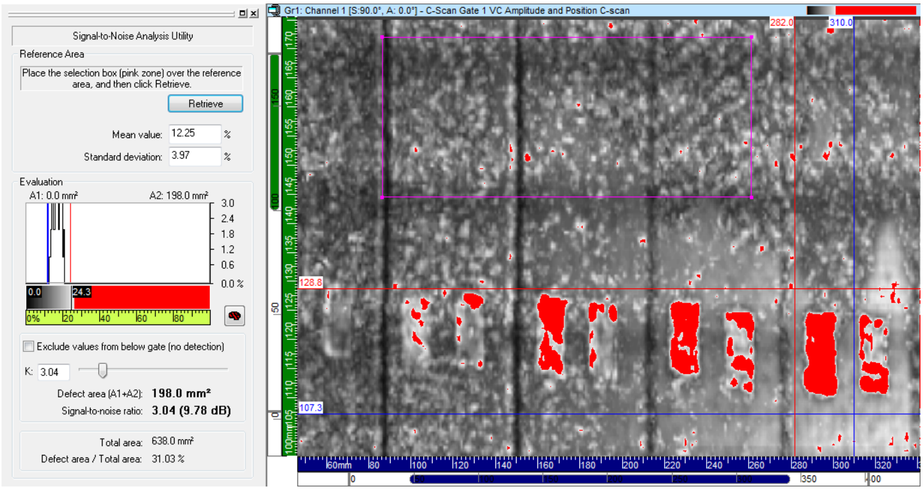


Figure 7-33 Example of the SNR analysis with K equal to 3.04

In the example shown in Figure 7-34 on page 373:

- K = 8.76 corresponds to an SNR value equal to 18.85 dB.
- The defect area is equal to 97.5 mm² which represents 15.28 % of the total area defined between the reference and measurement cursors.

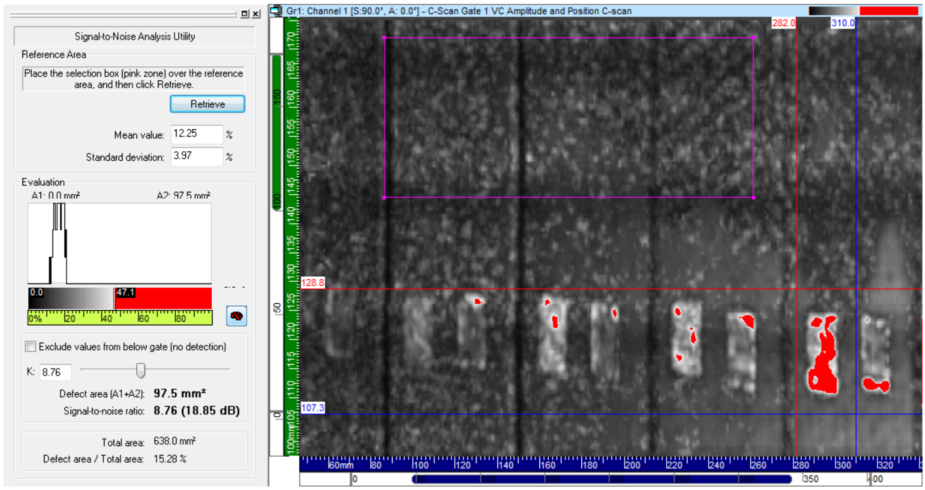


Figure 7-34 Example of the SNR analysis with K equal to 8.76

7.8.2 SNR Analysis Conventions

The parameters involved in the SNR calculations are presented in Table 15 on page 373 and are illustrated in Figure 7-35 on page 374.

Table 15 SNR parameters

Parameter	Symbol
Mean value inside reference area	m
Standard deviation inside reference area	σ
Lower Threshold	S_-
Upper Threshold	S_+
Lower Area (area of amplitude below S_-)	$A_- = A1$
Upper Area (area of amplitude above S_+)	$A_+ = A2$
Adjustable factor	K

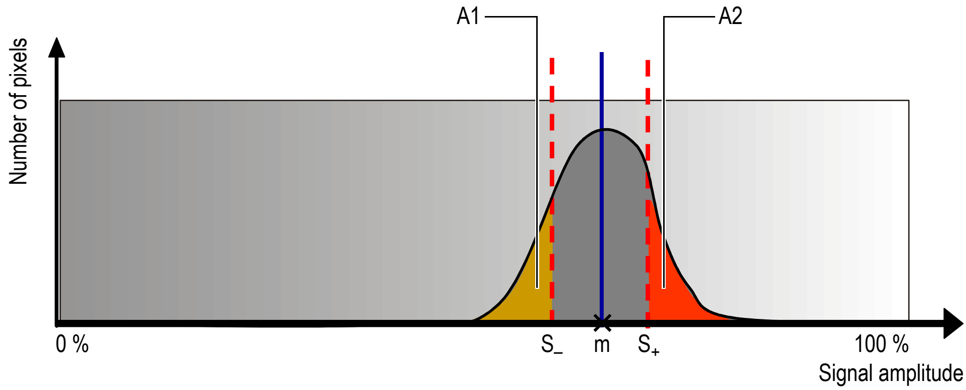


Figure 7-35 The SNR distribution

The mathematical expression for the SNR is:

$$\mathrm{SNR} = 20 \times \log(K)$$

where:

$$S_+ = m + K \times \sigma$$

$$S_- = m - K \times \sigma$$

The defect area value represents the set of pixels with values above the SNR ratio. The mathematical expression for the defect area is:

$$\text{Defect Area} = A_+ + A_-$$

When the **K** slider is at the maximum left position:

- K = 0
- The defect SNR is not applicable.
- The defect area is equal to the total area between the reference and measurement cursor positions.

When the **K** slider is at the maximum right position:

$$K = \max\left[\log\left(\frac{|A_{max} - m|}{\sigma}\right), \log\left(\frac{|m|}{\sigma}\right)\right]$$

where:

- A_{max} = 100 % (for an amplitude C-scan)
- A_{max} = End of gate position (for a TOF C-scan)

$$\text{Defect SNR} = 20 \times \max\left[\log\left(\frac{|A_{max} - m|}{\sigma}\right), \log\left(\frac{|m|}{\sigma}\right)\right]$$

- Defect Area = 0

7.9 Working with the Soft C-Scan Process

When working in analysis mode, you can use the **Soft C-Scan** module to create new C-scan data from previously recorded A-scan data. This process creates both position and amplitude C-scan data groups.

The TomoView software saves the created C-scan data with the **Attributes** files (.A01). Thus, if deemed necessary, the previously created C-scan data is immediately available upon reopening the data file (.rdt).

To create a soft C-scan from previously recorded A-scan data

1. Open a data file.
2. Position the reference and measurement cursors on a view to define the gate location and level.
3. On the menu, select **Processing > Create Soft C-Scan** to open the **Create Soft C-Scan** dialog box.

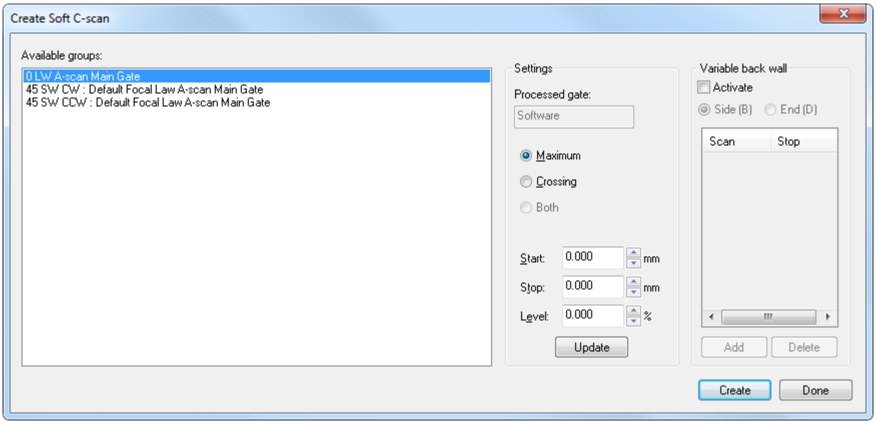


Figure 7-36 The Create Soft C-Scan dialog box

4. In the **Available** group box, select the groups for the soft C-Scan operation.
5. In the **Settings** group box, select the type of position information to be taken into account when creating the soft C-scan data group (**Peak**, **Crossing,** or **Both**).
6. In the **Gate location** command group, if necessary:
7. In the **Settings** group box, if necessary, define the **Start, Length**, and **Level** (threshold) gates to generate the soft C-Scan data group.

Variable Back Wall Soft C-Scan

It is possible to edit the A-scan range on which the soft C-Scan processing is performed to using the **Create soft C-scan** dialog box. This is particularly useful when performing analysis on complex geometries like nozzles or other parts with changing thicknesses.

To create a variable back wall soft C-scan

1. Repeat steps 1 to 7 described in the **To create a soft C-scan from previously recorded A-scan data** section.
2. In the **Create Soft C-scan** dialog box:
   a) In the **Variable Back Wall** group box, select the **Activate** group box.
   b) Select if you want to create the soft C-Scan from the Side (B) or End (D) scan.
   c) Create the points to be used for the software C-scan creation:

◆ Position the Reference and Measurement cursors on the view and click **Add**.
OR
Click **Add** and define the **Scan/Index** and **End** values.

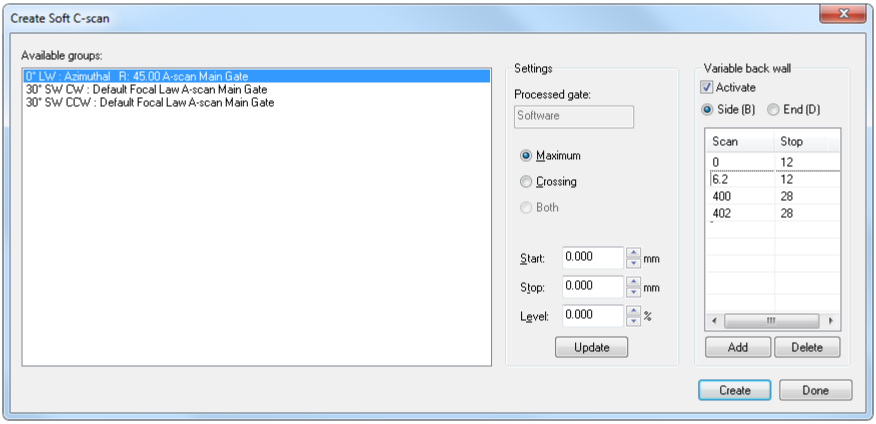


Figure 7-37 The Create Soft C-Scan with the Variable back wall activated

d) Click **Create**.

The points which were created will then appear on the Side (B) or End (D) scan.

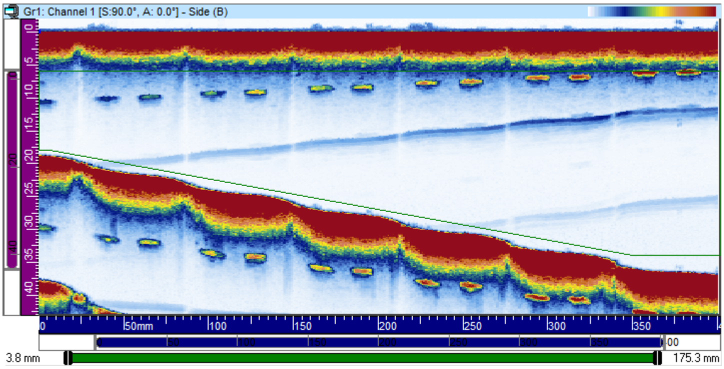


Figure 7-38 The B-Scan with a displayed variable back wall

3. Click **Done**.
4. In the menu, select **View > Contents** and add the software C-scan data to the display.

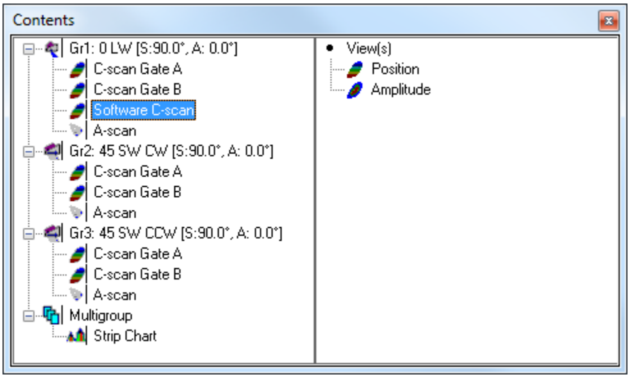


Figure 7-39 The Contents dialog box

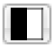
NOTE

You can turn On/Off the display of the points used for the software C-scan creation with the **Soft C-scan** check box in the **Overlay** tab of the **View Properties** dialog bar.

7.10 Working with the Binarizer Component

The Binarizer can be used to perform simple pass or fail analysis on data files by converting acquisition data to a color driven Good/No good display. The Binarizer is particularly useful when inspecting composite parts and can be used on uncorrected VC amplitude and position C-scan data.

To binarize a C-Scan using the Binarizer Component

1. In the **Components** toolbar, click  (Binarizer) to open the **Binarizer**. See Figure 7-40 on page 379.

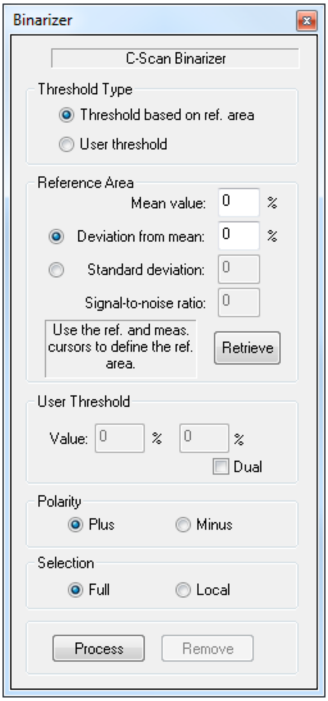


Figure 7-40 The Binarizer dialog box

2. In the **Threshold Type** section (see Figure 7-41 on page 379), select the criteria for the Binarization:
   a) Select **Threshold based on ref. area** to use a reference area for the noise level and continue to step 3.
   b) Select **User threshold** to manually define the binarization thresholds and continue to step 5.

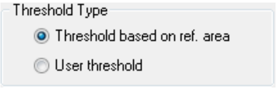


Figure 7-41 The Threshold Type group box

3. Place the red and blue cursors (by double-clicking with the left and right mouse buttons) in order to draw a rectangle over a zone which will be used as the reference area. See Figure 7-42 on page 380.

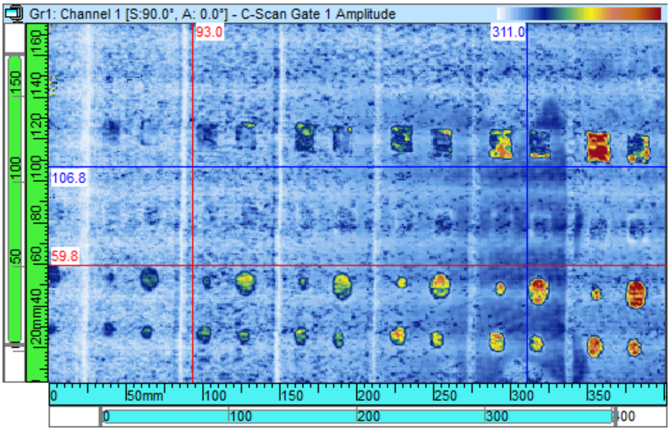


Figure 7-42 The Binarizer cursors

4. In the **Binarizer** dialog box, click on the **Retrieve** button to automatically calculate the mean and standard deviation of the noise for the reference area. The **User Threshold** will also be adjusted using either one of the two following options where the sign (+ or –) depends on whether the Plus or Minus polarity was selected.

   a) If **Deviation from mean** was selected then the following calculation will be performed:

   User threshold = Mean value ± Deviation from mean

   b) If **Standard Deviation** was selected then the following calculation will be performed:

   User threshold = Mean value ± Standard Deviation x SNR ratio

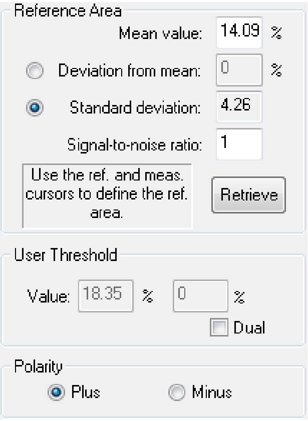


Figure 7-43 The Reference Area dialog box

5. Select how the binarization will be performed using the following parameters:
   a) If the **Dual** checkbox is unchecked and the **Polarity** is set to *Plus*, the Binarizer will sort the data based on the following threshold (see Figure 7-43 on page 381):

   User threshold = Mean value + [Selected deviation]

   b) If the **Dual** checkbox is unchecked and the **Polarity** (see Figure 7-44 on page 381) is set to *Minus* the Binarizer will sort the data based on the following threshold:

   User threshold = Mean value -[Selected deviation]

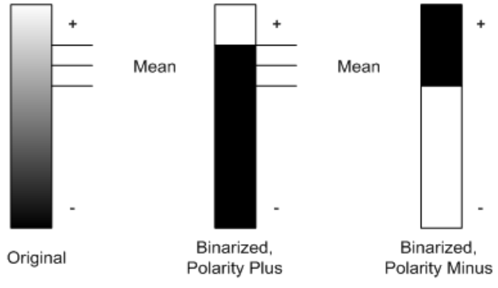


Figure 7-44 The Binarizer polarity

c) If the **Dual** checkbox is checked, two different threshold values can be defined and the Binarizer will sort the data based on the values which are between or outside the two Thresholds. See Figure 7-45 on page 382.

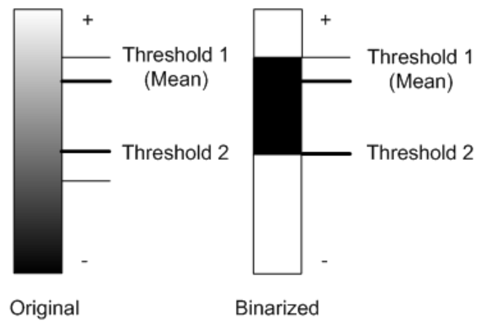


Figure 7-45 The Binarizer thresholds

6. In the **Selection** group box, select **Full** to process the full C-Scan or **Local** to only binarized the area defined by the Reference and Measurement cursors. Different areas can be processed and added to the final C-Scan individually.
7. Click **Process.**
8. In the menu, select **View > Contents** and add the Binarized data to the display. See Figure 7-46 on page 382 and Figure 7-47 on page 383.

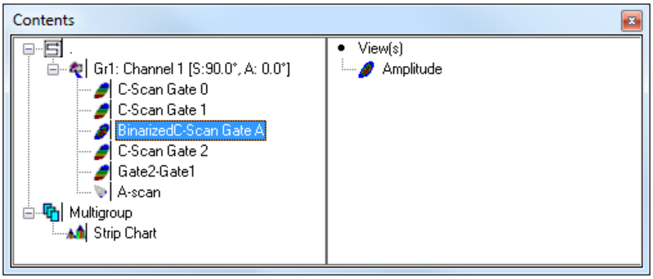


Figure 7-46 The Binarizer Contents dialog box

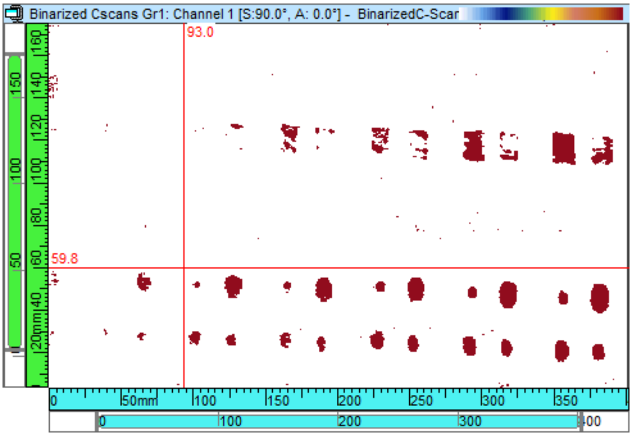


Figure 7-47 The Binarized data display

9. To remove parts of the binarized C-Scan select the area using the Reference and Measurement cursors and click **Remove**.

7.11 Working with Color Palettes

Depending on the application, it can be very helpful to modify the color palette associated with the specific view types in order to make certain types of indications easier to see. This section shows how the TomoView color palette can be edited.

7.11.1 Modifying Color Palettes

To modify a color palette

1. To open the **Palette Editor** dialog box (see Figure 7-48 on page 384):
   - In the **View Properties > Palette** dialog bar, click.

     OR

     Right-click on the active view color palette and select **Edit Color Palette**.
2. If the **Pane Properties** dialog box is not presently open, click **Properties** on the **Pane** menu.
3. Click the **Palette** tab.

4. Click **Edit Palette**.

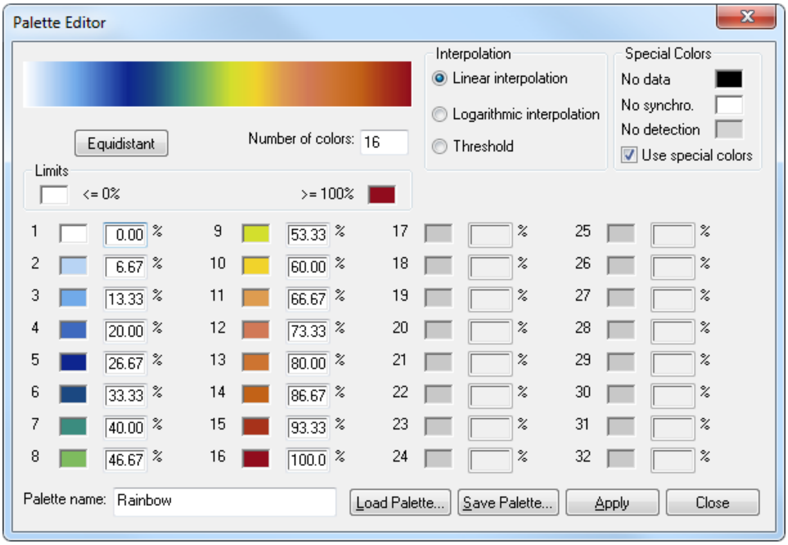


Figure 7-48 The Palette Editor dialog box

5. In the **Palette Editor:**
   a) Click **Load Palette** if you want to open an existing palette.
   b) Click **Equidistant** if you want to distribute the thresholds evenly between the colors.
   c) Enter the desired number of colors in the palette in the **Number of colors** box. The number of active boxes will increase or decrease accordingly.
   d) In the **Interpolation** group box, select the desired interpolation type from the colors.
   e) Select the **Use special colors** check box and select colors to be associated with the **No data, No synchro**, and **No detection** boxes if you want to have these colors highlighted on the display.
   - **No data** corresponds to locations where no data was acquired because of missed encoder steps, or because the location was outside the inspected volume.
   - **No synchro.** corresponds to locations where no signal crossed gate I (synchronization gate). This color is effective only for synchronization setup on echo.

- Click the **No detection** box to define the color assigned to locations where no signal crossed the considered gate.

f) To change a color, click on it and select an alternative color using the **Color** dialog box (see Figure 7-49 on page 385).

g) Click **Save Palette** if you want to save the current palette to a **.col** file.

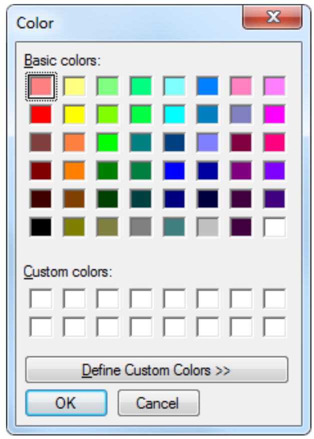


Figure 7-49 The Color dialog box

NOTE

In setup mode, **Special colors** functions are not applied to the Scrolling B-scan view, Scrolling Strip View (Amp), and Scrolling Strip View (Pos).

6. Click **Apply**, then **Close**.

7.11.2 Optimizing a Color Palette for Corrosion Visualization

Corrosion inspection is a good example of an application where an appropriate color palette configuration can be very useful. This section describes the steps to optimize a color palette for corrosion.

To optimize a color palette for corrosion visualization

1. Select the C-scan view for which you want to visualize the corrosion.
2. In the **Palette Editor**:
   a) Set **Number of colors** to **3**.
   b) In the **Interpolation** group box, select **Threshold**.
   c) Deselect the **Use special colors** check box.
   d) In the **Limits** group box, set the < = **0 %** color to grey and the > = **0 %** color to red.
   e) Set **color 1** to green, **color 2** to yellow, **color 3** to orange, and **color 4** to red.
   f) Click Save Palette and save the current color palette to a **.col** file.

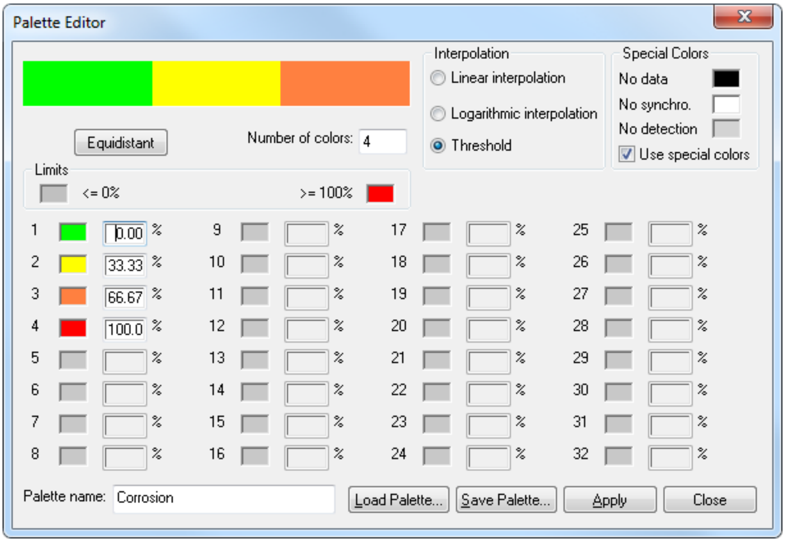


Figure 7-50 Color palette optimized for corrosion visualization

3. In the **View properties > Palette** dialog bar, select the **Reverse color order** check box.
4. Click  to open the **Gain Information** dialog bar.
   a) Adjust the top and bottom color palette cursors to match, respectively, the maximum and minimum corrosion thicknesses of your specimen (see the example in Figure 7-52 on page 387)

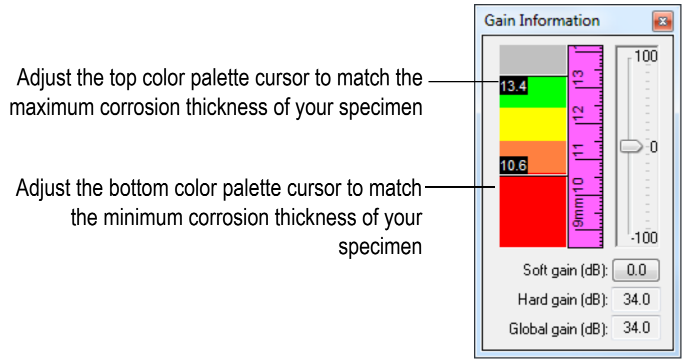


Figure 7-51 Example of palette limits adjusted to specimen corrosion limits

The C-scan pane view should now highlight the corrosion of the specimen, such as in the example in Figure 7-52 on page 387.

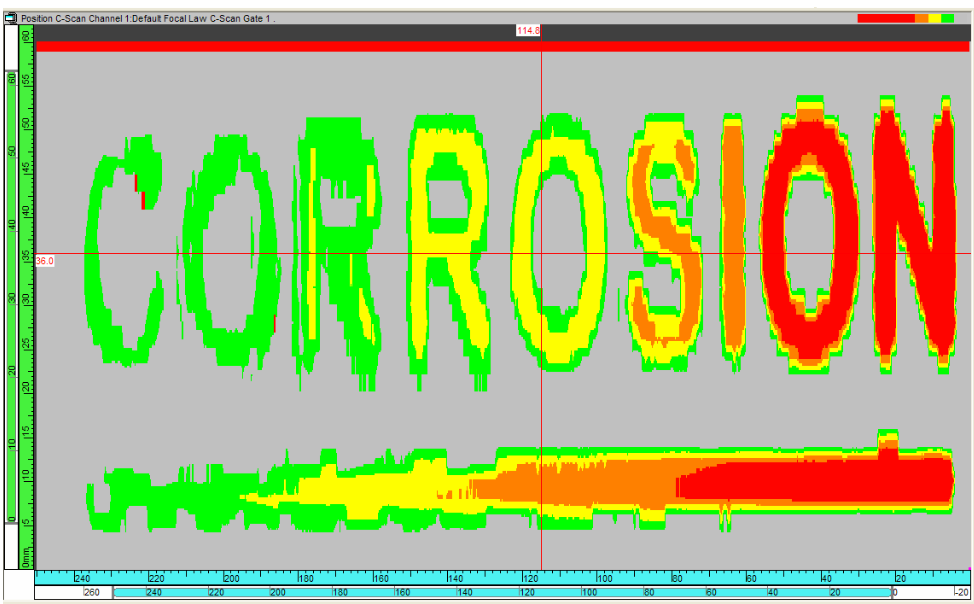


Figure 7-52 Example of corrosion visualization on a demonstration specimen

7.12 Working with the Image Analysis Tools

This section provides information on the cursor and zone tools as well as on the display of rebound and overlays.

TomoView offers powerful tools to facilitate advanced data analysis. Table 16 on page 388 provides a list of these image analysis tools along with their associated buttons (toggled to change the visibility of the tools) and their descriptions.

Table 16 Summary of image analysis tools

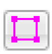
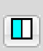
Icon	Name	Function
	Zoom tool	To select and zoom in a specific region on a view.
	Segment tool	To make 3-D measurements on a C-scan view by clicking and dragging. Also works on Top/Side/End views.
	Zone tool	To select a region on a top, side, end, or C-scan view by clicking and dragging. **Tip**: When the Zone tool is not selected, press and hold the CTRL key, and then click and drag on a view to perform the same task.
	Move tool	To move the graphical data relative to the view frame.
	Split view horizontally in two	To divide the active view into two views with the same horizontal dimension.
	Split view vertically in two	To divide the active view into two views with the same vertical dimension.
	Split view in four	To divide the active view into four views with the same horizontal and vertical dimensions.
	Delete view	To delete the active view.
	Empty view	To remove the contents of the active view.

Table 16 Summary of image analysis tools *(continued)*

Icon	Name	Function
	Maximize view	To maximize the size of the active view by hiding the other views. Clicking this button when the view is already maximized restores the view to its original size and displays the other views.
	Edit view properties	To open the **View Properties** dialog box, used to configure the parameters of the active view.
	Edit view contents	To open the **Contents** dialog box, used to select the data type and the view type to be displayed in the active view.
	Toggle echo dynamics curves	To toggle the display of the echo dynamics curve on the active view. This button is available only for top, side, end, and C-scan views.
	Toggle view information group 1	To toggle the display of information group 1 in the active view (reference cursor parameters by default).
	Toggle view information group 2	To toggle the display of information group 2 in the active view (measurement cursor parameters by default).
	Toggle view information group 3	To toggle the display of information group 3 in the active view (by default, parameters related to the combination of the reference and measurement cursors).
	Toggle view information group 4	To toggle the display of information group 4 in the active view (3-D cursor, Zone tool, and acquisition parameters by default).
	Hide view information groups	To hide all information groups displayed in the active view.
	Edit view information groups contents	To open the **Information Groups** dialog box, used to edit information group contents, that is, the parameter sets displayed in the information groups. You can also double-click a reading to open this dialog box.

Table 16 Summary of image analysis tools *(continued)*

Icon	Name	Function
	Envelope	Toggles the activation of the envelope mode for the online and off-line A-scan view.
	Rectify data	To apply a software rectification to unrectified signal data.
	Fit image to view	To display the complete data contents of the active view.
	Fit inspection domain to view	To display the complete inspection domain in the active view.
	Toggle reference cursor	To toggle the display of the active view reference cursors.
	Toggle measurement cursor	To toggle the display of the active view measurement cursors.
	Add a note	To add a callout on top of a view to point out and comment on a given element. You can freely move and resize the callout. The callout notes are included in screen captures but cannot be saved.
	US Customary units	In TomoView Lite and TomoViewer editions only, changes length measurement units to US Customary when pressed.

As shown in the example in Figure 7-53 on page 391, the reference and measurement cursors appear on the various views as horizontal and vertical colored lines on each of the three axes (index, scan, and ultrasonic) to mark a specific data point. The Zone tool appears as a pink rectangle and the 3-D Cursor appears as a pink line.

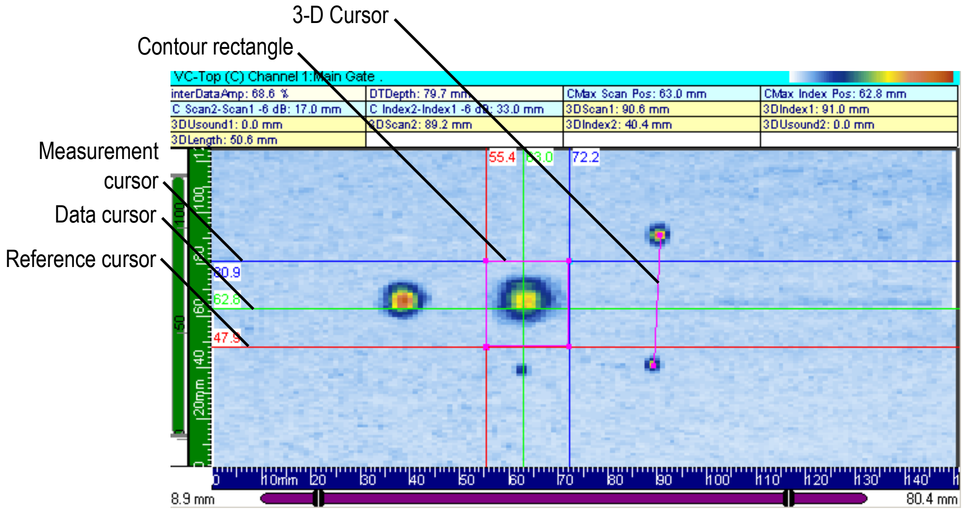


Figure 7-53 Example of cursor and zone tools

7.12.1 Measurement Cursors and Measures

With TomoView, you can use the reference and measurement cursors on a view to measure the distance along the various axes. Each cursor has a label indicating the current coordinate on the view. You can display cursor parameters values at the top of the view (see section 7.12.1.6 on page 393).

7.12.1.1 Relationship Between Cursors

Two types of relationship can exist between cursors: a fixed gap or a link.

A *fixed gap* is a relationship created between the reference and the measurement cursors for a given view. You can use it to simultaneously move both cursors with a fixed gap between them.

A *link* is a relationship created between cursors of same type in different views. You can use it to simultaneously move either the reference or the measurement cursors to the same coordinate in two different views.

7.12.1.2 Moving a Cursor with the Mouse

To move a cursor with the mouse

1. Place the mouse pointer over the reference or measurement cursor you want to move.

   The pointer shape will change to or .
2. Click, drag, and release the cursor onto the new position.

OR

◆ Double-click using the left or right mouse button on the desired position to move the reference or measurement cursor directly to the new position.

7.12.1.3 Moving a Cursor with the Keyboard

To move a cursor with the keyboard

Select the view on which you want to move the reference or measurement cursors.

◆ Use the arrow keys to move the reference cursor.
  OR
  Use the arrow keys while holding down the SHIFT key to move the measurement cursor.

The cursors only move on the active view, unless they are linked to other views.

7.12.1.4 Moving Cursors with a Fixed Gap

To move the reference and measurement cursors with a fixed gap

1. Place the reference cursor on a given position.
2. Place the measurement cursor at a distance from the reference cursor corresponding to the gap you want to create.
3. Place the mouse pointer over one of the cursors you want to move with a fixed gap (the pointer shape will change to or).
4. Press the mouse button while holding down the CTRL key and drag the cursors to the new position.

Both reference and measurement cursors will follow each other with the specified gap. The cursors will only move on the active view, unless they are linked to other views.

7.12.1.5 Creating a Link Between Cursors

To create a link between cursors

1. Click on the view where you want to create a cursor link.
2. Open the **View Properties > View Linking** dialog bar.
3. In the **Linked items group box,** select the **Scan/Index cursors** check box.

The scan and index cursors will automatically move to the same position as will do any linked cursor in another view.

7.12.1.6 Displaying Cursor Parameters

As the reference and measurement cursors are often used to take measurements inside the different views, it is very convenient to display the various readings related to these cursors.

To display the cursor parameters

1. Click on the view where you want to display the cursor parameters.
2. On the **View** toolbar, click .
3. In the **Information Groups** dialog box:
   a) Select the **Group n > Favorite Readings** category.
   b) Select the **Reference Cursor (r), Measurement Cursor (m),** and **Cursors** readings that you want to display.
   c) Click **Ok**.

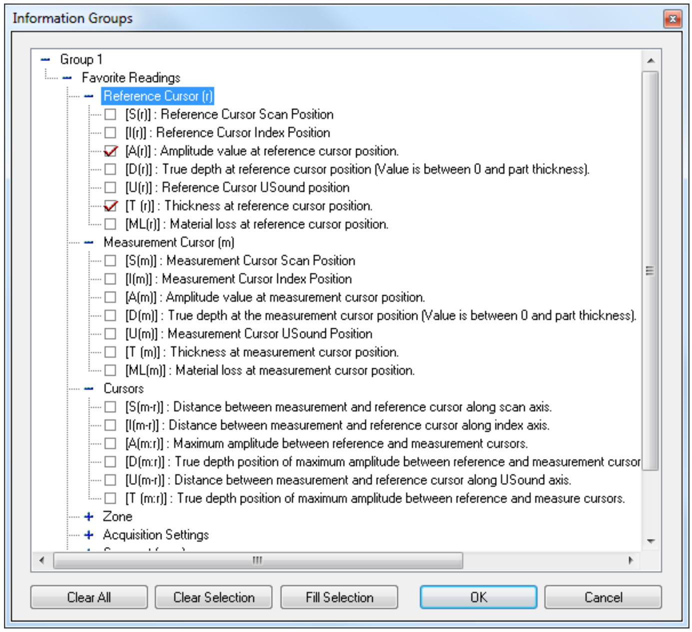
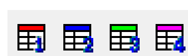


Figure 7-54 The cursor parameters

4. In the View toolbar, click  to activate the readings that you have defined for the current view.

7.12.2 Zone Tool

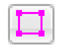
You can use the **Zone tool** to select a part of an image. A zone is a rectangular shape that can be used on volume-corrected volumetric views (only one zone can be active at a time).

NOTE

Because contours are 3-D measurement tools, they are always linked from one volumetric view to another.

7.12.2.1 Creating a Zone

To create a zone

1. Click on the view where you want to create a zone.
2. On the **View** toolbar, click .
3. Drag and release the mouse button on a view to create the zone, which appears as a magenta-colored rectangle.

7.12.2.2 Resizing a Zone

To resize a zone

1. Click on the view where you want to resize a zone.
2. Place the mouse pointer over a zone corner to be resized until the pointer changes to an oblique arrow ( or ).
3. Drag and release the zone corner using the mouse button until it has reached its arrival point.
4. If necessary, move another zone corner by repeating these steps.

7.12.2.3 Moving a Zone

To move a complete zone

1. Click on the view where you want to move the zone.
2. Place the mouse pointer over one of the zone lines until the pointer changes to a cross ().
3. Drag and release the mouse button to move the zone to its new position.

7.12.2.4 Hiding a Zone

To hide a zone

◆ In the menu, select **Tools > Hide Zone**.

7.12.2.5 Displaying Zone Parameters

No information group displays the coordinates related to the zone. To display zone parameters, you must first edit an information group.

To display the zone parameters

1. Click on the view where you want to display the zone parameters.
2. On the **View** toolbar, click .
3. In the **Information Groups** dialog box:
   a) Select the **Group *n* > Favorite Readings** category.
   b) Select the **Zone** readings that you want to display (see Figure 7-55 on page 396).
   c) Click **Ok**.

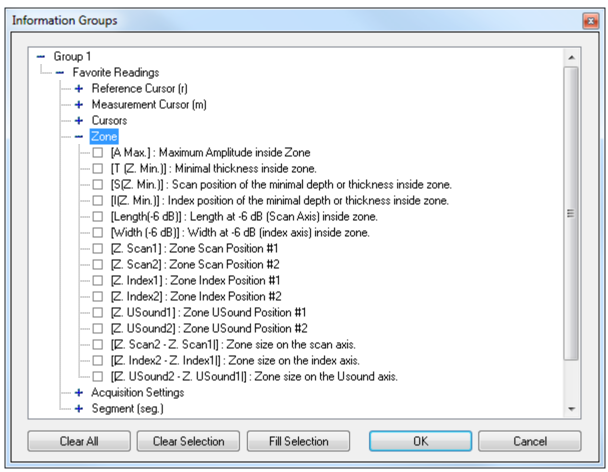


Figure 7-55 The Statistics zone parameters

4. On the **View** toolbar, click  to activate the readings that you have defined in the current view.

7.12.3 3-D Cursor

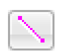
You can use the 3-D Cursor tool to select a 3-D segment and make measurements in the inspected part. A 3-D Cursor is a line-shaped tool that can be used on volumetric views (only one 3-D cursor can be active at a time).

NOTE

Because 3-D Cursors are 3-D measurement tools, they are always linked from one volumetric view to another.

7.12.3.1 Creating a 3-D Cursor

To create a 3-D Cursor

1. Click on the view where you want to create the 3-D Cursor.
2. On the **View** toolbar, click .
3. Drag and release the mouse button on a view to create the 3_D cursor, which will appear as a magenta-colored line.

7.12.3.2 Moving a 3-D Cursor

To move a 3-D Cursor

1. Click on the view where you want to move the 3-D Cursor.
2. Place the mouse pointer over a control point of the cursor to be moved until the pointer changes to a reticle shape ().
3. Drag and release the 3-D Cursor point using the mouse button until it has reached its arrival point.
4. If necessary, move the other cursor control point by repeating these steps.

7.12.3.3 Hiding a 3-D Cursor

To hide a 3-D Cursor

- In the menu, select **Tools > Hide 3-D Cursor**.

7.12.3.4 Displaying 3-D Cursor Parameters

To display 3-D Cursor parameters

1. Click on the view where you want to display the zone parameters.
2. On the **View** toolbar, click .
3. In the **Information Groups** dialog box:
   a) Select the **Group n > Favorite Readings** category.
   b) Select the **Segment (seg.)** readings that you want to display (see Figure 7-56 on page 398).
   c) Click **Ok**.

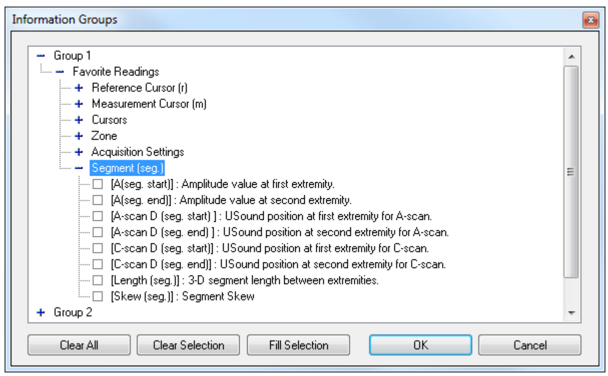


Figure 7-56 The 3-D Cursor (segment) parameters

7.13 Working with the A-scan Resynchronization Process

When performing acquisitions using two-dimensional encoded data, the data can be easily offset between the different scan lines for various reasons. The **A-scan Resynchronization** helps to correct this error, making the data more coherent for further analysis.

To resynchronize the A-scan

1. On the A-scan view, position the reference and measurement cursors around the interface entry echo.

2. On the menu, select **Processing > Set A-scan Resynchronization gate** to define a new purple gate between the reference and measurement cursors which will be used as the reference for resynchronization (see Figure 7-57 on page 399).

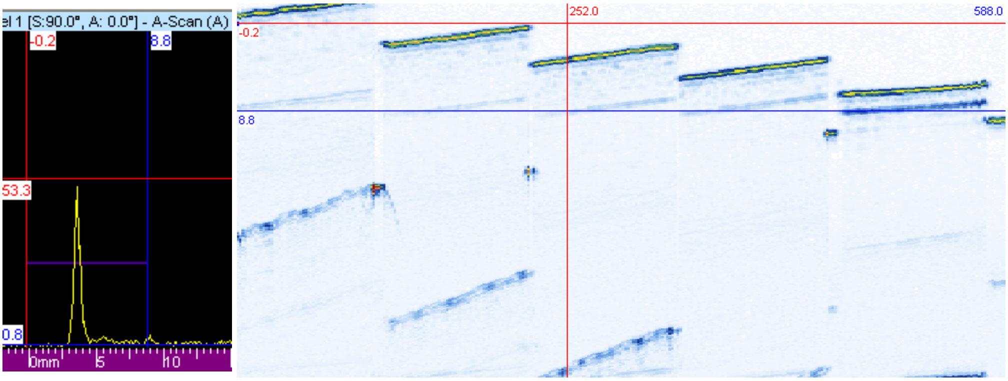


Figure 7-57 Example before resynchronization

3. On the menu, select **Processing > Resynchronize A-scans** to resynchronize the A-scans.
   Figure 7-58 on page 399 shows the resulting A-scan and C-scan.

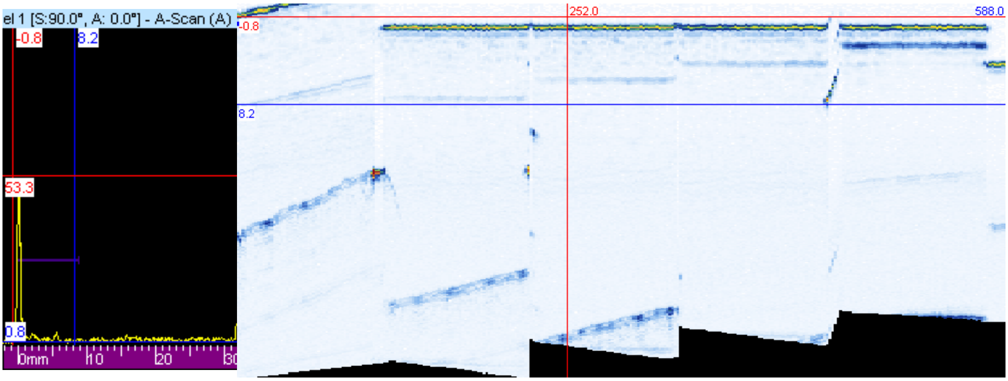


Figure 7-58 Example after resynchronization

7.14 Working with the Matrix Filters Process

The **Matrix Filters** can be used to reduce the noise on C-Scan data by processing every data point using the surrounding points. The **Matrix Filters** are particularly useful when analyzing noisy composite parts, therefore allowing clearer data representation for analysis of data files with high noise levels.

To filter a C-Scan using the Matrix Filters Component

1. In the menu, select **Processing > Matrix Filters Parameters.**
   a) Set **Algorithm** to **Average**, **Minimum**, or **Maximum**. See Figure 7-59 on page 400.
   b) Define the patch size which will be used as the reference pixel surrounding.
   c) Click **Create.**

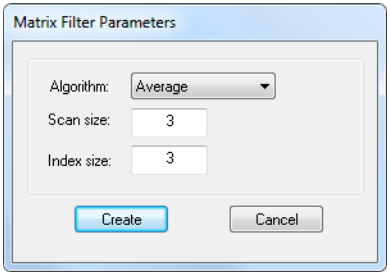


Figure 7-59 The Matrix Filters Parameters dialog box

2. In the menu, select **View > Contents** and add the matrix filtered data to the display. See Figure 7-60 on page 401.

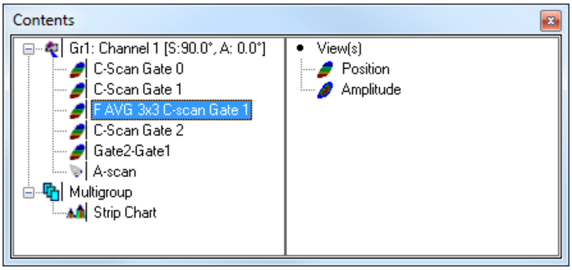


Figure 7-60 The Contents dialog box

7.15 Working with the Cylindrical Correction

This section describes the cylindrical correction tool provided in TomoView. In this section, we refer often to *Volume Corrected* views (called the **VC-X View**). To set a volumetric view as a Volume Corrected view, right click in the **View** bar, and select **Set Volume Corrected Display Mode** (see Figure 7-61 on page 401).

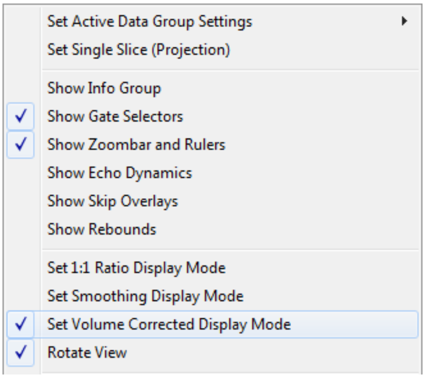


Figure 7-61 The drop down menu for Set Volume Corrected Display Mode

7.15.1 Description of the Cylindrical Correction Tool

TomoView software can display various types of information concerning the active data file and the data views. View information consists of the various view-related parameters, which can be displayed in the upper part of the pane. The information to be displayed is defined in the **Information Groups** dialog box. (For a complete description of the various information groups, consult section 7.3 on page 342.) In TomoView, all information concerning the position of indications in a user-defined cylindrical geometry has been organized in the **Cylindrical correction** group (see Figure 7-62 on page 402).

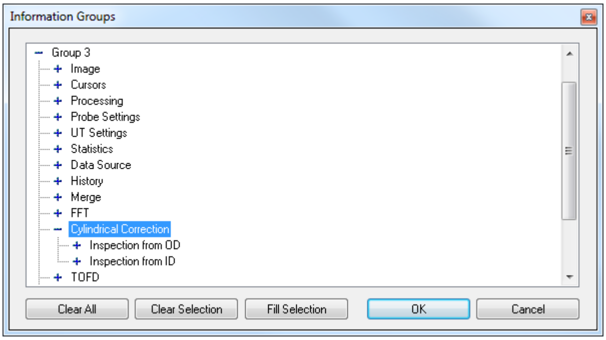


Figure 7-62 The Cylindrical correction information group

This information group contains the parameters shown in Figure 7-63 on page 403. These parameters provide the position, in the user-defined cylindrical geometry, of a point indicated by the reference cursor and/or the measurement cursor, or by the zone tool.

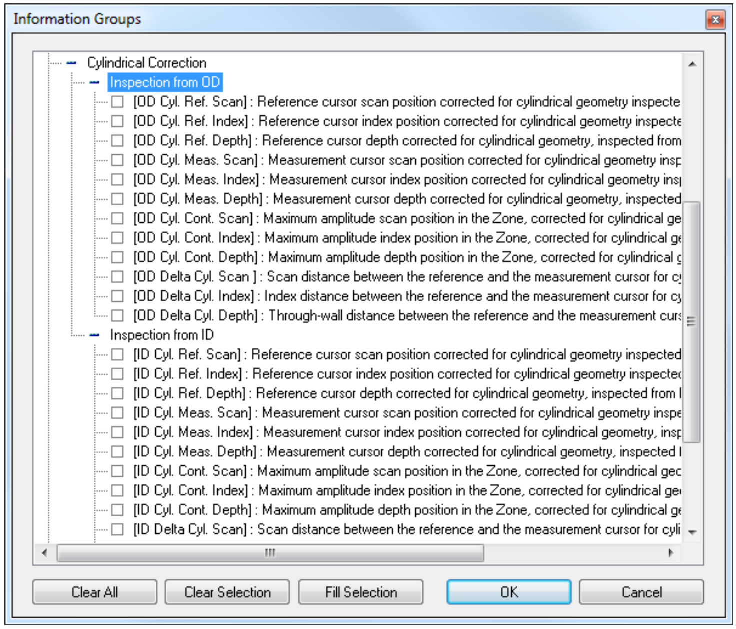


Figure 7-63 The Cylindrical correction parameters

Cylindrical correction parameters are available for an inspection conducted from the outside diameter (OD) or from the inside diameter (ID). The feature is applicable for data files acquired with scan or index axes that are calibrated in distance units (millimeters or inches) or rotation units (degrees). In addition, you can convert the units in analysis mode.

In addition to the position of the points determined by the two cursors, the distance between the cursors in the user-defined cylindrical geometry can be calculated along the considered axes.

The position of the maximum amplitude within a user-defined zone can be calculated in the user-defined cylindrical geometry.

IMPORTANT

The feature is not relevant for merged views resulting from various focal laws, as the data might be deduced from different refracted angles.

7.15.2 Case of a Skew Angle of 0° or 180°

The cylindrical correction for probe skew angles of 0° or 180° is applicable for all inspection configurations where both the scanning axis and the probe beam are oriented circumferentially, for instance:

- Raster scanning sequence for the inspection of a longitudinal pipe weld used to find defects parallel to the weld axis
- Raster scanning sequence for the inspection of a circumferential pipe weld used to find defects perpendicular to the weld axis

To obtain the cylindrical correction for the points determined by the reference and measurement cursors

1. Make sure that at least the reference (red) cursor is present in the Volume Corrected Side (B) view before activating the cylindrical correction.
2. In the VC-Side (B) view, select the applicable information fields (either OD- or ID-related) as follows:
   a) Click to activate the VC-Side (B) view.
   b) On the **View** toolbar, click  to display information group 3 in the upper part of the pane.
   c) On the **View** toolbar, click  to open the **Information Groups** dialog box.
   d) Double-click **Group 3,** and then **Cylindrical correction**.
   e) From the **Cylindrical correction** group, double-click to select the applicable information check boxes (see example in Figure 7-64 on page 405 showing a selection of OD-related parameters).
   f) Click **OK**.

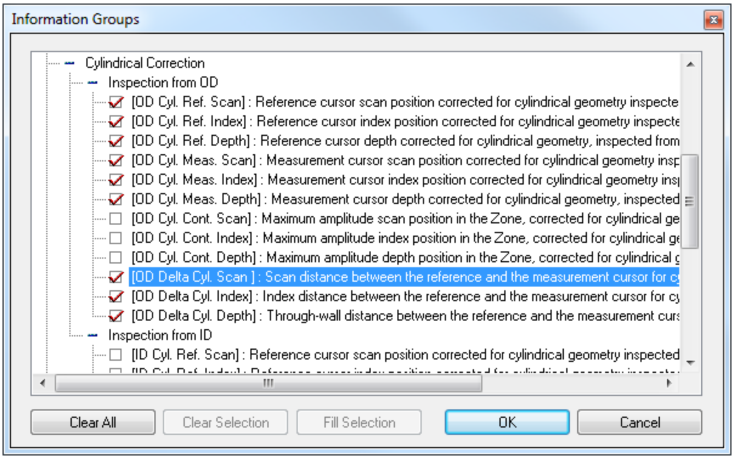


Figure 7-64 Selection of OD-related parameters

3. On the **Parameters** tab of the **Pane Properties** dialog box, click **Specimen Definition**.
4. In the **Part Definition** dialog box that appears (see Figure 7-65 on page 406), define the specimen as follows:
   a) Click **Cylindrical**.
   b) In the **Thickness** box, type the thickness of the considered cylindrical geometry.
   c) In the **Outside diameter** box, type the appropriate value for the outside diameter.
   d) Under **Cylindrical geometry**, click the **OD** or **ID** option button to specify whether the inspection is performed from the outside diameter or from the inside diameter.
   e) Click **OK**.

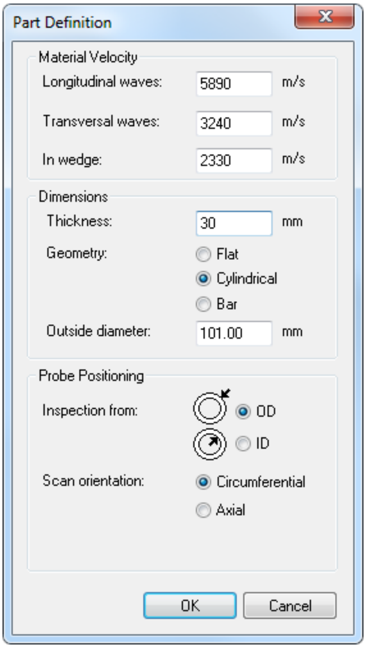


Figure 7-65 The Part Definition dialog box

When the specimen has been defined for one channel or one focal law, it is also applied for all channels or focal laws contained in the considered data file, and it can be saved in the .A01 file upon closing the data file.

- The **OD Cyl. Ref. Scan** and **OD Cyl. Ref. Depth** information fields provide the position (scan and depth axes), in the user-defined cylindrical geometry, of the point determined by the intersection of the horizontal reference (red) cursor and the angled reference (pink) cursor, in the VC-Side (B) view, or by the horizontal and the vertical reference (red) cursors in the Side (B) view.
- The information fields **OD Cyl. Meas. Scan** and **OD Cyl. Meas. Depth** provide the position (scan and depth axes) in the user-defined cylindrical geometry, of the point determined by the intersection of the horizontal measurement (blue) cursor and the angled measurement (cyan) cursor, in the VC-Side (B) view, or by the horizontal and the vertical measurement (blue) cursors in the Side (B) view.
- The information fields **OD Delta Cyl. Scan** and **OD Delta Cyl. Depth** provide the distance (scan and depth axes), in the user-defined cylindrical geometry, between the points determined by the reference and measurement cursors.

- The provided position information in the cylindrical geometry takes into account multiple rebounds of the ultrasonic beam from both OD and ID, considering the wall thickness entered by the operator in the **Part Definition** dialog box (see Figure 7-66 on page 407).

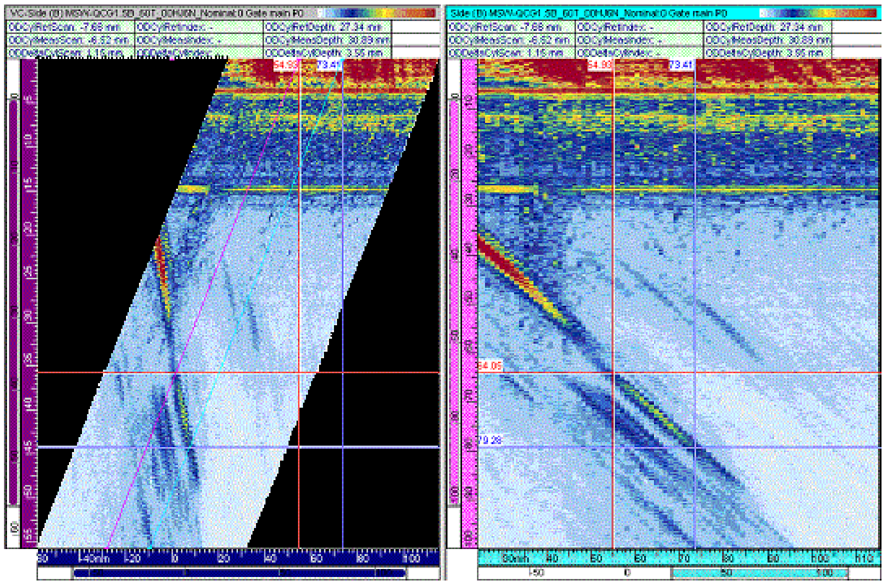


Figure 7-66 Cylindrical correction: example 1

- The provided position information in the cylindrical geometry is displayed in millimeters (or inches) if the scan axis is in distance units, and in degrees (or radians) if the scan axis is in rotation units.

To obtain the cylindrical correction for the position of the maximum amplitude within a user-defined zone

1. Make sure that at least the reference (red) cursor is present in the VC-Side (B) view before activating the cylindrical correction.
2. In the VC-Side (B) view, select the applicable information fields (either OD- or ID-related) as follows:
   a) Click in the VC-Side (B) view to make the active view.

b) On the **View** toolbar, click  to display information group 3 in the upper part of the pane.

c) On the **View** toolbar, click  to open the **Information Groups** dialog box.

d) Double-click **Group 3** and then **Cylindrical correction**. See Figure 7-67 on page 408.

e) From the **Cylindrical correction** group, double-click to select the applicable information check boxes (see example in Figure 7-64 on page 405 showing a selection of OD-related parameters).

f) Click **OK** to close the dialog box.

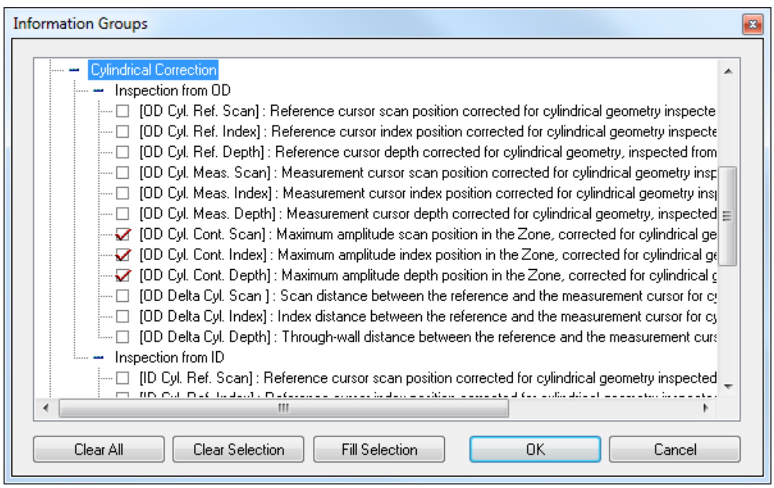


Figure 7-67 Selection of OD-related parameters

3. Position the gate selectors for the ultrasonic axis so that the flaw indication to be sized is located in the gate. (It might also be useful to display the gate cursors in the volume-corrected views.)
4. In the VC-Side (B) view, create a zone containing the considered flaw indication.
5. In the VC-Top (C) view, create a zone containing the considered flaw indication.

   You have now, in fact, created a volumetric zone defined by the zone created in the top view and the gate selectors on the ultrasonic axis (see Figure 7-68 on page 409). Additionally, the selected parameters in the information group give the position of the maximum amplitude in the zone along the scan and depth axes.

The position of the maximum along the index axis (flat geometry) can be obtained by using the standard parameter **C. Max. Index Pos.** from the **Statistics** information group.

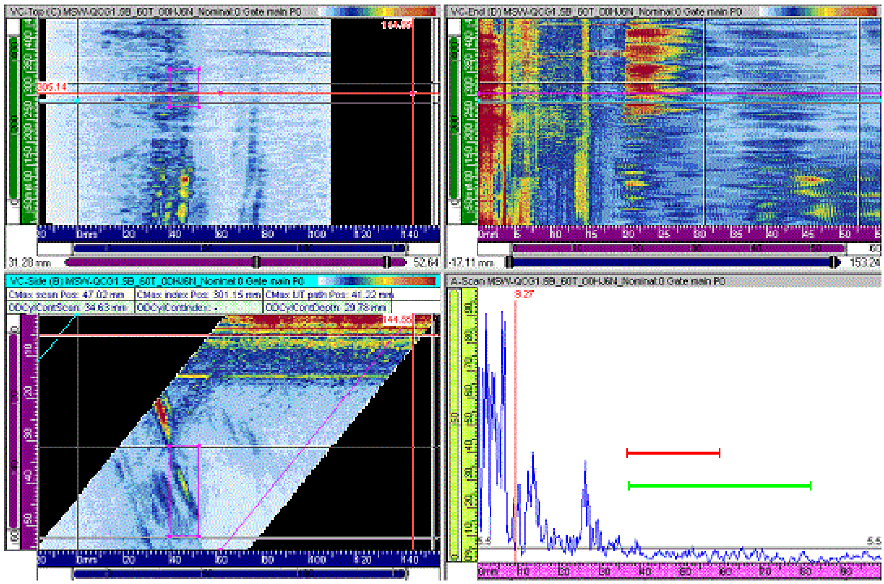


Figure 7-68 Cylindrical correction: example 2

7.15.3 Case of a Skew Angle of 90° or 270°

The cylindrical correction for probe skew angles of 90° or 270° is applicable for all inspection configurations where both the index axis and the probe beam are oriented circumferentially, for instance:

- Raster scanning sequence or line scanning sequence for the inspection of a longitudinal pipe weld used to find defects parallel to the weld axis
- Raster scanning sequence or line scanning sequence for the inspection of a circumferential pipe weld used to find defects perpendicular to the weld axis

To obtain the cylindrical correction for the points determined by the reference and measurement cursors

1. Make sure that at least the reference (red) cursor is present in the VC-End (D) view before activating the cylindrical correction.
2. In the VC-End (D) view, select the applicable information fields (either OD- or ID-related) as follows:
   a) Click in the VC-End (D) view to make it the active view.
   b) On the **View** toolbar, click  to display information group 3 in the upper part of the pane.
   c) On the **View** toolbar, click  to open the **Information Groups** dialog box.
   d) Double-click **Group 3** and then **Cylindrical correction**.
   e) From the **Cylindrical correction** group, double-click to select the applicable information check boxes (see example in Figure 7-69 on page 410 showing a selection of ID-related parameters).
   f) Click **OK**.

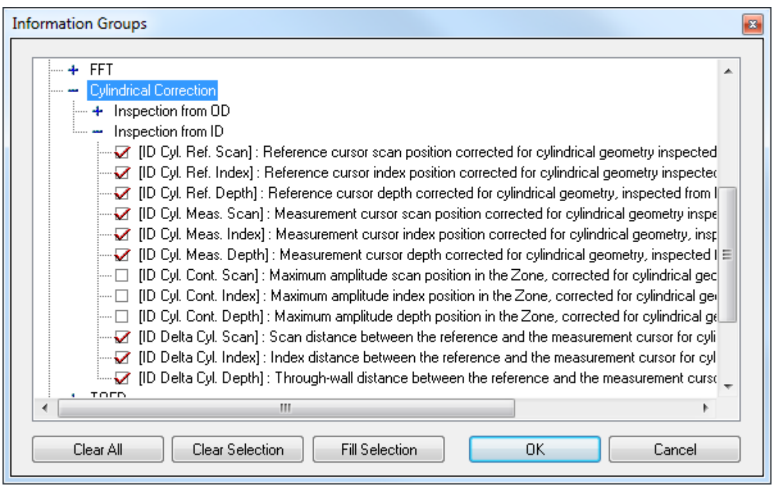


Figure 7-69 Selection of ID-related parameters

3. On the **Parameters** tab of the **Pane Properties** dialog box, click **Specimen Definition**.

4. In the **Part Definition** dialog box that appears (see Figure 7-70 on page 411), define the specimen as follows:
   a) Click **Cylindrical**.
   b) In the **Thickness** box, type the thickness of the considered cylindrical geometry.
   c) In the **Outside diameter** box, type the appropriate value for the outside diameter.
   d) Under **Cylindrical geometry**, click the **OD** or **ID** option button to specify whether the inspection is performed from the outside diameter or from the inside diameter.
   e) Click **OK**.

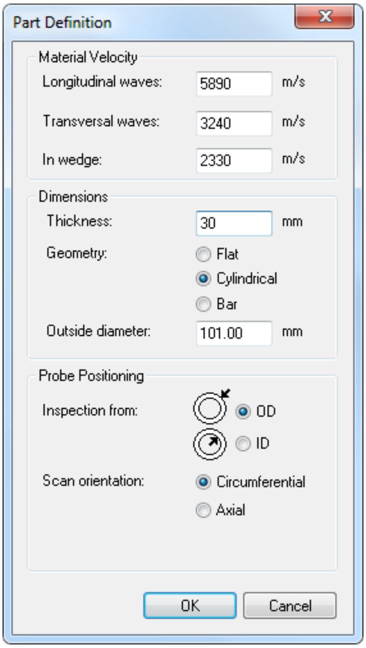


Figure 7-70 The Part Definition dialog box

When the specimen has been defined for one channel or one focal law, it is also applied for all channels or focal laws contained in the considered data file, and it can be saved in the .A01 file upon closing the data file.

- The **ID Cyl. Ref. Index** and **ID Cyl. Ref. Depth** information fields provide the position (index and depth axes), in the user-defined cylindrical geometry, of the point determined by the intersection of the horizontal reference (red) cursor and the angled reference (pink) cursor, in the VC-End (D) view, or by the horizontal and the vertical reference (red) cursors in the End (D) view.
- The information fields **ID Cyl. Meas. Index** and **ID Cyl. Meas. Depth** provide the position (index and depth axes) in the user-defined cylindrical geometry, of the point determined by the intersection of the horizontal measurement (blue) cursor and the angled measurement (cyan) cursor, in the VC-End (D) view, or by the horizontal and the vertical measurement (blue) cursors in the End (D) view.
- The information fields **OD Delta Cyl. Scan** and **OD Delta Cyl. Depth** provide the distance (index and depth axes), in the user-defined cylindrical geometry, between the points determined by the reference and measurement cursors.
- The provided position information in the cylindrical geometry takes into account multiple rebounds of the ultrasonic beam from both OD and ID, considering the wall thickness entered by the operator in the **Part Definition** dialog box (see Figure 7-71 on page 412).

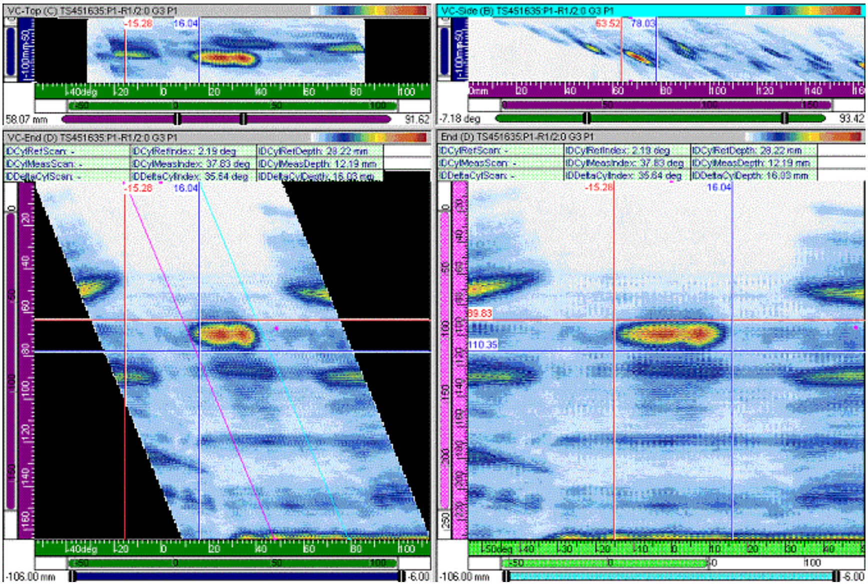


Figure 7-71 Cylindrical correction: example 3

- The provided position information in the cylindrical geometry is displayed in millimeters (or inches) if the index axis is in distance units, and in degrees (or radians) if the index axis is in rotation units.

To obtain the cylindrical correction for the position of the maximum amplitude within a user-defined zone

1. Make sure that at least the reference (red) cursor is present in the VC-End (D) view before activating the cylindrical correction.
2. In the VC-End (D) view, select the applicable information fields (either OD- or ID-related) as follows:
   a) Click to activate the VC-End (D) view.
   b) On the **View** toolbar, click  to display information group 3 in the upper part of the pane.
   c) On the **View** toolbar, click  to open the **Information Groups** dialog box.
   d) Double-click **Group 3** and then **Cylindrical correction**.
   e) From the **Cylindrical correction** group, double-click to select the applicable information check boxes (see example in Figure 7-72 on page 413 showing a selection of ID-related parameters).
   f) Click **OK**.

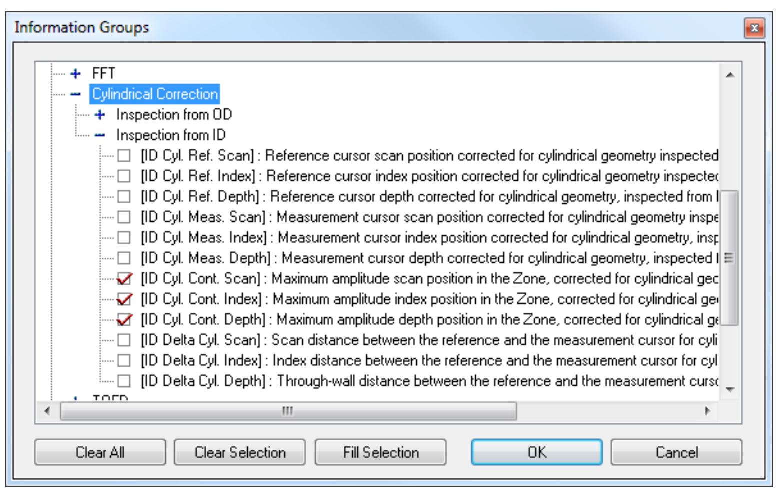


Figure 7-72 Selection of ID-related parameters

3. Position the gate selectors for the ultrasonic axis so that the flaw indication to be sized is located in the gate. (It might also be useful to display the gate cursors in the volume-corrected views.)
4. In the VC-End (D) view, create a contour containing the considered flaw indication.
5. In the VC-Top (C) view, create a contour containing the considered flaw indication.

   You have now, in fact, created a volumetric contour defined by the contour created in the top view and the gate selectors in the ultrasonic axis (see Figure 7-73 on page 414); the selected parameters in the information group give the position of the maximum amplitude in the contour along the index and depth axes. The position of the maximum along the scan axis (flat geometry) can be obtained by using the standard parameter **C. Max. Scan Pos.** from the **Statistics** information group.

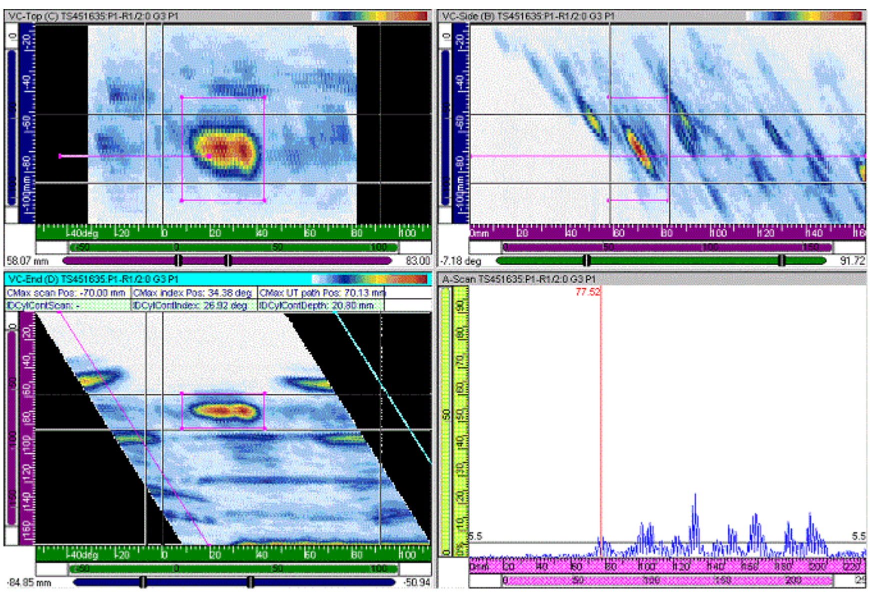


Figure 7-73 Cylindrical correction: example 4

7.15.4 Using Cylindrical Corrections

For the inspection of cylindrical parts, the Advanced Calculator takes into account the cylindrical corrections of the beams so that the readings of depth and index values are valid. The reading of depth (ODCylRefDepth) is taken perpendicularly to the tangent external surface of the part, and the reading of the index (ODCyRefIndex) is taken along the circumference of the external surface of the part (see Figure 7-74 on page 415).

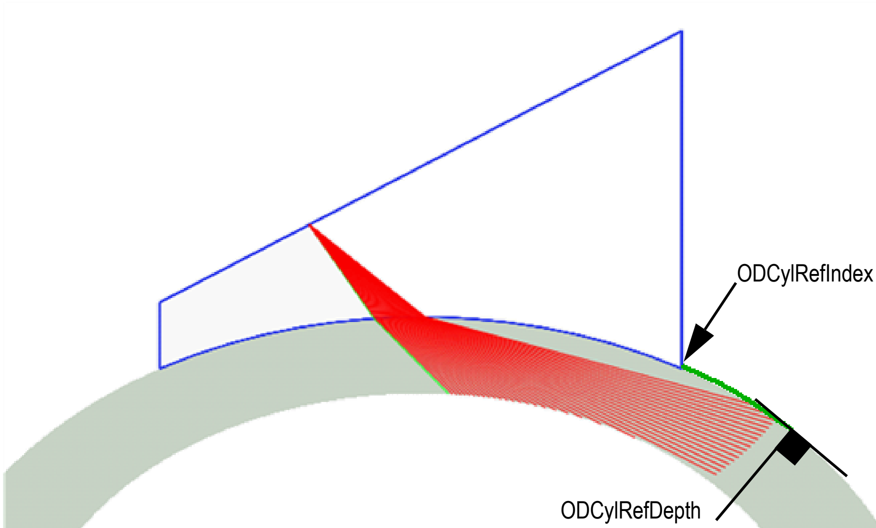


Figure 7-74 Reading of depth and index for a cylindrical part

7.15.5 Calibrating the Wedge

Before performing the inspection of a cylindrical part, you need to configure the Advanced Calculator with the appropriate part and wedge dimensions. To achieve this, you need to calibrate both the height of the first element of the probe and the wedge angle. This calibration compensates for inaccurate nominal wedge dimensions or wedge wear. Calibrate the wedge using a linear scan at 0 degree, oriented towards the center of the cylinder. The interface echo of a correctly calibrated wedge appears at 0 mm depth anywhere along the probe.

To calibrate the wedge

1. In the **1-D Linear array** tab of the Advanced Calculator (see Figure 7-75 on page 417), perform the following configuration steps:
   a) In the **Scan** group box, select **Linear** in the **Type** drop-down combo box.
   b) In the **Beam angles selection** group box, select **Refracted angle (deg)**, and set the **Start** value to 0.0.
   c) In the **Wedge** group box, set the nominal values specified for your wedge.

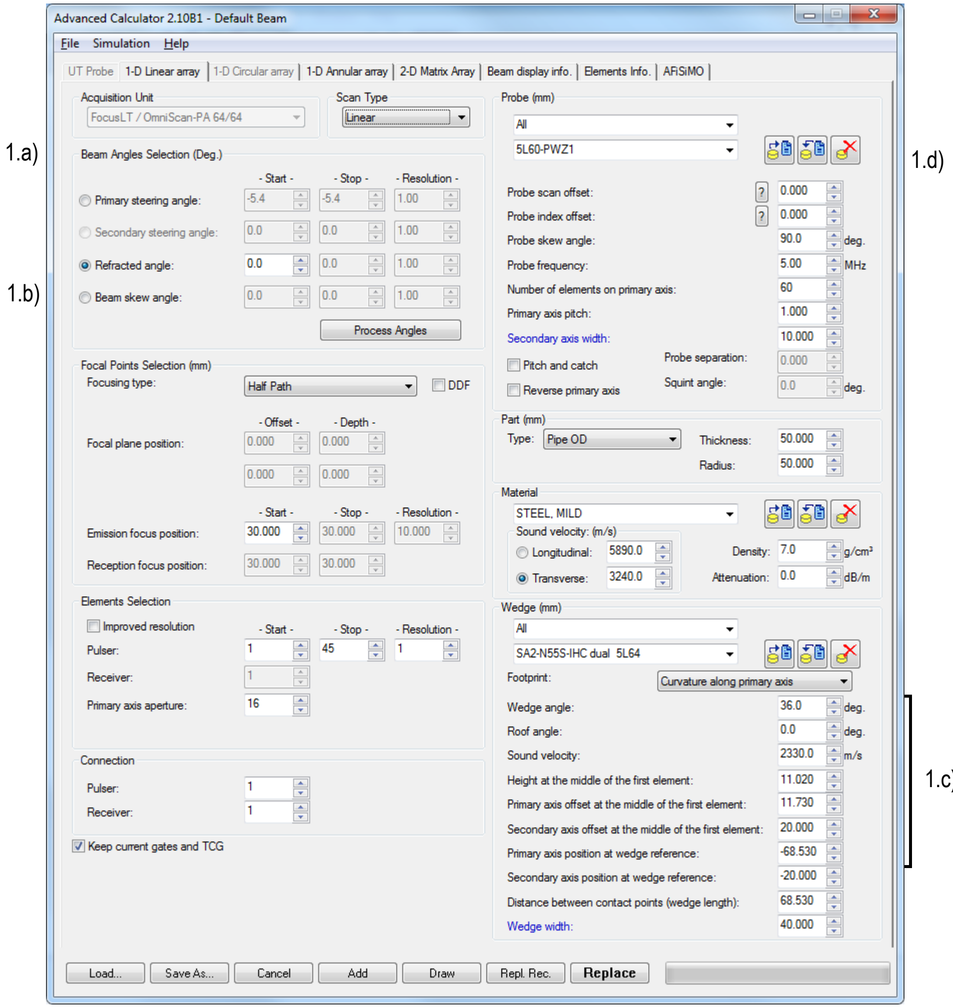


Figure 7-75 Parameters for wedge calibration

d) In the **Probe** group box, set the **Probe index offset**.

Measure the **Probe index offset** along the circumference of the outside diameter of the part, as shown in Figure 7-76 on page 418, not along a straight line. The **Probe index offset** value is negative when the **Probe skew angle** is 90°, and is positive when the **Probe skew angle** is 270°.

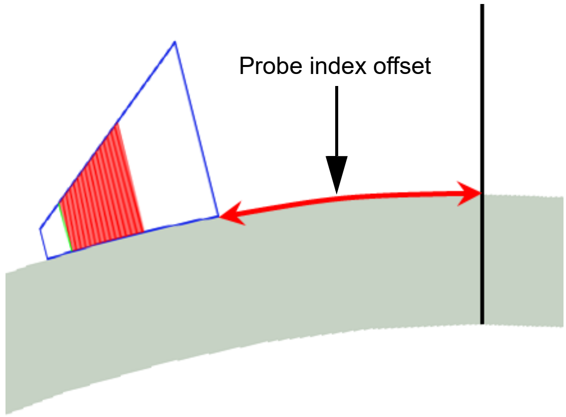


Figure 7-76 Measurement of the Probe index offset

2. Click **Save As** to save the focal laws.
3. In TomoView, load the focal law that you just saved, and examine the interface echo.

   A interface echo of correctly calibrated wedge will appear exactly at 0 mm depth anywhere along the probe, as shown in the example of Figure 7-77 on page 419. If this is the case for your application, you do not need to continue this procedure.

   If the wedge needs calibration, the interface echo will not appear exactly at 0 mm depth anywhere along the probe (as shown in the example of Figure 7-78 on page 419), and the echo line around 0 mm in the sectorial scan is not horizontal.

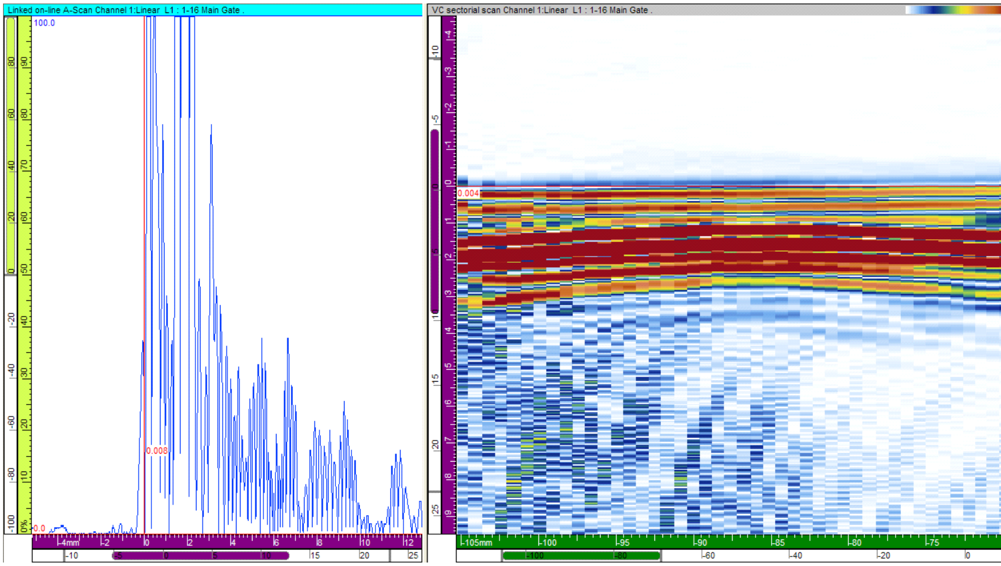


Figure 7-77 Example of the interface echo for a correctly calibrated wedge

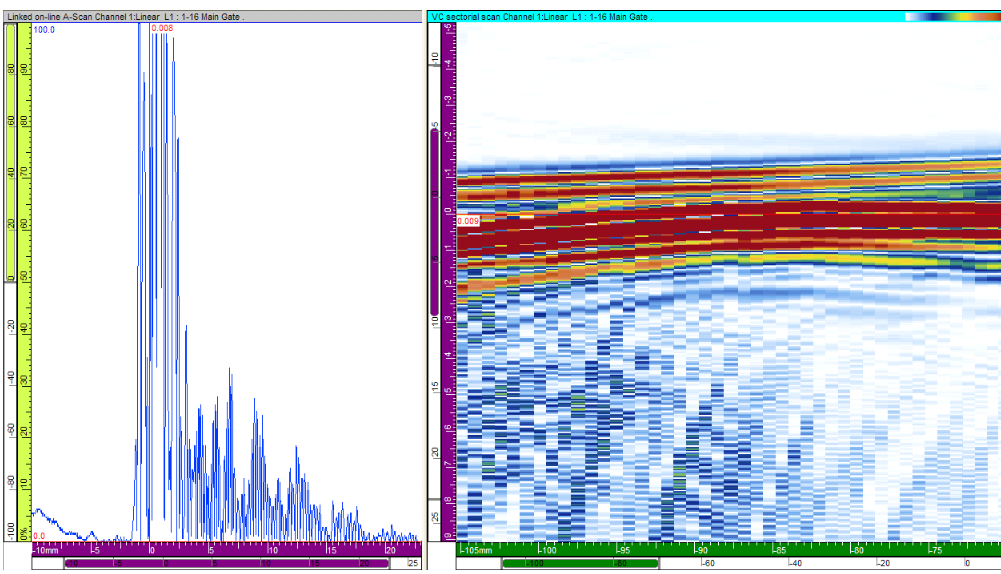


Figure 7-78 Example of the interface echo for a wedge needing calibration

4. If your wedge needs calibration, select the **1-D Linear array** tab in the Advanced Calculator.
5. In the **Wedge** group box, adjust the values for both the **Wedge angle** and the **Height at the middle of the first element** to better represent the real dimensions of the wedge.
6. Repeat steps 2 to 5 until the interface echo line is exactly at 0 mm depth anywhere along the probe as shown in the example of Figure 7-77 on page 419.

7.15.6 Analyzing Cylindrical Readings in TomoView

In TomoView, you can analyze the cylindrical readings for sectorial and linear scans.

NOTE

In TomoView, the cylindrical readings are only available in the Analysis mode, not in the Setup and Inspection modes.

To analyze cylindrical readings for sectorial scans

1. Open the data file to be analyzed.
2. Refer to Figure 7-79 on page 421 and perform the following actions:
   a) In the **VC sectorial scan** view, select the desired focal law to display and place the reference cursor.
   b) In the **Linked VC-END (D)** view, read the cylindrical correction values.

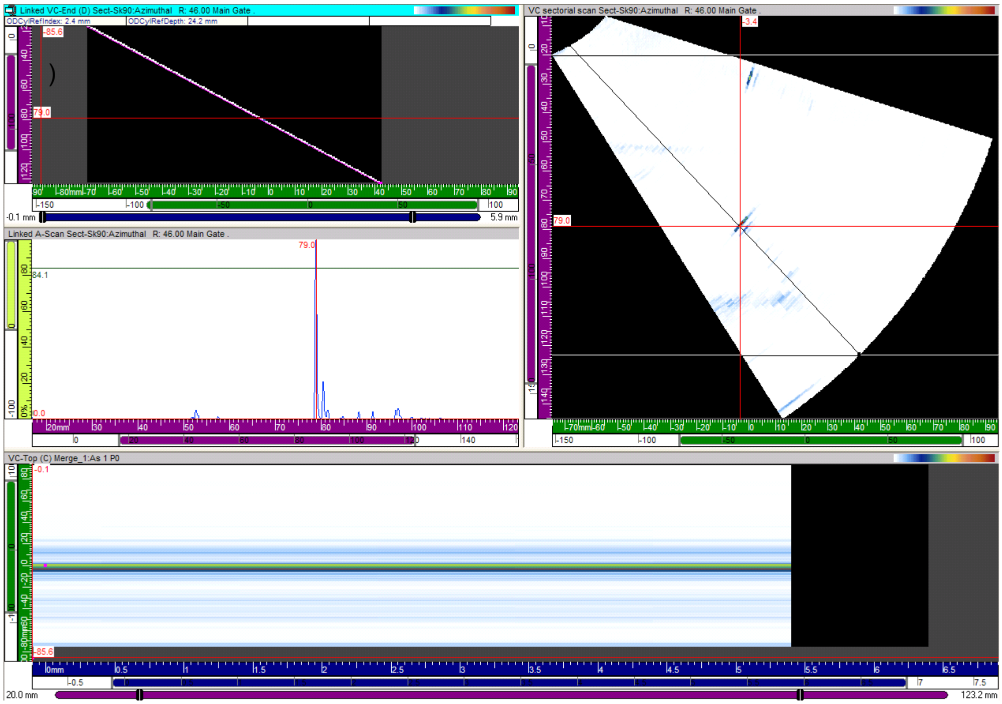


Figure 7-79 Analyzing cylindrical readings for a sectorial scan

NOTE

Do not use merged or polar views to measure cylindrical values.

To analyze cylindrical readings for linear scans

1. Open the data file to be analyzed.
2. Refer to Figure 7-80 on page 422 and perform the following actions:
   a) Set the reference cursor at the probe index position to correctly display the A-scan.
   b) In the **VC-END (D)** view, read the cylindrical correction values.

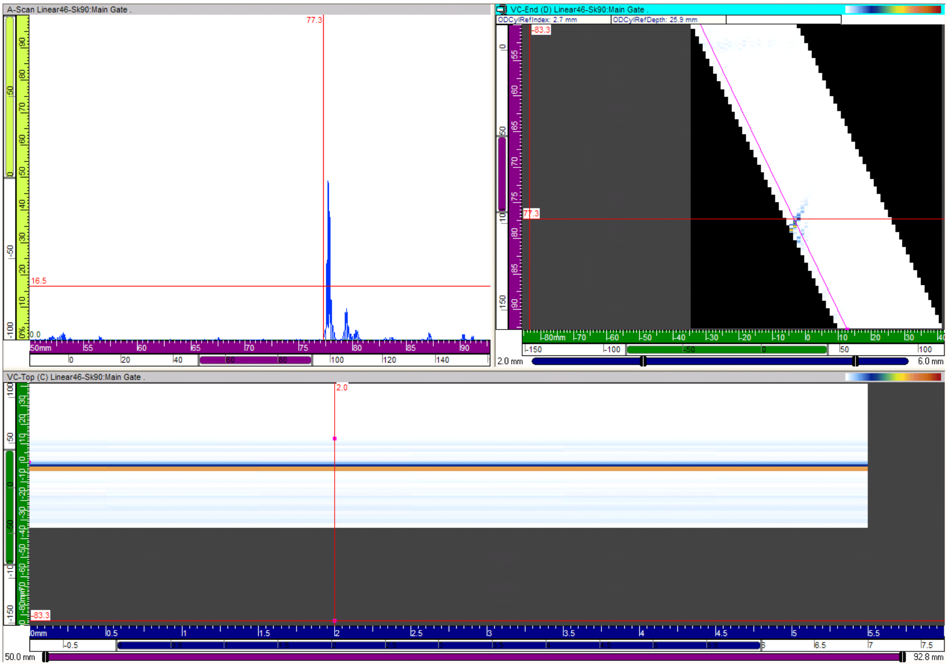


Figure 7-80 Analyzing cylindrical readings for a linear scan

NOTE

Do not use merged or polar views to measure cylindrical value.

7.16 Working with the Export Data Utility

This section describes how to use the **Export Datagroup** features to export A-scan and C-scan data to a simple text file format, which can be read with Microsoft Excel.

To export a data group

1. Open a data file.

2. Position the reference and measurement cursors on views to determine the volume of the recorded data to be exported.
3. In the menu, select **File > Export Datagroup** to file. See Figure 7-81 on page 423

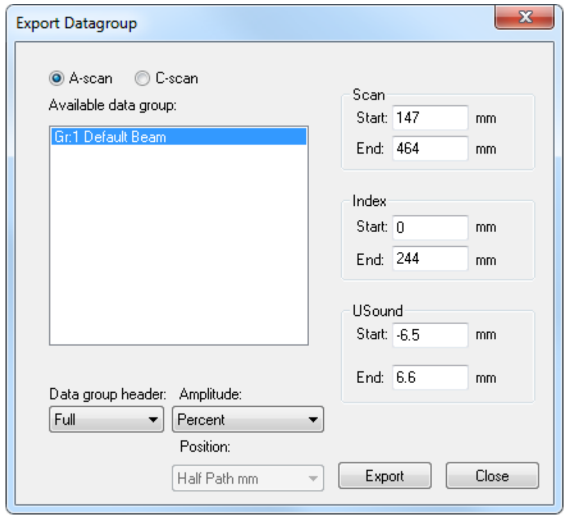


Figure 7-81 The Export Datagroup dialog box

4. In the **Export Datagroup** dialog box, select from **A-scan** or **C-Scan** data, depending on what you want to export (this will modify the **Available data group** list according to contents of the data file).
5. Select the data that you want to export in the **Available data group** list.
6. Adjust the **Start** and **End** values of the **Scan, Index,** and **USound** parameters to determine the data volume to export**.**
7. Set the **Data group header** to determine the header format that will be added to the exported data.
8. Select if you want to have the **Amplitude** data exported as **Percent** or as **Raw** data.
9. If exporting C-Scan data, select the format of the **Position** data.
10. Click **Export.**
11. Select a name and location for the exported file and click **Save**.

The data will exported in the following formats:

For A-scan

— 1st line

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

— 2nd line

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

— 3rd line

A-scan amplitude data a **Scan Start** + 2, **Index Start** position.

IndexQTY + 1st line

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

IndexQTY + 2nd line

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

For C-scan

— 1st line

C-scan data of line along the scan axis, at index position **Index Start**.

— 2nd line

C-scan data of line along the scan axis, at index position **Index Start** + 1.

— 3rd line

C-scan data of line along the scan axis, at index position **Index Start** + 2.

NOTE

The values are delimited by the TAB key (ASCII character code: 9).

7.17 Using the Hysteresis Correction Module

This section provides a detailed description on how to use the **Hysteresis Correction** module. Using this module, you can improve visualization of the previously recorded ultrasonic data from bidirectional scanning sequences by drastically reducing the effect from the mechanical backlash of the manipulator on the scan axis. The **Hysteresis Correction** module leaves the raw inspection data unaffected, whereas the processed data can be saved in separate data groups in the **Attributes** file .A01.

NOTE

The **Hysteresis Correction** module is not available in the **Lite** edition.

To perform a hysteresis correction

1. Open a data file (.rdt).
2. Display a C-scan, a Top (C), or a Volume Corrected Top (C) view.
3. Position the reference and measurement cursors on the mechanical grid to perform the wanted correction for the scan and index axis (see the example in Figure 7-82 on page 425).

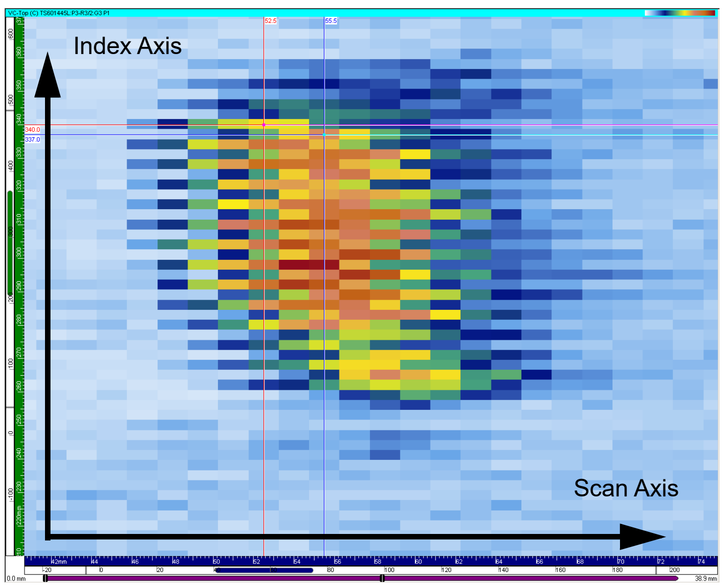


Figure 7-82 Position of the cursors

4. In the menu, select **Processing > Hysteresis correction** to display the **Hysteresis Correction** dialog box (see Figure 7-83 on page 426).

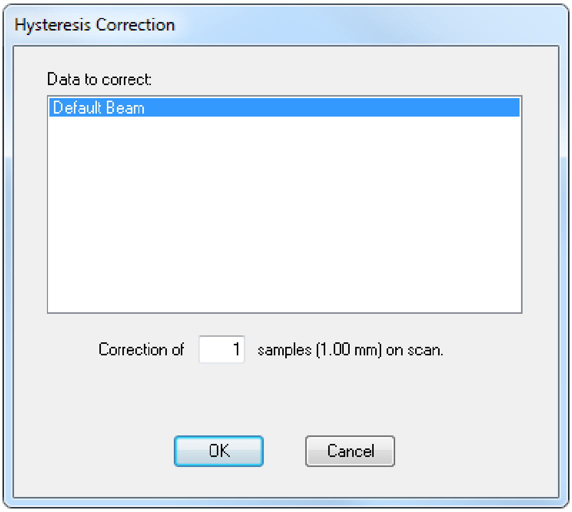


Figure 7-83 The Hysteresis Correction dialog box

5. Select the channel to be corrected.
6. In the **Correction of** box, to indicate the position shift between two successive scan lines, specify the number of samples to be used for the hysteresis correction.
7. Click **OK**.

   The correction has now been applied to the selected data group (see Figure 7-84 on page 427).

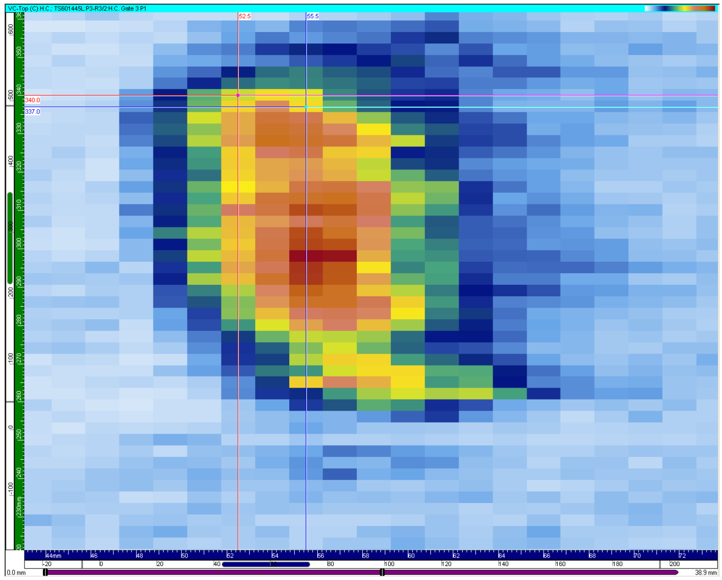


Figure 7-84 Example of a hysteresis correction

7.18 Working with the Log-to-Linear Data Conversion Process

This section describes the TomoView **Convert Log to Linear** function. In data analysis mode, you can use this function to convert recorded logarithmic data to linear data. This process creates a new data group containing the converted data.

The TomoView software saves the new linear data with the **Attributes** files (.A01). Thus, if deemed necessary, the previously created linear data is immediately available upon reopening the data file (.rdt).

To convert logarithmic data to linear data

1. Open a data file containing logarithmic data.
2. In the menu, select **Processing > Convert Log to Linear** (see Figure 7-85 on page 428).

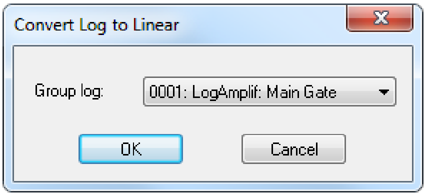


Figure 7-85 The Convert Log to Linear dialog box

3. In the **Convert Log to Linear** dialog box, select the group to be converted.
4. Click **OK**.

   The newly converted data is available in the **Contents** dialog box as a new group (see Figure 7-86 on page 428).

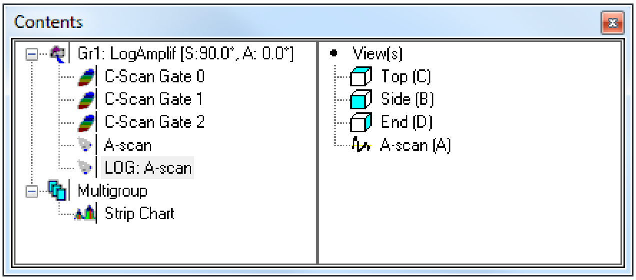


Figure 7-86 The Contents dialog box with the Log-To-Linear data group

8. Shortcut Keys

Table 17 on page 430 provides a list of the keyboard shortcuts that you can use with TomoView. You can use the shortcut keys to activate certain commands without going through all the menus or clicking a toolbar button.

Shortcut keys available for menu commands appear on the menu to the right of the command that they activate (see Figure 8-1 on page 429).

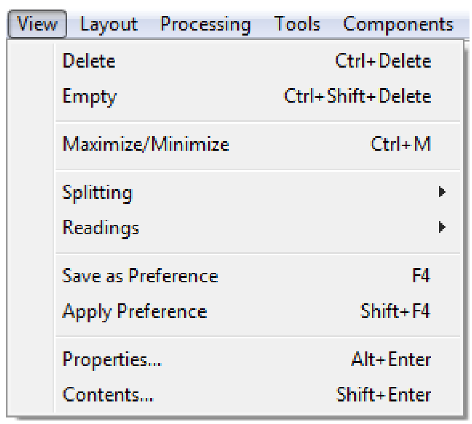


Figure 8-1 Example of shortcut keys available for menu commands

Table 17 Shortcut keys

Shortcut key	Toolbar button – Menu command Function description
CTRL+[n]	**Layout > Layout[n]** With [n] equal to 1, 2, 3,...,9, or 0, activates the n^{th} layout as when you select one of the ten layouts from the TomoView Manager under **Layout**. CTRL+0 activates the tenth layout.
CTRL+SHIFT+[n]	**Layout > Save Current Layout > Save as layout**[n] With [n] equal to 1, 2, 3,..., 9, or 0, opens the **Save Current Layout as** dialog box, allowing you to name and to save the n^{th} layout.
CTRL+A	**Processing > Set A-Scan Resynchronization Gate** In Analysis mode, after positioning the reference and measurement cursors around the interface entry echo, creates a resynchronization gate (black) between the cursors.
CTRL+SHIFT+A	**Processing > Resynchronize A-Scans** Performs the synchronization of A-scans based on the crossing in the resynchronization gate.
CTRL+C	Copies the selected text onto the Clipboard.
CTRL+E	**Tools > Clear Envelope** Resets the envelope mode.
CTRL+SHIFT+E	 – **Tools > Envelope** Toggles the envelope mode.
CTRL+F	**Tools > Freeze** Alternately freezes and resumes the display update of the active A-scan view.
CTRL+SHIFT+F	**Tools > Freeze All** Alternately freezes and resumes the display update of all views.
CTRL+H	Hides the menu and the toolbars.
CTRL+I	**File > Import OmniScan File** Opens the **Open** dialog box to select an OmniScan data file to load.
CTRL+M	 – **View > Maximize/Minimize** Alternately maximizes and restores the size of the active view.

Table 17 Shortcut keys *(continued)*

Shortcut key	Toolbar button – Menu command Function description
CTRL+N	**File > Load Default Configuration** In Inspection mode, creates a new configuration using the default values.
CTRL+O	 **– File > Open** Opens the **Open** dialog box to select a file to load.
CTRL+S	 **– File > Save Configuration As** Opens the **Save As** dialog box to select the path, name, type, and content of the file to save.
CTRL+V	Pastes the text previously copied onto the Clipboard into a text box.
CTRL+X	Cuts the selected text and copies it onto the Clipboard.
CTRL+ click and drag	On a C-scan, draws a rectangular zone, independently from the selected tool. On a cursor, moves two cursors together. For a zoom bar, changes the bar start and end values.
CTRL+SHIFT	On a cursor, moves the cursor label.
ALT+C	**Processing > Automatic Volumetric Merge by Group** Performs the automatic volumetric merge by group on the opened data file containing more than one beam per group and more than one group.
ALT+V	**Processing > Automatic Volumetric Merge** Performs the automatic volumetric merge on the opened data file containing more than one beam.
F1	**Help > Help** Opens the User's Manual
F2	 **– View > Splitting > Split Horizontally** Horizontally splits the active view into two views.
F3	 **– View > Splitting > Split Vertically** Vertically splits the active view into two views.

Table 17 Shortcut keys *(continued)*

Shortcut key	Toolbar button – Menu command Function description
F4	**View > Save as Preference** Saves the current view properties as the default property settings.
SHIFT+F4	**View > Apply Preference** Applies the default settings view properties.
ALT+F4	**File > Exit** Terminates the execution of TomoView after validating if you want to save unsaved data.
F5	**Tools > Previous Group** Selects the previous group.
F6	**Tools > Next Group** Selects the next group.
F7	**Tools > Previous Beam** In a view containing more than one beam, selects the previous beam.
F8	**Tools > Next Beam** In a view containing more than one beam, selects the next beam.
CTRL+F8	**Tools > Change Group** Opens the **Change Group** dialog box where you can select one of the groups.
F9	**– Tools > Zoom** Enables the zoom tool.
F10	**– Tools > 3-D Cursor** Enables the 3-D Cursor tool.
F11	**– Tools > Zone** Enables the zone tool (see also “CTRL+ click and drag” on page 431).
F12	**– Tools > Move** Enables the move tool.

Table 17 Shortcut keys *(continued)*

Shortcut key	Toolbar button – Menu command Function description
ALT+ENTER	 – **View > Properties** Alternately opens and closes the **Properties** dialog box.
SHIFT+ENTER	 – **View > Contents** Alternately opens and closes the **Contents** dialog box.
DELETE	**Layout > Delete Active Note** Eliminates the active note from the view.
CTRL+DELETE	 – **View > Delete** Deletes the active view.
CTRL+SHIFT+ DELETE	 – **View > Empty** Clears the content of the active view.
PAGE UP	Selects the next gate selection.
PAGE DOWN	Selects the previous gate selection.
TAB	Selects the next box or option in a dialog box.
SHIFT+TAB	Selects the previous box or option in a dialog box.
CTRL+TAB	Selects the next tab in a dialog box.
CTRL+SHIFT+ TAB	Selects the previous tab in a dialog box.
ALT+Up, Down, Left, or Right	Moves the reference (red) cursor.
SHIFT+ALT+Up, Down, Left, or Right	Moves the measurement (blue) cursor.

9. Troubleshooting

This section contains troubleshooting information that can be helpful when you encounter a problem with TomoView.

9.1 TomoView Only Starts with the File and the Help Menus

TomoView starts in the TomoViewer edition skipping the **Startup Selection** dialog box when no security hardware key is connected to the computer. In the TomoViewer edition, only the **File** and **Help** menus appear.

If you expect to start TomoView in one of the other editions, ensure that the security hardware key is connected to the computer before starting TomoView (see section 1.4 on page 28 for details).

9.2 About Firewalls and TomoView

A firewall generally blocks the communication paths between TomoView, acquisition units, and the Bootp Server program. If you need a firewall on the computer running TomoView, you need to configure the firewall to allow communication between these Evident products.

The TomoView installer deactivates the Microsoft Windows XP and Windows Vista firewalls for the network adaptor selected for the connection of the acquisition unit. This is to prevent the firewall from blocking the communication between the acquisition unit, TomoView, and the Bootp Server.

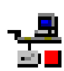
If you are using a third-party firewall, refer to the documentation of the firewall software, and then configure the firewall to allow TCP/IP packets communication from the acquisition unit IP address for the following applications:

- TomoView210.exe (default folder: [Installation Folder]\TomoView210)
- _srvbootp.exe (default folder: [Installation Folder]\Bootp).

9.3 Resolving Communication Problems Between the TomoView and the Acquisition Unit

You might encounter cases where TomoView fails to detect or looses connection with an acquisition unit. For example, when you start TomoView, the list in the **Select Device Configuration** dialog box might be empty even when your acquisition unit appears is connected and turned on.

To restore the communication between TomoView and the acquisition unit

1. Verify that you performed all steps of the TomoView installation (see section 1.5 on page 30).
2. Verify the configuration of the network adaptor used to connect the acquisition unit (see section 9.4 on page 439).
3. Close TomoView and turn off your acquisition unit.
4. On the Windows toolbar, in the Navigation area, if the Bootp Server Monitor icon includes a red indicator (), perform the following actions:
   a) Right-click the Bootp Server Monitor icon.
   b) Click **Check Status** in the contextual menu.
   c) In the **Bootp Status** dialog box (see Figure 9-1 on page 437):
   d) Read the **Problem**, **Type**, **Description**, and **Possible cause**.
   e) Follow the **Resolution** information.
   f) Click **Click here to open the Bootp Server Configuration** at the bottom of the dialog box.

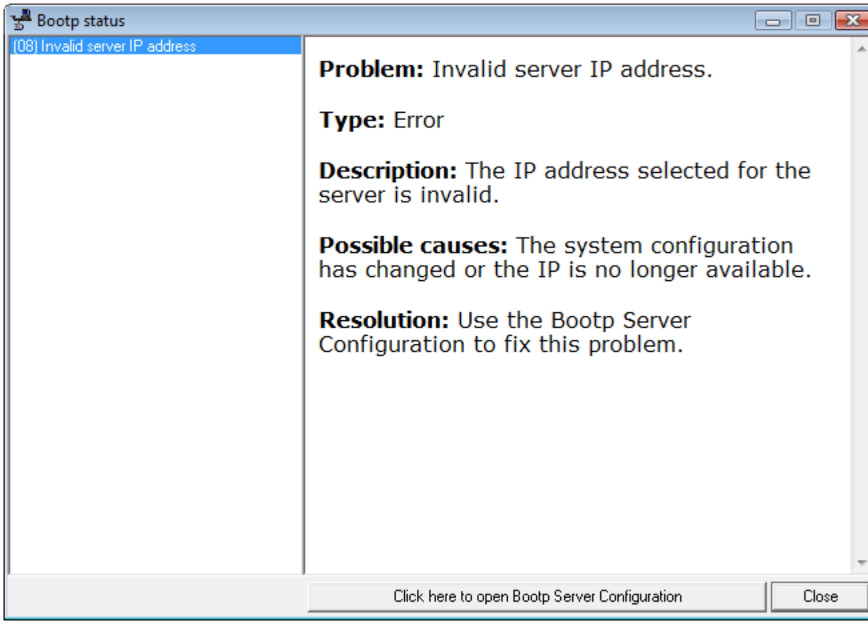


Figure 9-1 Example of the Bootp Status dialog box

g) In the **Bootp Server Configuration (Administrator)** dialog box, when the **Bootp** or the **FTP** (file transfer protocol) indicator is red, click **Restart** (see Figure 9-2 on page 438).
The Bootp Server restarts and the error condition should disappear.

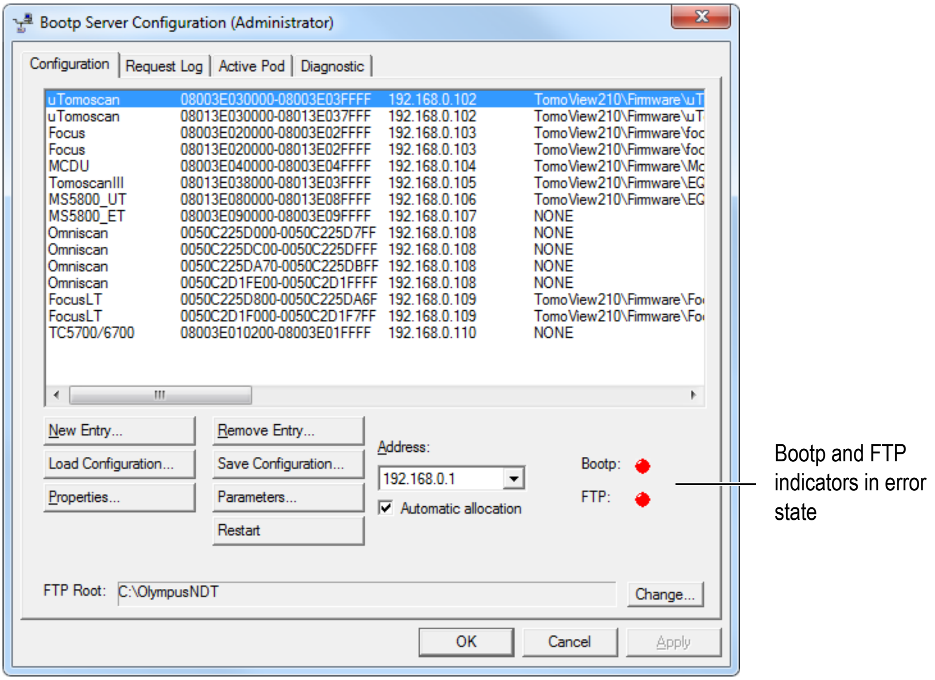


Figure 9-2 The Bootp Server Configuration dialog box in error state

h) When the **Bootp** or the **FTP** indicator remains in error condition (red) after restarting the server, restart the computer.

i) If the Bootp Server Monitor icon still includes a red indicator () after rebooting the computer, reinstall TomoView.

5. On the Windows toolbar, in the Navigation area, if the Bootp Server Monitor icon includes a no indicator (), start your acquisition unit, following the steps in the procedure found in section 1.8 on page 36 for the OmniScan and in section 1.7 on page 34 for the FOCUS LT.

9.4 Configuring the Network Adaptor

The configuration of the network adaptor, to which you connect an acquisition unit, is normally done during the TomoView installation. However, you cannot simultaneously use a network adaptor to connect to an acquisition unit and to a local area network (LAN).

If your computer has only one network adaptor, as described in the following procedure, you need to change the network adaptor configuration each time you physically change the network connection of your computer from the acquisition unit to the LAN, and vice versa.

To change the network adaptor configuration

1. On the Windows task bar, in the Navigation Area, right-click the Bootp Server Monitor icon (see Figure 9-4 on page 440).
2. In the contextual menu, click **Network Adaptor Selection**.

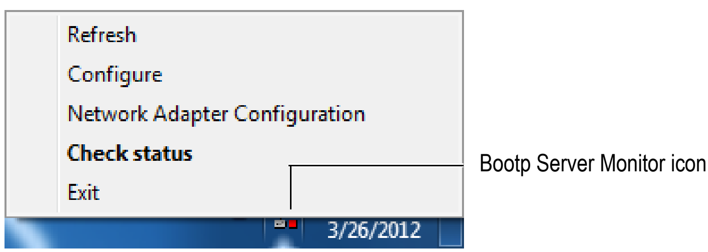


Figure 9-3 Opening the Network Adaptor Configuration dialog box

3. In the **Network Adaptor Configuration** dialog box perform the following tasks (see Figure 9-4 on page 440):
   a) Click **Configure for acquisition (Static IP)** when you want to configure your computer to connect to the acquisition unit.
   b) Click **Configure for Internet (DHCP)** when you want to configure your computer to connect to a network.
   c) Click **Finish**.

TIP

Click **Skip** to cancel the operation.

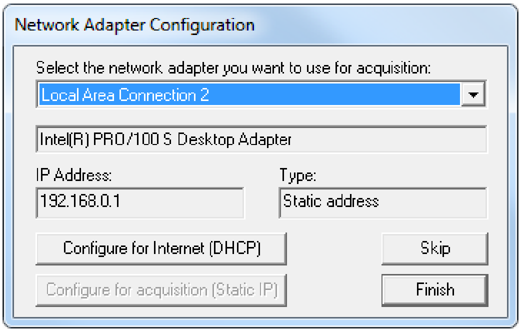


Figure 9-4 Configuring the network adaptor to connect to an acquisition unit

9.5 Handling Large Data Files with East Asian Languages

Microsoft Windows provides text service that you can use to enter text in East Asian languages using a standard keyboard (see Japanese example in Figure 9-5 on page 440). Windows loads the text service modules in the middle of the virtual address space. This fragments the available memory space and causes problems for large TomoView data files (up to 1 GB).

Figure 9-5 The Japanese language bar

To avoid this problem, you must start TomoView before loading the East Asian text service modules. The solution is to set English as the default input language in Windows (see section 9.5.1 on page 441), and then start TomoView (see section 9.5.2 on page 443).

9.5.1 Setting English as the Default Input Language

The following procedure uses Japanese screen captures to illustrate the tasks.

To set English as the default input language

1. On the Windows taskbar, click **Start**, and then click **Control Panel**.
2. In the **Control Panel**, click **Regional and Language Options** (see Figure 9-6 on page 441).

Figure 9-6 Regional and Language Options

3. In the **Regional and Language Options** dialog box (see Figure 9-7 on page 442):
   a) Click the **Language** tab.
   b) In the **Text services and input languages** group box, click **Details**.

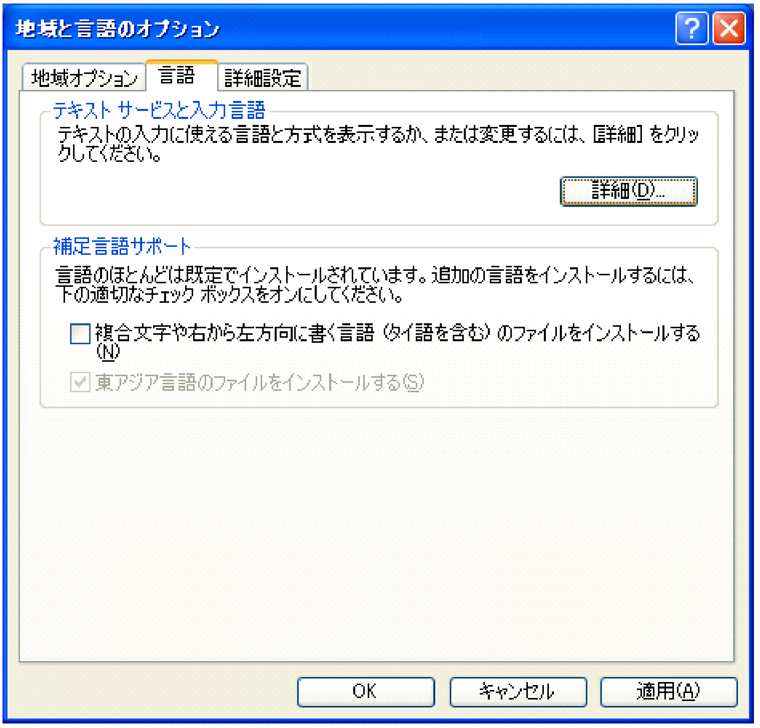


Figure 9-7 The Language tab

4. In the **Text services and input languages** dialog box, on the **Settings** tab, in the **Default input language** group box, select **English (United States - US)** [see Figure 9-8 on page 443].
5. If no English keyboard is installed, in the **Installed services** group box, click **Add** to install the **English (United States - US)** keyboard, and then execute the task in step 4.

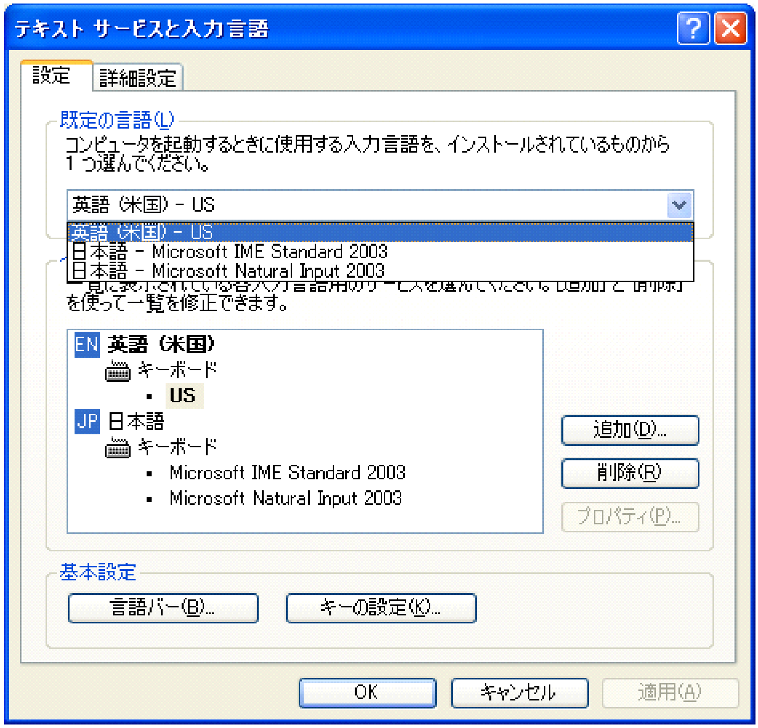


Figure 9-8 Setting English as default input

6. Click **OK**, and then close all dialog boxes.
7. Restart the computer.

9.5.2 Starting TomoView to Handle Large Data Files with East Asian Languages

The following procedure uses Japanese screen captures to illustrate the tasks.

To start TomoView to handle large data files with East Asian languages

1. Ensure that English is set as the default input language in the Windows taskbar (see Figure 9-9 on page 444).

Figure 9-9 Ensure that English is the default input language

2. If English is not the default input language, refer to section 9.5.1 on page 441 to set it.
3. Start TomoView by double-clicking its icon (see Figure 9-10 on page 444).

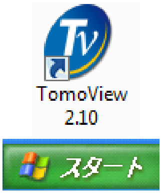


Figure 9-10 Starting TomoView

4. Once TomoView is started, change the input language to Japanese from the language bar. See Figure 9-11 on page 444.

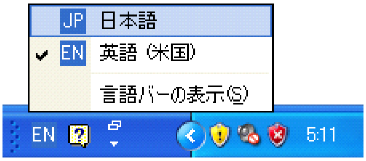


Figure 9-11 Switching the input language to Japanese from the language bar

TomoView can now handle large data files (up to 1 GB) with Japanese as the input language.

10. Additional Features

10.1 Working with the Import PASS Files Utility

This section describes how to use TomoView to visualize and analyze .mnp data files, generated by PASS software.

NOTE

The import PASS function is not available in the **Lite** edition

10.1.1 Importing and visualizing an .mnp File

To visualize an .mnp file

1. Open any data file in TomoView.
2. In the menu, select **File > Import PASS file**.
3. In the **Open** dialog box that appears (see Figure 10-1 on page 446), select the PASS file you want to open, and click **Open**.

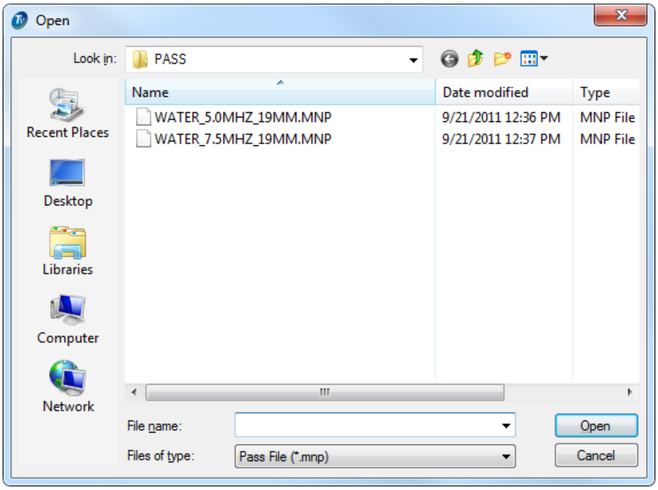


Figure 10-1 The Open dialog box for opening a PASS file

4. In the menu, select **View > Contents**.
5. In the **Contents** dialog box (see Figure 10-2 on page 446), select the desired view or views for the imported PASS file.

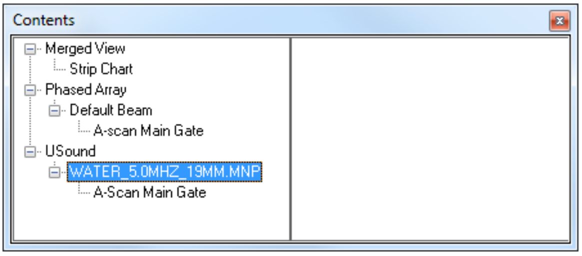


Figure 10-2 Example of the Contents dialog box with a PASS file

6. Display a Side (B) view to visualize the PASS files. (Figure 10-3 on page 447 shows an example of a possible layout, including the echo dynamics along the reference cursor.)

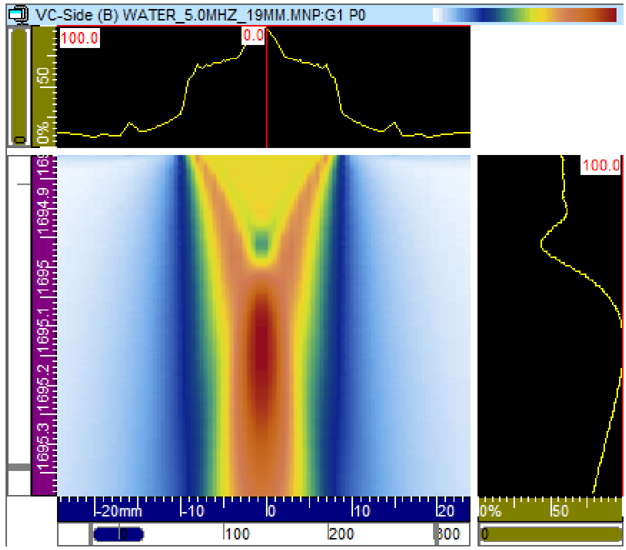


Figure 10-3 Example of layout for a PASS file

Several .mnp files can be imported into the same .rdt file.

10.1.2 Saving PASS Data in an .rdt File

To save PASS data in an .rdt file

1. Follow the steps described in the previous section for importing a PASS file into TomoView.
2. In the title bar of the document window, click  to close the data file.
3. In the message box that appears (see Figure 10-4 on page 448), click **Yes** to save the imported PASS data in the Blank.A01 file.

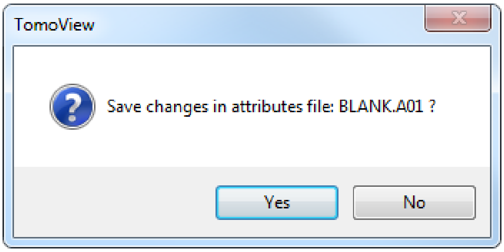


Figure 10-4 Message box for saving imported PASS data

NOTE

To save the .rdt file under a user-defined name, you have to rename the file Default.rdt in Windows Explorer either before or after importing the PASS data.

10.1.3 Determining the Characteristics of an Ultrasonic Beam

To determine the characteristics of an ultrasonic beam simulated with PASS software, you can use either TomoView graphical tools (cursors) or several of the different available readings. Make sure that the **Expert** mode is activated to have all the readings visible (for more information on the **Expert** mode, see section 3.16 on page 220).

- The **Statistics** > **[MaxPASS]: Maximum soundfield amplitude in AU (only for PASS files)** parameter provides a measurement of the sound field amplitude in the acquisition unit.
- The **Favorite Readings** > **Zone** > **[A-Zone]: Maximum Amplitude inside Zone** is the position of the maximum amplitude in the sound field inside the zone.
- Using the 3-D cursor, the **Favorite Readings** > **Segment (seg.)** > **[Length (seg.)]: 3-D segment length between extremities** reading can be used to determine the length and width of the focal spot.

10.2 Changing the Color of an Element

To change the color of a selected element

1. On the **File** menu, click **Preferences**.
2. In the **Preferences** dialog box, click the **View Colors** or **Tool Colors** tab.
3. Click the button corresponding to the element for which you want to change the color.

   The **Color** dialog box appears (see Figure 10-5 on page 449).

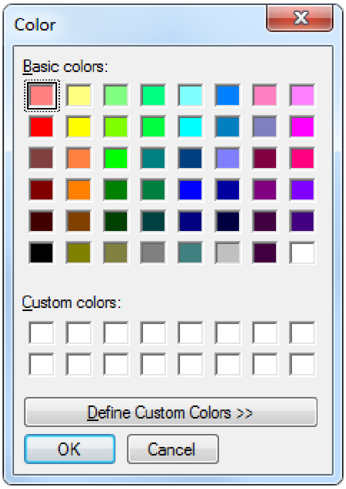


Figure 10-5 The Color dialog box

4. In the **Color** dialog box, click the color you want in the **Basic colors** palette or in the **Custom colors** palette.
5. If none of the available colors are appropriate, click **Define Custom Colors**.

   The **Color** dialog box unfolds to reveal more options (see Figure 10-6 on page 450).

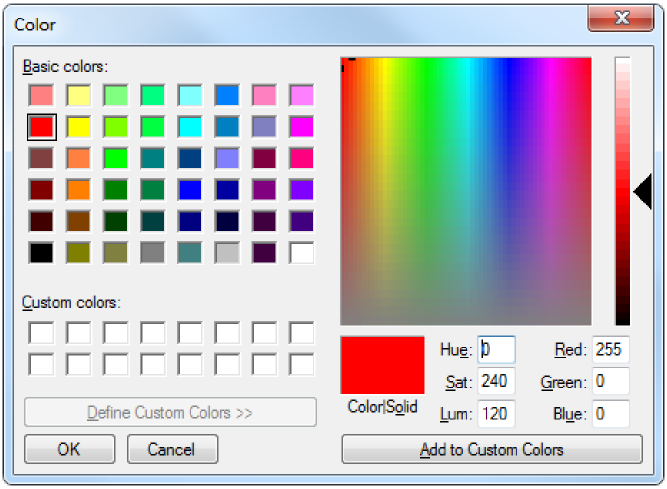


Figure 10-6 The Color dialog with full color palette

6. Click the color in the **Basic colors** palette that best corresponds to the color you want.

   The selected color appears in the **Color > Solid** zone.

7. To select a custom color:

   - Press the mouse button and, while holding it down, drag the pointer to select the color you want from the color matrix. Use the slider at the right of the dialog box to modify the brightness.

     OR

     Specify the color by setting the **Hue/Sat/Lum** or **Red/Green/Blue** parameters.

8. Click **Add to Custom Colors**.

   The new color appears in the selected color box of the **Custom colors** palette.

9. Click **OK**.

   Back in the **Preferences** dialog box, the selected color appears on the selected button.

10.3 Configuring Links Between TomoView Elements

With TomoView, you can link the behavior of elements. For example, moving a cursor in one view automatically moves the same linked cursor in another view.

You can set the general link configuration for all views in the **Link** tab of the **Preferences** dialog box (see section 3.14.2 on page 214). You can also customize the link configuration for a specific view.

To set the general link preferences

1. On the menu, select **File > Preferences**.
2. Select the **Linking** tab.
3. In the two-column list on the left, select the element for which you want to configure the linking behavior.
4. For cursor and ruler items:
   a) Under **Uncorrected Views**, select the desired linking behavior to be applied for all uncorrected views.
   b) Under **Corrected Views**, select the desired linking behavior to be applied for all corrected views.
5. For the other elements, select the desired linking behavior to be applied for all views.

To customize the link configuration for a specific view

1. Select the view for which you want to customize the link configuration.
2. On the menu, select **View > Properties**.
3. In the **Properties** dialog box (see Figure 10-7 on page 451), select the **View Linking** tab, and then select the check box for the items that you want to be linked for this view.

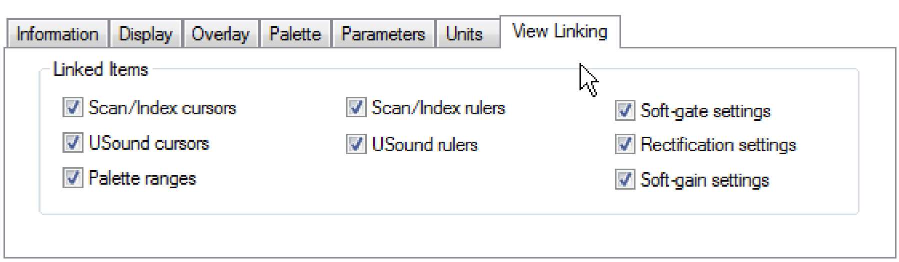


Figure 10-7 The View Linking tab of the Properties dialog box

4. On the menu, select **View > Save as Preferences** to save your view linking changes in the display setup (.rst) file.

TIP

You can press the F4 and SHIFT-F4 shortcut keys to respectively save view preferences to the display setup (.rst) file and load view preferences from the display setup file.

10.4 Working with the Excel Exchanges Process

TomoView can exchange data with a Microsoft Excel worksheet using its **Excel Exchanges** component. You can use this tool to create user-defined readings in the information groups.

The **Excel Exchanges** components dynamically exports selected TomoView reading values to a Microsoft Excel worksheet. You can configure your Excel worksheet to perform computation on the imported data. You can also configure the **Excel Exchanges** component to dynamically import the result back into TomoView, into an information group **Custom info field**.

You can use the **Excel Exchanges** component in Setup or in Analysis mode, provided that the selected readings are available. Once created, your user-defined readings are saved when you save a hardware setup (.acq files) or a display setup (.rst file). When you load these setup files, they become immediately available for a new data file.

IMPORTANT

The TomoView **Excel Exchanges** component only works when Microsoft Excel is installed on the computer.

In the defect sizing example used in the following procedure, you can use the **Excel Exchanges** to calculate in Excel the actual scan size [S*(m-r)] of a defect in a cylinder (see Equation (1) on page 452 and Figure 10-8 on page 453), using the S(m-r) and U(r) values provided by TomoView.

$$S^*(m\text{-}r) = \frac{(r_\varphi - U^*(r))S(m\text{-}r)}{r_\varphi} \quad (1)$$

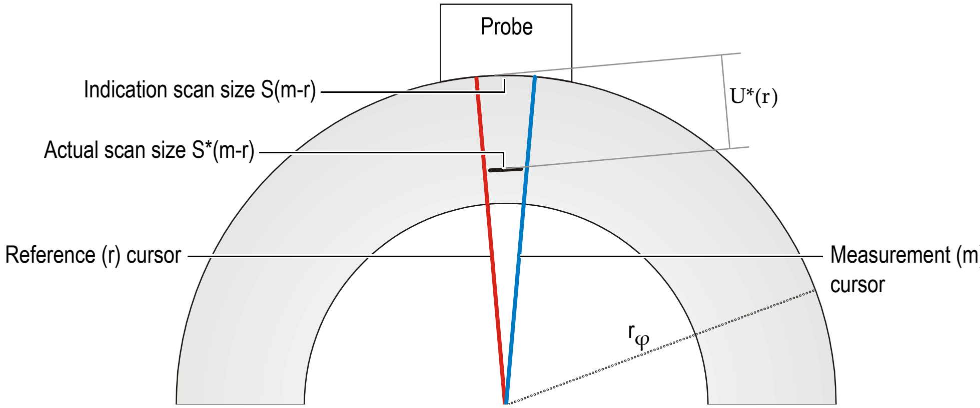


Figure 10-8 The actual scan size of a defect

To exchange data between TomoView and Excel

1. In Microsoft Excel:
   a) Create a file containing information similar to what is shown in Figure 10-9 on page 454.
   b) Type the value for the radius in cell C8.
   c) In cell B8, type following formula for Equation (1) on page 452:
   =(A8*(C8-D8))/C8

NOTE

The values in cell A8 and D8 will be imported from TomoView.

---

   d) Save the Excel spreadsheet with an appropriate name, in the folder of your choice.

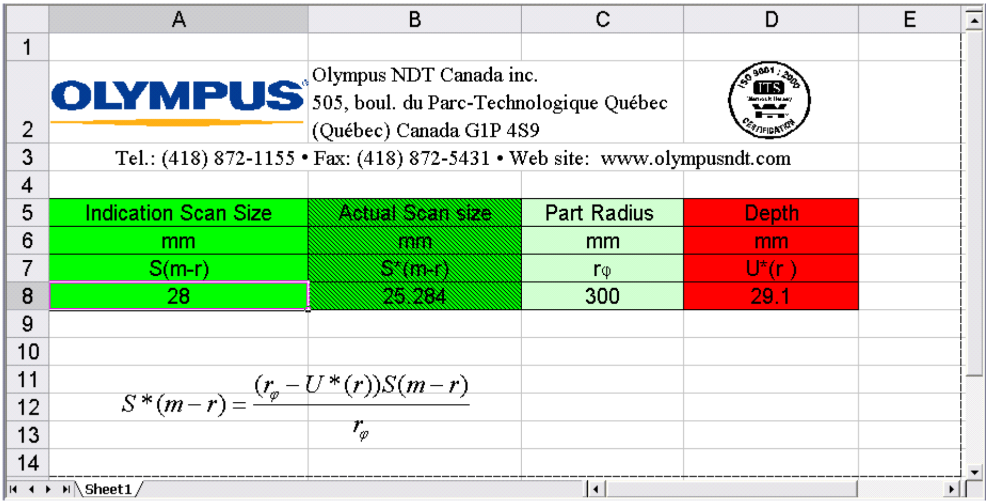


Figure 10-9 Example of an Excel worksheet content

2. In TomoView, open and setup the appropriate data file similar to the example shown in Figure 10-10 on page 455:
    a) Select a layout and a view where you clearly see the indication that you want to size.
    b) Select to show the U(r) and S(m-r) readings in the view (see section 4.10 on page 276 for details).
    c) Position the reference (red lines) and measurement (blue lines) cursors on each side of the indication.

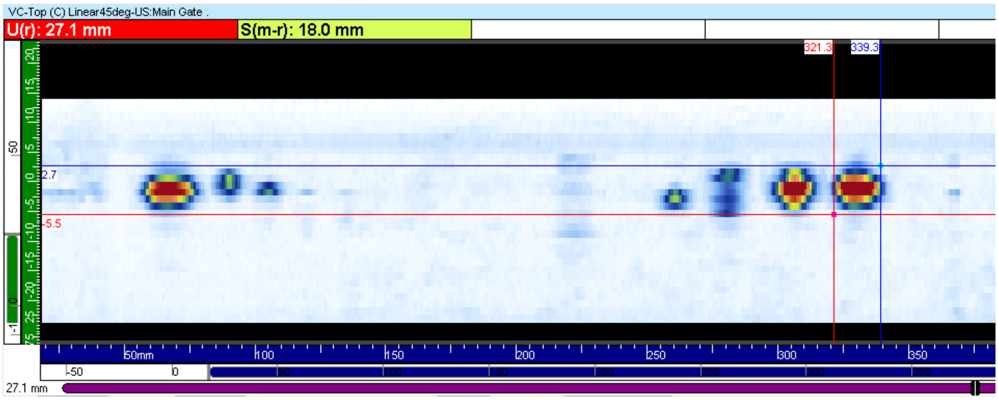


Figure 10-10 Setting up the view and the readings

3. On the menu, select **Processing > Excel Exchanges**.
4. In the **Excel Exchanges** dialog box (see Figure 10-11 on page 455):

   a) Click the browse button (...) on the right of the **Excel file** text box.

   b) In the **Open** dialog box, select the Excel file that you created in step 1.*d*, and then click **Open**.

   c) Click **Add Export**.

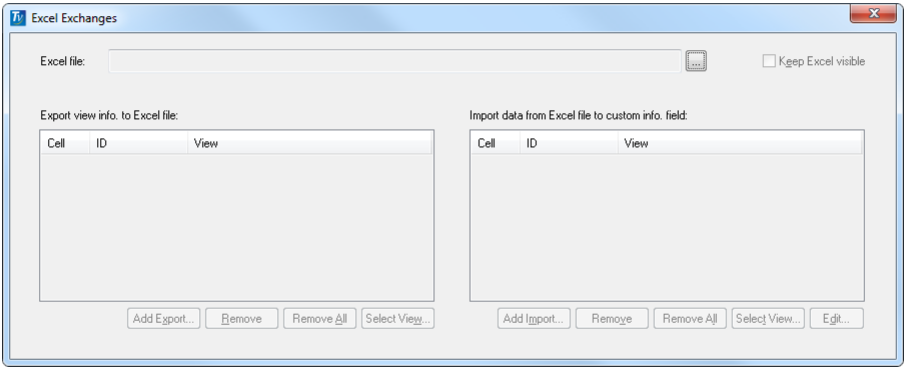


Figure 10-11 The Excel Exchanges dialog box

5. In the **Information Groups** dialog box, select the field you want to export [U(r) and S(m-r) in the case of this example], and then click **OK** (see Figure 10-12 on page 456).

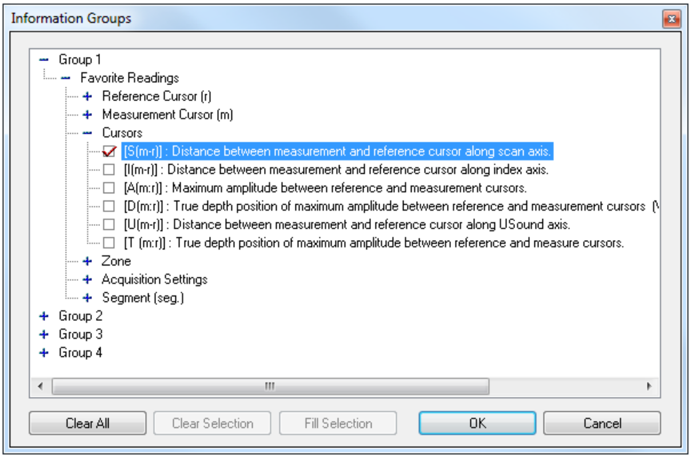


Figure 10-12 The Information Groups dialog box with available readings for the Excel Exchanges

6. Back in the **Excel Exchanges** dialog box, in the **Export view info. to Excel file** list:
   a) Double-click the **Cell** column on the line of the added export field, and then enter the corresponding Excel worksheet cell [D8 for U(r) in this example].
   b) Select the cell to export, and then click **Select View**.
7. In the **View Selector** dialog box (see Figure 10-13 on page 457), highlight the view in which you want the field to appear, and then click **Select**.

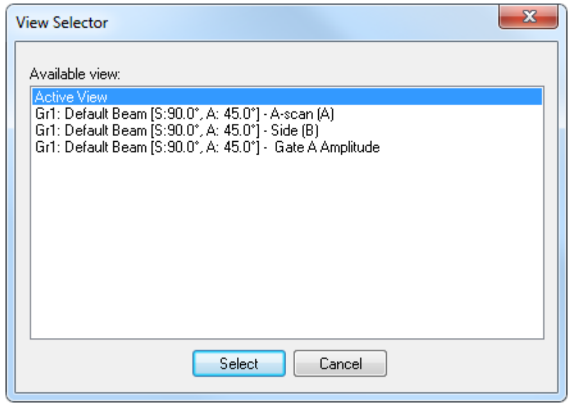


Figure 10-13 The View Selector dialog box

8. Repeat steps 5 to 7 for each exported information field.
9. In the **Excel Exchanges** dialog box, click **Add Import**.
10. In the **New View Info** dialog box, type the appropriate information for the reading you want to import, and then click **OK** (see the example shown in Figure 10-14 on page 457).

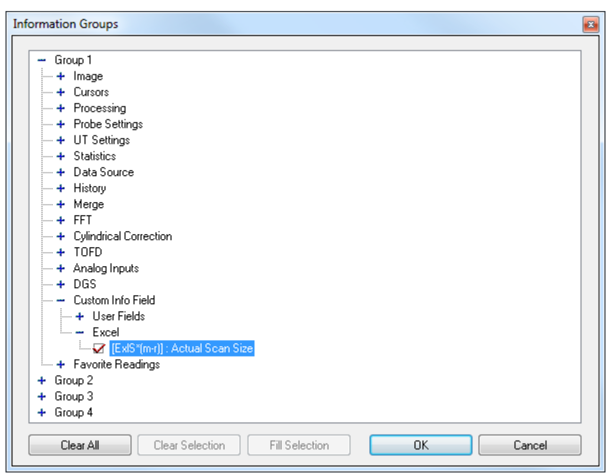


Figure 10-14 The New View Info dialog box

11. Back in the **Excel Exchanges** dialog box, in the **Import data from Excel file to custom info. field** list:
    a) Double-click the **Cell** column on the line of the added import field, and then enter the corresponding Excel worksheet cell [B8 for S*(m-r) in this example].
    b) Select the cell to import, and then click **Select View**.
12. In the **View Selector** dialog box, highlight the view in which you want the field to appear, and then click **Select**.
13. Repeat steps 9 and 12 when you have other imported information fields.
14. When your configuration is complete:
    - ◆ Close the **Excel Exchanges** dialog box to close it.

      OR

      Leave the **Excel Exchanges** dialog box open.

    In both cases, the dynamic data exchange remains active until you change the **Excel Exchanges** dialog box configuration. Similarly, you can close Excel or leave it open without affecting the dynamic data exchange. The Excel Exchange configuration data is saved in the .rst file.
15. Activate the Expert mode to see all available readings:
    a) On the menu, select **File > Preferences**.
    b) On the **General Settings** tab, under **Interface**, select the **Expert Mode** check box, and then click **OK**.
16. Add the reading imported from Excel to the information groups:
    a) Double-click the readings on the active view.
    b) In the **Information Groups** dialog box, expand **Group 3**, select **[ExlS*(m-r)]: Actual Scan Size**, and then click **OK** (see Figure 10-15 on page 459).

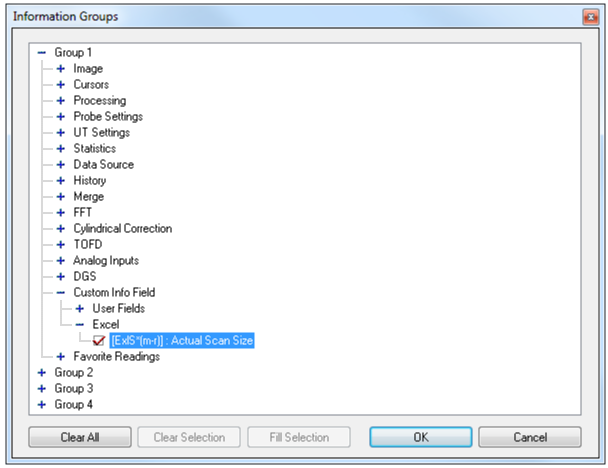


Figure 10-15 Selecting the reading imported from Excel

The imported reading appears in the Information Groups and its value is dynamically linked to the position of the cursors (see Figure 10-16 on page 460).

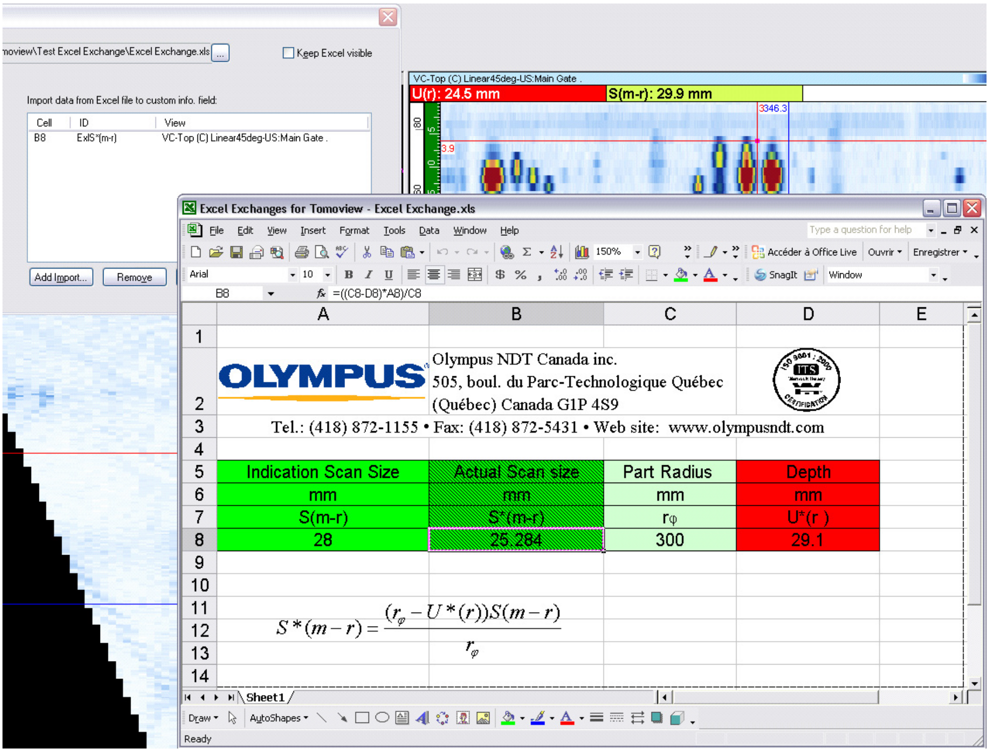


Figure 10-16 The dynamic data exchange between TomoView and Excel

10.5 Editing System Preferences

To edit system preferences

1. On the **File** menu, click **Preferences**.

   The **Preferences** dialog box appears (see Figure 10-17 on page 461).

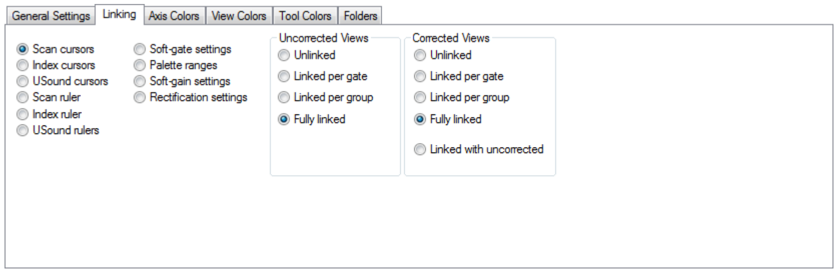


Figure 10-17 The Preferences dialog box

2. Display the category of preferences you want by clicking the corresponding tab.
3. Modify the options and properties as required.

   For details on each tab of the **System Preferences** dialog box, refer to the following subsections.
4. Click **OK** to save your modifications and close the dialog box.

   OR

   Click **Apply** to save your modifications without closing the dialog box, to continue modifying properties on another tab.

   OR

   Click **Cancel** to close the dialog box without saving your modifications.

10.6 Working with the Screen Capture Utility

TomoView features screen capture functions that are useful to quickly copy the full screen, windows, or a rectangular area. You can save the captured image to a file or copy it to the Clipboard to easily paste it in another application.

To capture a TomoView image with the Screen Capture utility

1. On the file menu, select **File > Save Screen as Bitmap**.
   The **Screen Capture** utility window appears.
2. In the **Screen Capture** window, perform the following tasks:

   a) To capture the whole computer screen as it currently appears, on the menu, select **Capture > Screen**.

   b) To only a specific TomoView window:

(1) On the menu, select **Capture > Window**.

(2) Move the capture pointer () over the window you wish to capture. An outline appears around the window below the pointer.

(3) Click the outlined window.

c) To capture a selected rectangle area on the screen:

(1) On the menu, select **Capture > Rectangle**.

(2) Click and drag from one corner of the rectangle area you wish to capture. A contour appears indicating the number of pixels of the selected rectangle area.

(3) Release the pointing device button at the opposite corner of the desired rectangle area.

The image of the selected image appears in the **Screen Capture** utility (see Figure 10-18 on page 462).

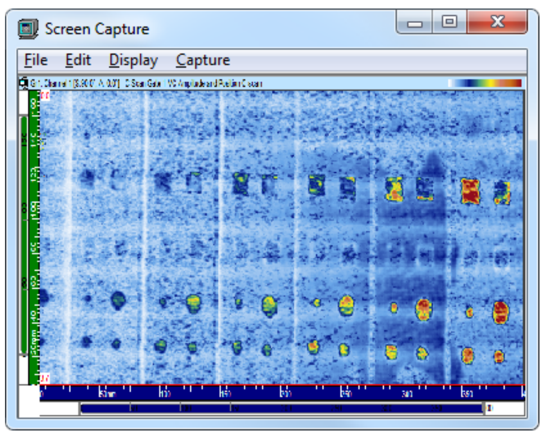


Figure 10-18 Example of a window capture with the Screen Capture utility

d) On the menu, select **Edit > Copy To Clipboard** to copy the captured image to the Clipboard.

e) On the menu, select **File > Save** to open the **Save Image** dialog box to save the image to a bitmap (.BMP) file of your choice.

To save the active view to an image file

NOTE

This function only works in Analysis mode.

1. Open a data file and select the desired layout.
2. Select the view with the content that you want to save in an image file.
3. On the menu, select **File > Save Active View as Image**.
4. In the **Save Image** dialog box that appears:
   a) Select a folder, a file name, and the file type (**BMP Files** or **TIFF Files**) to which you want to save the image.
   b) Click **Save**.

10.7 Working with the Beam Simulation Component

The **Beam Simulation** component is an ultrasonic beam simulator working on-line, in Setup mode with a .acq file, or off-line in Analysis mode while analyzing a specific .rdt file. Using this component, you can:

- Simulate the trajectories and dimensions of ultrasonic beams generated by conventional and phased array probes in a user-defined geometry.
- Evaluate the examination coverage provided by the ultrasonic beam configurations programmed in the current TomoView setup.
- Simultaneously simulate beams from multiple groups.
- See the beam spread and the rebounds on the defined part.

NOTE

The **Beam Simulation** component is not available in the **Lite Aero** and **Lite Weld** editions.

10.7.1 Enabling a Beam Simulation View

IMPORTANT

Use the **Beam Simulation** component with the following skew angles: 0°, 90°, 180°, and 270°. Attempts to visualize other skew angles generates invalid images.

To enable a beam simulation view

1. In Setup mode, open a setup file (.acq), or create a new setup.

   OR

   In Analysis mode, open a data file (.rdt).
2. On the menu, select **Part & Overlay > Part Definition**.
3. In the **Part Definition** dialog box, in the **Thickness** text box, enter the thickness of your part.

   The entered value is used to define both coordinates of the **Specimen properties** table (the **Depth** values). These points define the default border of the specimen.

   The default **Scan** (or **Index**) values are automatically defined by the projection of the ultrasonic beams with the considered skips and the **Scan Settings** (scan or index axis **Start** and **Stop** values).
4. Click the Beam Simulation button ( ).
5. In the **Beam Simulation** dialog box, select the **Enable simulation** check box.

10.7.2 Displaying a Beam Simulation View

To display a beam simulation view

1. In the **TomoView Manager** dialog box, select the layout you have reserved for the beam simulation (for example, layout 8), and then select the view in which you want to see the beam simulation.
2. On the menu, select **View > Contents**.
3. In the **Contents** dialog box, locate the **Beam Simulation** data group, and then double-click one of the available views.
   The corresponding beam simulation appears in the view (see example shown in Figure 10-19 on page 465).

The selection of a view depends on the skew angle used to define the ultrasonic beam.

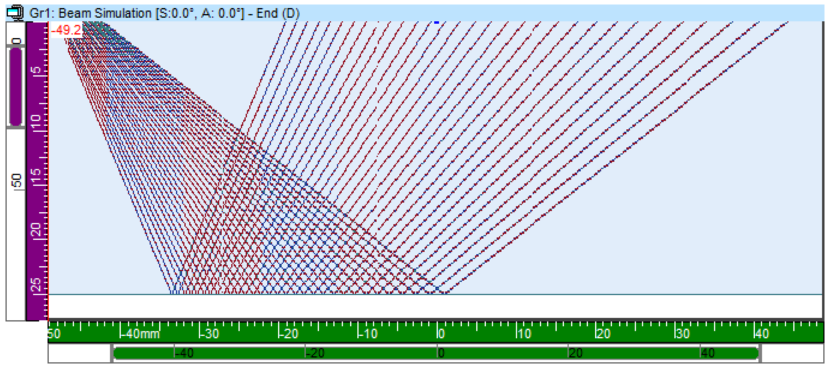


Figure 10-19 Beam simulation example

10.7.3 Description of the Beam Simulation Dialog Box

The **Beam Visualization** tab is used to display the parameters related to the visualization of the calculated beam (see Figure 10-20 on page 465).

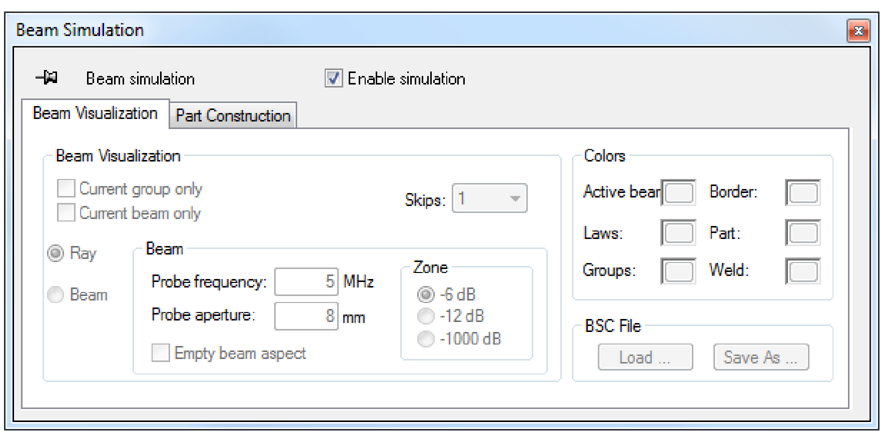


Figure 10-20 The Beam Visualization tab

The **Beam Visualization** tab contains three group boxes: **Beam visualization**, **Colors**, and **BSC file**.

Beam visualization group box

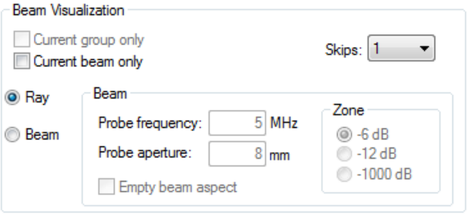


Figure 10-21 The Beam Visualization dialog box

The **Beam Visualization dialog box** contains the following. See Figure 10-21 on page 466.

Current group only

Select this check box to display only the focal laws of the current active group in the beam simulation view.

Current law only

Select this check box to display only the current active focal law in the beam simulation view.

Skips

This list box defines the number of half skips to be displayed in the beam simulation view.

Ray

Select this option button to display the ultrasonic beam as rays (one line per beam).

Beam

Select this option button to display the ultrasonic beam as a beam with its opening.

Beam group box

When the **Beam** option button is selected, the parameters in the **Beam** group box are enabled and are used to define the beam characteristics for the **Beam Simulation** calculations:

- **Probe frequency**: This text box sets the frequency of the probe.
- **Probe aperture**: This text box sets the aperture of the probe. For a phased array group, the aperture is set by the Advanced Calculator (considering the number of active elements and the pitch). In conventional groups, the aperture must be defined by the user.
- **Empty beam aspect**: Select this check box to display the ultrasonic beam with just two lines at the edge of the beam opening.
- **Zone**: Select one of the three option buttons to define the attenuation at which the beam opening edges (zone) is to be considered.

Colors group box

Figure 10-22 The Colors group box

Colors group box

The **Colors** group box can be used to change the color of the elements inside the beam simulation view. See Figure 10-22 on page 467.

When you click one of the colored squares, the standard **Color** dialog box that appears is used to select a different color for the associated item.

BSC file group box

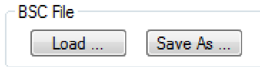


Figure 10-23 The BSC File group box

The **BSC** (beam simulation configuration) files are used to save the part construction parameters defined on the **Part Construction** tab. See Figure 10-23 on page 468.

Load

Opens a standard **Load** dialog box; used to load a .bsc file.

Save As

Opens a standard **Save As** dialog box; used to save the beam simulation configuration.

The **Part Construction** tab is used to display the parameters related to the part construction (see Figure 10-24 on page 468).

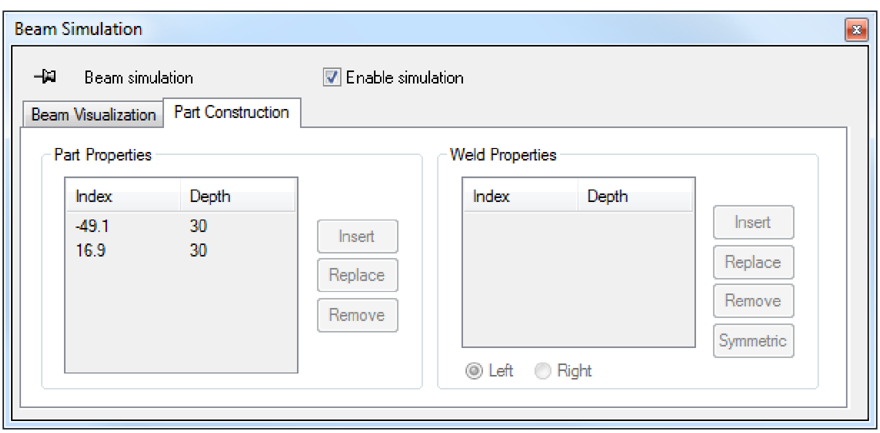


Figure 10-24 The Part Construction tab

The **Part Construction** tab contains two group boxes: **Part properties** and **Weld properties**.

Part properties group box

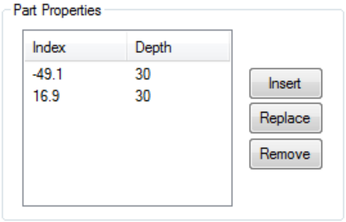


Figure 10-25 The Part Properties group box

The **Part Properties group box** contains the following. See Figure 10-25 on page 469.

Table
: This table is used to define the part border shape.

Scan (or Index)
: This column indicates the position of the points on the scan (or index) axis.

Depth
: This column indicates the position of the points on the ultrasonic axis.

Insert
: Click this button to insert a new point in the table defined by the position of the reference cross-hair cursor in the beam simulation view.

Replace
: Click this button to replace the selected point in the table by another one that has been defined by the position of the reference cross-hair cursor in the beam simulation view.

Remove
: Click this button to remove the selected point from the table. This action alters the part border shape.

Weld properties group box

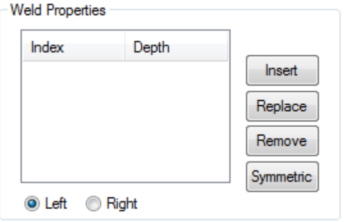


Figure 10-26 The Weld Properties group box

The **Weld Properties group box** contains the following. See Figure 10-26 on page 470.

Table
This table is used to define the part border shape.

Left
Select this option button to define the left side of the weld.

Right
Select this option button to define the right side of the weld.

Scan (or Index)
This column indicates the position of the points on the scan (or index) axis.

Depth
This column indicates the position of the points on the ultrasonic axis.

Insert
Click this button to insert a new point in the table that has been defined by the position of the reference cross-hair cursor in the beam simulation view.

Replace
Click this button to replace the selected point in the table by another one that has been defined by the position of the reference cross-hair cursor in the beam simulation view.

Remove
Click this button to remove the selected point from the table.

Symmetric

When one side of the weld preparation is defined (for example, left) and the other is selected (for example, right), click **Symmetric** to automatically define this side symmetrically to the other using the reference cross-hair cursor position as the weld centerline.

10.8 User Fields

This section describes how to define the customized information fields that can be added to the file content. These additional elements of information are called *user fields*. You can use the User fields to enter specific, user-defined information that is not predefined in TomoView (for example: name of the acquisition operator, name of the power plant, weld identification, etc.).

To define user fields

1. In the menu, select **Tools > Edit User Fields**.
2. In the **Page Name** dialog box, click in the text box and type the appropriate name for the first user field page, and then click **OK**.

   The **Edit User Fields** dialog box appears (see Figure 10-27 on page 472).

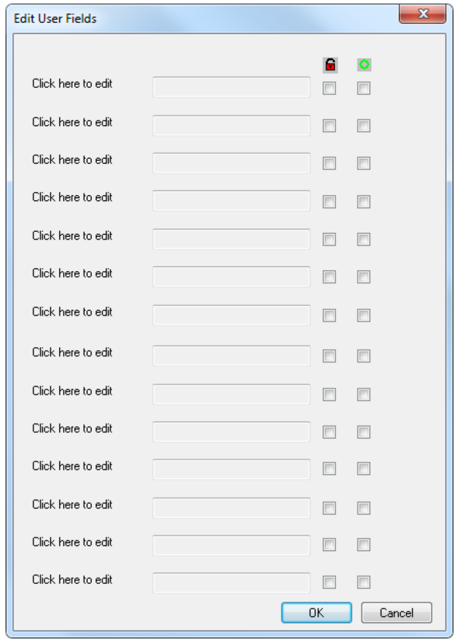


Figure 10-27 The Edit User Field dialog box

3. Using the mouse, select a **click here to edit** box label and enter the parameter name.
4. Click in the box to the right of the parameter name you just typed, and then type the value that corresponds to the parameter or leave the box empty to define the value later.
5. In the right column:
   - Select the  check box to protect the parameter against any modification in Analysis mode.
   - Select the  check box to make the parameter mandatory for every acquisition.
6. Click **Close**.

   The user field page is saved in the current setup.

10.9 Working with the Fast Fourier Transforms (FFT) Component

The **Fast Fourier Transform (FFT)** tool can be used to determine the spectral components of the signal received by an ultrasonic transducer. The **FFT** tool can be used at any time but will require data files acquired in the following conditions to be used in analysis mode. See Figure 10-28 on page 473.

To configure a setup for the FFT component

1. In the **Pulser/Receiver** tab of the **UT Settings** dialog bar, set **Rectification** to **RF**.
2. In the **Digitizer** tab of the **UT Settings** dialog bar, set **Compression** to **1**.
3. In the **TomoView Manager**, set the scan mode to **Free running.**
4. In the **Component** toolbar, click  (FFT).
5. In an A-scan view, place the Reference and Measurement cursors around the portion of the signal that you want to analyze.

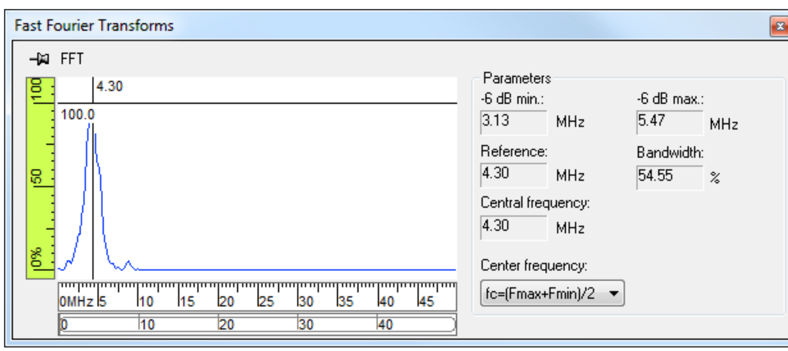


Figure 10-28 The Fast Fourier Transforms dialog box

List of Figures

Figure 1-1 TomoView Lite Weld edition display example 24
Figure 1-2 TomoView Lite Aero edition display example 25
Figure 1-3 The Startup Selection dialog box 26
Figure 1-4 The About TomoView dialog box indicating the TomoView edition 28
Figure 1-5 The HASP security hardware key models 28
Figure 1-6 The Startup Selection dialog box with no security hardware key 29
Figure 1-7 Missing hardware key message box 29
Figure 1-8 Configuring power options for Windows 7 30
Figure 1-9 Configuring power options for Windows XP 31
Figure 1-10 The Bootp Server Monitor icon and tooltip 32
Figure 1-11 The Bootp Server Monitor icon and contextual menu 33
Figure 1-12 Bootp status dialog box content example 33
Figure 1-13 The FOCUS LT front panel 35
Figure 1-14 The FOCUS LT Rackmount front panel 35
Figure 1-15 The Bootp Server icon when ready for operation 36
Figure 1-16 The network adaptor on the OmniScan MX 37
Figure 1-17 The OmniScan MX unit 38
Figure 1-18 Example of the OmniScan initial screen 38
Figure 1-19 The dialog box to type a new option key number 39
Figure 1-20 Example of the TomoView Control application screen 40
Figure 1-21 The Bootp Server icon when ready for operation 41
Figure 1-22 The Startup Selection dialog box 42
Figure 1-23 Example of the Select Device Configuration dialog box 43
Figure 1-24 The Configuration Selection dialog box 44
Figure 2-1 TomoView user interface example 46
Figure 2-2 The TomoView interface 48
Figure 2-3 The Menu bar 49
Figure 2-4 The File menu 50
Figure 2-5 The Save Custom Layout dialog box 51
Figure 2-6 Importing .law or .pac files dialog box 52

Figure 2-7 Data File Merge dialog box 53
Figure 2-8 Export Datagroup dialog box 53
Figure 2-9 The Toolbars menu 54
Figure 2-10 The Part & Overlay menu 55
Figure 2-11 The View menu 56
Figure 2-12 The Layout menu 58
Figure 2-13 The Save Current Layout submenu 59
Figure 2-14 The Processing menu 60
Figure 2-15 The Tools menu 62
Figure 2-16 The Components dialog box 64
Figure 2-17 The Window menu 66
Figure 2-18 The Help menu 66
Figure 2-19 The About TomoView dialog box 67
Figure 2-20 The Component toolbar 68
Figure 2-21 The Document toolbar 69
Figure 2-22 The View toolbars 70
Figure 2-23 Components of the TomoView Manager dialog box 74
Figure 2-24 Example of a document window containing three views 75
Figure 2-25 The set of ten layouts 76
Figure 2-26 The Status Bar 76
Figure 2-27 The UT Settings dialog box 79
Figure 2-28 The Scan and Mechanical Settings dialog box 80
Figure 2-29 The View Properties dialog box 80
Figure 2-30 Floating (top) and docking (bottom) of a dialog box 81
Figure 3-1 The main elements of the TomoView user interface 83
Figure 3-2 Switching between modes 85
Figure 3-3 The mode indicated in the status bar 85
Figure 3-4 The Group area in the TomoView Manager dialog box 86
Figure 3-5 Example of two scans from two different groups 87
Figure 3-6 Predefined scan sequences available from the TomoView Manager 88
Figure 3-7 The Scan tab of the Scan and Mechanical Settings dialog box for the Encoded - 2 axis predefined sequence 88
Figure 3-8 Example of a probe and a wedge illustration 90
Figure 3-9 Raster scan plate inspection with 0° probe skew 91
Figure 3-10 Plate inspection with 90° probe skew 92
Figure 3-11 Rivet inspection with 0° probe skew 92
Figure 3-12 Weld inspection using a scanner with 90° and 270° probe skews 93
Figure 3-13 Disc or wheel inspection with 0° and 180° probe skews 94
Figure 3-14 Tube weld inspection with 90° and 270° probe skews 95
Figure 3-15 The set of ten layouts 96
Figure 3-16 Selecting predefined layouts 96
Figure 3-17 Example of an active A-scan view 97

Figure 3-18 Example of the title bar of an active view 97
Figure 3-19 Example of data view types available for phased array data in the Contents window 99
Figure 3-20 Example of an Online A-Scan view 100
Figure 3-21 Example of Uncorrected (top-left), USound corrected (top-right) and VC sectorial (bottom-left) scans 101
Figure 3-22 Example of ultrasonic views [Top (C), Side (B), and End (D)] with probe skew angle of 90° 103
Figure 3-23 Example of Side (B) view 104
Figure 3-24 Example of Top (C) view 105
Figure 3-25 Example of End (D) view 106
Figure 3-26 Example of a Polar View 107
Figure 3-27 Example of Scrolling B-Scan view 108
Figure 3-28 Example of Scrolling View 109
Figure 3-29 Example of the contextual menu for a view 110
Figure 3-30 Examples of shown echo dynamics curves and rebound 112
Figure 3-31 Example of reading groups appearing at the top of a view 115
Figure 3-32 Example of gate A in an A-scan 116
Figure 3-33 Example of gate A in an S-scan 116
Figure 3-34 Gate colors 117
Figure 3-35 The UT Settings dialog box 118
Figure 3-36 The General tab 118
Figure 3-37 The Gain group box 119
Figure 3-38 The Time base group box 120
Figure 3-39 The Auto Values group box 121
Figure 3-40 The Auto Values dialog box 122
Figure 3-41 The Calibrate dialog box in True Depth mode 123
Figure 3-42 The Gates tab 124
Figure 3-43 Maximum gate detection 125
Figure 3-44 Crossing gate detection 126
Figure 3-45 First peak maximum gate detection 126
Figure 3-46 First peak maximum gate detection 127
Figure 3-47 The TCG tab 128
Figure 3-48 The DGS tab 129
Figure 3-49 The Digitizer tab 130
Figure 3-50 The Digitizing group box 130
Figure 3-51 The PRF group box 131
Figure 3-52 The Data Sample Size group box 131
Figure 3-53 The Data group box 132
Figure 3-54 Example of a compression ratio of 4 133
Figure 3-55 The Multipeak group box 133
Figure 3-56 The Pulser/Receiver tab 134

Figure 3-57 The Configuration group box 135
Figure 3-58 The Pulser group box 135
Figure 3-59 The Receiver group box 136
Figure 3-60 The Filters group box 136
Figure 3-61 The Probe tab 137
Figure 3-62 The Material and Interface group box 137
Figure 3-63 The Selection group box 138
Figure 3-64 The Position group box 139
Figure 3-65 The Adjust Scan data group resolution dialog box 140
Figure 3-66 The Beam orientation group box 140
Figure 3-67 Refracted angle 141
Figure 3-68 The Alarms tab 141
Figure 3-69 The I/O tab 142
Figure 3-70 The Analog inputs group box 143
Figure 3-71 The General outputs group box 144
Figure 3-72 The Digital out group box 144
Figure 3-73 The Transmitter tab 145
Figure 3-74 The Receiver tab 146
Figure 3-75 The Sequence tab for Bidirectional sequence 148
Figure 3-76 The Scan tab with Angular sequence type 149
Figure 3-77 The Scan tab with Helicoidal sequence type 150
Figure 3-78 The Scan Controls tab 152
Figure 3-79 The Inspection Parameters for Sequence dialog box 153
Figure 3-80 The File Size dialog box 153
Figure 3-81 The Encoders tab 155
Figure 3-82 The Calibration of Encoder dialog box 156
Figure 3-83 The MCDU dialog box 156
Figure 3-84 The Scan axis group box 157
Figure 3-85 The Options tab 159
Figure 3-86 The File Naming Options group box 159
Figure 3-87 The MCDU Custom File group box 160
Figure 3-88 The External File group box 161
Figure 3-89 Example of an External File in a Notepad window 161
Figure 3-90 The MCDU Control tab 162
Figure 3-91 The SCN File group box 162
Figure 3-92 The Advanced group box 163
Figure 3-93 The Motor Control group box 163
Figure 3-94 The Motor State group box 165
Figure 3-95 The Advanced MCDU Control dialog box for servomotors 166
Figure 3-96 The MCDU I/O tab 168
Figure 3-97 The MCDU State group box 169
Figure 3-98 The Last Error group box 169

Figure 3-99 The Motor 1 and Motor 2 group box 170
Figure 3-100 The Relays group box 171
Figure 3-101 The EEPROM group box 171
Figure 3-102 The View and Pane Properties dialog box 172
Figure 3-103 The Information tab 173
Figure 3-104 The Display tab for an A-scan view 174
Figure 3-105 The Cursors group box 174
Figure 3-106 Zooms and Rulers group box 175
Figure 3-107 The Grid group box 176
Figure 3-108 Polar View Image Processing group box 176
Figure 3-109 The Auto-Scroll Zoom group box 177
Figure 3-110 The Configuration group box 178
Figure 3-111 Group Gates group box 179
Figure 3-112 The Display tab for a Polar View 180
Figure 3-113 Part Definition dialog box 180
Figure 3-114 The Material Velocity group box 181
Figure 3-115 The Dimensions group box 181
Figure 3-116 The Probe Positioning group box 182
Figure 3-117 Contents dialog box with a Polar view 183
Figure 3-118 The Echo Dynamics tab 183
Figure 3-119 The Display group box 184
Figure 3-120 The Grid group box 184
Figure 3-121 The Curve Construction group box 185
Figure 3-122 The Peak Selection for Drop Sizing (-X dB) group box 186
Figure 3-123 The Overlay tab 186
Figure 3-124 The Show Selected Overlays group box 187
Figure 3-125 The Palette tab 187
Figure 3-126 The Color Palette dialog box 188
Figure 3-127 The Data Source tab 190
Figure 3-128 The Parameters tab 191
Figure 3-129 UT Settings group box 191
Figure 3-130 The Sound Velocity dialog box 192
Figure 3-131 The Delay dialog box 193
Figure 3-132 The Mechanical Settings group box 194
Figure 3-133 The Mechanical Settings dialog box 195
Figure 3-134 The Beam Orientation group box 196
Figure 3-135 The Refracted Angle dialog box 196
Figure 3-136 The Skew angle dialog box 197
Figure 3-137 The Part Definition dialog box 198
Figure 3-138 The Material Velocity group box 199
Figure 3-139 The Dimensions group box 199
Figure 3-140 The Probe Positioning group box 200

Figure 3-141 Contents dialog box with a Polar view 200
Figure 3-142 The Units tab 201
Figure 3-143 The Mechanical Axis Calibration dialog box 202
Figure 3-144 The Time / Half Path dialog box 203
Figure 3-145 The TOFD dialog box 204
Figure 3-146 The True Depth dialog box 206
Figure 3-147 The Translation / Rotation dialog box 207
Figure 3-148 The View Linking tab 208
Figure 3-149 The Rebounds tab 208
Figure 3-150 The Strip tab 209
Figure 3-151 The Preferences dialog box 211
Figure 3-152 The General Settings tab of the Preferences dialog box 211
Figure 3-153 Example of dialog box for an incompatible OmniScan firmware program 213
Figure 3-154 The Linking tab of the Preferences dialog box 215
Figure 3-155 The Axis Colors tab of the Preferences dialog box 216
Figure 3-156 The View Colors tab of the Preferences dialog box 217
Figure 3-157 The Tool Colors tab of the Preferences dialog box 217
Figure 3-158 The Folders tab of the Preferences dialog box 218
Figure 3-159 The Advanced Calculator dialog box 219
Figure 3-160 The categories of readings in the expert mode 220
Figure 3-161 The Select Device Configuration dialog box 221
Figure 3-162 TomoView file organization 223
Figure 4-1 The Open dialog box 228
Figure 4-2 The Open SRD Document dialog box 229
Figure 4-3 The Save As dialog box 230
Figure 4-4 Selecting the newly created group 231
Figure 4-5 The Group Creation Wizard dialog box 232
Figure 4-6 The Advanced Calculator dialog box 233
Figure 4-7 Selecting the probe 234
Figure 4-8 Specifying the inspected part geometry 234
Figure 4-9 Specifying the inspected part material 235
Figure 4-10 Specifying the wedge 235
Figure 4-11 Example of the Beam Display Info. tab in the Advanced Calculator 237
Figure 4-12 The Phased Array Parameters Source dialog box 239
Figure 4-13 The Group Creation Wizard dialog box 240
Figure 4-14 The Part Definition dialog box 241
Figure 4-15 Selecting the UT group in the TomoView Manager 241
Figure 4-16 The Group Creation Wizard dialog box 243
Figure 4-17 The message box for deleting a channel 244
Figure 4-18 Example of cursors around an indication in a Sector (S) view 245
Figure 4-19 Curves for the beam delay calibration 246

Figure 4-20 Red line appearing between tolerance lines after calibration 247
Figure 4-21 Sensitivity example after a first scan 248
Figure 4-22 Curves for the sensitivity calibration 249
Figure 4-23 The General tab of the UT Settings dialog box for a UT group 250
Figure 4-24 Selecting the probe in the UT Settings dialog box 250
Figure 4-25 Calibrating with the Time / Half Path dialog box 251
Figure 4-26 The Predefined Weld and Weld Parameters (for Type 1) dialog boxes 253
Figure 4-27 Example of a weld overlay with three flips 254
Figure 4-28 The TCG tab of the UT-Settings dialog box 255
Figure 4-29 The A-scan before adding the first TCG point. 256
Figure 4-30 The A-scan after adding the first TCG point 257
Figure 4-31 Example of an A-scan view showing the TCG curve 258
Figure 4-32 TCG curve construction example after a first scan 259
Figure 4-33 The TCG points import text 260
Figure 4-34 The set of ten layouts 261
Figure 4-35 The Template Layout menu on the Control bar 262
Figure 4-36 The list of available layouts 262
Figure 4-37 Example of the Contents dialog box 263
Figure 4-38 Default layout folder 264
Figure 4-39 The Save As dialog box 265
Figure 4-40 The Save Custom Layout dialog box 266
Figure 4-41 The Custom Layout saved templates 266
Figure 4-42 The DGS tab of the UT Settings dialog box 268
Figure 4-43 The IIW and DSC calibration blocks 270
Figure 4-44 Finding the maximum amplitude using the Envelope tool 270
Figure 4-45 Example of the DGS registration and warning level curves 271
Figure 4-46 An example of the Advanced Calculator 1-D Linear array tab configured for the DGS 273
Figure 4-47 The DGS readings in the Information Groups dialog box 275
Figure 4-48 The Favorite Readings categories in the Information Groups dialog box 277
Figure 4-49 Illustration of the A, D, T, and U readings 278
Figure 4-50 Corrosion readings example 279
Figure 4-51 Example of a reading tooltip 279
Figure 4-52 Example of an empty reading 279
Figure 4-53 The Tool Colors tab of the Preferences dialog box 280
Figure 4-54 The Color dialog box 281
Figure 4-55 The View Information Properties dialog box 282
Figure 4-56 The gate parameters in the TomoView Manager dialog box 284
Figure 4-57 The Gates tab of the UT Settings dialog box 284

Figure 4-58 S-scan gate positions automatically adjusted when the current beam changes when programmed in half-path mode 286
Figure 4-59 Example of a C-scan showing and not showing the data below the gate 287
Figure 4-60 Example of gate usage in a water immersed inspection 288
Figure 4-61 The Firing Sequencer (Interleave) dialog bar 290
Figure 4-62 The Default dialog button 290
Figure 4-63 Example of firing sequence with 2 Zones interlacing pairs 291
Figure 4-64 Example of firing sequence with 2 Z. All interlacing pairs 291
Figure 4-65 The Digitizer tab 292
Figure 4-66 The Alarms tab 292
Figure 4-67 Acquisition with full A-scan recording 293
Figure 4-68 Acquisition with conditional A-scan recording 293
Figure 5-1 Linear scan 296
Figure 5-2 The Sequence tab for the One-line scan sequence 296
Figure 5-3 The Sequence tab for Free Running sequence 297
Figure 5-4 Bidirectional surface scan 298
Figure 5-5 Reference system of the scanning mechanism 299
Figure 5-6 Unidirectional surface scan 301
Figure 5-7 The Sequence tab for Unidirectional sequence 302
Figure 5-8 The Sequence tab for Helicoidal sequence 303
Figure 5-9 Angular surface scan 305
Figure 5-10 The Sequence tab for Angular sequence 306
Figure 5-11 The Load custom program file dialog box 307
Figure 5-12 Selecting the encoder type on the Encoders tab 307
Figure 5-13 Quadrature and normal reading of the resolution 308
Figure 5-14 The Encoders tab of the Scan and Mechanical Settings dialog box for an Encoded - 2 axis scan type 310
Figure 5-15 The Calibration of Encoder dialog box for encoder 1 on the scan axis 311
Figure 5-16 Synchronization of setup and data files 312
Figure 5-17 The Position Dependent Setup dialog bar 312
Figure 5-18 The File Naming Options dialog box 315
Figure 6-1 The Open dialog box 318
Figure 6-2 The Open dialog box 320
Figure 6-3 The Use Layout list 320
Figure 6-4 The Save As dialog box 322
Figure 6-5 The Data File Merger dialog box 323
Figure 6-6 The indication table 324
Figure 6-7 Example of the indication table, the cursors, and the Zone tool used to document an indication 325
Figure 6-8 Selecting a predefined reading category in the indication table 326
Figure 6-9 Adding a comment and an additional reading to an indication 327

Figure 6-10 Example of the General Information dialog box 329
Figure 6-11 Example of a report header and user fields 329
Figure 6-12 Example of a report footer 330
Figure 7-1 The Create Thickness C-Scan dialog box 334
Figure 7-2 Selecting a thickness C-Scan view 334
Figure 7-3 Example of a thickness C-Scan 335
Figure 7-4 The Groups Selection group box 337
Figure 7-5 The New Group Information, Merge Type, and Generated Data group boxes 339
Figure 7-6 The Merge dimensions and Overall dimensions group boxes 340
Figure 7-7 The Data information and Volume information group boxes 341
Figure 7-8 Example of the volumetric merge group created 341
Figure 7-9 General Settings tab 343
Figure 7-10 Statistics parameter category of the Information Groups dialog box 346
Figure 7-11 Data display examples 347
Figure 7-12 Examples of flaw indication sizing with the zone tool 348
Figure 7-13 The Gain dialog bar 349
Figure 7-14 The TOFD Manager dialog box 351
Figure 7-15 The TOFD readings in the Information Groups dialog box 351
Figure 7-16 The TOFD Manager dialog box 352
Figure 7-17 TOFD dialog box 353
Figure 7-18 Selection of a reference A-scan 357
Figure 7-19 TOFD data after lateral wave synchronization 358
Figure 7-20 Selection of reference A-scan 360
Figure 7-21 TOFD data after lateral wave removal 361
Figure 7-22 Definition of the aperture 362
Figure 7-23 Overview of a SAFT ultrasonic image 363
Figure 7-24 Two files merge example 364
Figure 7-25 Gate A and Gate B C-scan merging example 365
Figure 7-26 Example of the C-Scan Merge dialog box (first step) 366
Figure 7-27 Example of the C-Scan Merge dialog box (second step) 367
Figure 7-28 Example of the C-scan merge group created 367
Figure 7-29 Selecting the noise reference area with the Zone tool 368
Figure 7-30 Example of reference area analysis results 369
Figure 7-31 Positioning the cursors around the defect area 370
Figure 7-32 Adjusting the K value 371
Figure 7-33 Example of the SNR analysis with K equal to 3.04 372
Figure 7-34 Example of the SNR analysis with K equal to 8.76 373
Figure 7-35 The SNR distribution 374
Figure 7-36 The Create Soft C-Scan dialog box 376
Figure 7-37 The Create Soft C-Scan with the Variable back wall activated 377
Figure 7-38 The B-Scan with a displayed variable back wall 377

Figure 7-39 The Contents dialog box 378
Figure 7-40 The Binarizer dialog box 379
Figure 7-41 The Threshold Type group box 379
Figure 7-42 The Binarizer cursors 380
Figure 7-43 The Reference Area dialog box 381
Figure 7-44 The Binarizer polarity 381
Figure 7-45 The Binarizer thresholds 382
Figure 7-46 The Binarizer Contents dialog box 382
Figure 7-47 The Binarized data display 383
Figure 7-48 The Palette Editor dialog box 384
Figure 7-49 The Color dialog box 385
Figure 7-50 Color palette optimized for corrosion visualization 386
Figure 7-51 Example of palette limits adjusted to specimen corrosion limits 387
Figure 7-52 Example of corrosion visualization on a demonstration specimen 387
Figure 7-53 Example of cursor and zone tools 391
Figure 7-54 The cursor parameters 394
Figure 7-55 The Statistics zone parameters 396
Figure 7-56 The 3-D Cursor (segment) parameters 398
Figure 7-57 Example before resynchronization 399
Figure 7-58 Example after resynchronization 399
Figure 7-59 The Matrix Filters Parameters dialog box 400
Figure 7-60 The Contents dialog box 401
Figure 7-61 The drop down menu for Set Volume Corrected Display Mode 401
Figure 7-62 The Cylindrical correction information group 402
Figure 7-63 The Cylindrical correction parameters 403
Figure 7-64 Selection of OD-related parameters 405
Figure 7-65 The Part Definition dialog box 406
Figure 7-66 Cylindrical correction: example 1 407
Figure 7-67 Selection of OD-related parameters 408
Figure 7-68 Cylindrical correction: example 2 409
Figure 7-69 Selection of ID-related parameters 410
Figure 7-70 The Part Definition dialog box 411
Figure 7-71 Cylindrical correction: example 3 412
Figure 7-72 Selection of ID-related parameters 413
Figure 7-73 Cylindrical correction: example 4 414
Figure 7-74 Reading of depth and index for a cylindrical part 415
Figure 7-75 Parameters for wedge calibration 417
Figure 7-76 Measurement of the Probe index offset 418
Figure 7-77 Example of the interface echo for a correctly calibrated wedge 419
Figure 7-78 Example of the interface echo for a wedge needing calibration 419
Figure 7-79 Analyzing cylindrical readings for a sectorial scan 421
Figure 7-80 Analyzing cylindrical readings for a linear scan 422

Figure 7-81 The Export Datagroup dialog box ... 423
Figure 7-82 Position of the cursors ... 425
Figure 7-83 The Hysteresis Correction dialog box ... 426
Figure 7-84 Example of a hysteresis correction ... 427
Figure 7-85 The Convert Log to Linear dialog box ... 428
Figure 7-86 The Contents dialog box with the Log-To-Linear data group ... 428
Figure 8-1 Example of shortcut keys available for menu commands ... 429
Figure 9-1 Example of the Bootp Status dialog box ... 437
Figure 9-2 The Bootp Server Configuration dialog box in error state ... 438
Figure 9-3 Opening the Network Adaptor Configuration dialog box ... 439
Figure 9-4 Configuring the network adaptor to connect to an acquisition unit ... 440
Figure 9-5 The Japanese language bar ... 440
Figure 9-6 Regional and Language Options ... 441
Figure 9-7 The Language tab ... 442
Figure 9-8 Setting English as default input ... 443
Figure 9-9 Ensure that English is the default input language ... 444
Figure 9-10 Starting TomoView ... 444
Figure 9-11 Switching the input language to Japanese from the language bar ... 444
Figure 10-1 The Open dialog box for opening a PASS file ... 446
Figure 10-2 Example of the Contents dialog box with a PASS file ... 446
Figure 10-3 Example of layout for a PASS file ... 447
Figure 10-4 Message box for saving imported PASS data ... 448
Figure 10-5 The Color dialog box ... 449
Figure 10-6 The Color dialog with full color palette ... 450
Figure 10-7 The View Linking tab of the Properties dialog box ... 451
Figure 10-8 The actual scan size of a defect ... 453
Figure 10-9 Example of an Excel worksheet content ... 454
Figure 10-10 Setting up the view and the readings ... 455
Figure 10-11 The Excel Exchanges dialog box ... 455
Figure 10-12 The Information Groups dialog box with available readings for the Excel Exchanges ... 456
Figure 10-13 The View Selector dialog box ... 457
Figure 10-14 The New View Info dialog box ... 457
Figure 10-15 Selecting the reading imported from Excel ... 459
Figure 10-16 The dynamic data exchange between TomoView and Excel ... 460
Figure 10-17 The Preferences dialog box ... 461
Figure 10-18 Example of a window capture with the Screen Capture utility ... 462
Figure 10-19 Beam simulation example ... 465
Figure 10-20 The Beam Visualization tab ... 465
Figure 10-21 The Beam Visualization dialog box ... 466
Figure 10-22 The Colors group box ... 467
Figure 10-23 The BSC File group box ... 468

Figure 10-24 The Part Construction tab 468
Figure 10-25 The Part Properties group box 469
Figure 10-26 The Weld Properties group box 470
Figure 10-27 The Edit User Field dialog box 472
Figure 10-28 The Fast Fourier Transforms dialog box 473

List of Tables

Table 1	Evident Data Acquisition Units compatible with TomoView	13
Table 2	Evident Data Acquisition Units compatible with TomoView	22
Table 3	TomoView edition feature comparison	26
Table 4	Bootp Server Monitor states	32
Table 5	The Component toolbar buttons	68
Table 6	The Document toolbar buttons	69
Table 7	The View toolbar buttons	71
Table 8	Alarm (Al.) indicator states	78
Table 9	Link indicator states	78
Table 10	Interpretation of OmniScan MX and FOCUS LT internal temperature	78
Table 11	File formats supported by TomoView	222
Table 12	Legacy file formats supported by TomoView	222
Table 13	Moving and resizing a gate using the mouse	283
Table 14	Parameters for flaw indication	344
Table 15	SNR parameters	373
Table 16	Summary of image analysis tools	388
Table 17	Shortcut keys	430